



# HEIDENHAIN



## TNC 620

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

NC Software  
817600-08  
817601-08  
817605-08

English (en)  
01/2021







Contents

1 Fundamentals.....19

2 Fundamentals / Overviews.....33

3 Using Touch Probe Cycles.....37

4 Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment..... 51

5 Touch Probe Cycles: Automatic Presetting..... 103

6 Touch Probe Cycles: Automatic Workpiece Inspection.....161

7 Touch Probe Cycles: Special Functions.....207

8 Touch Probe Cycles: Automatic Kinematics Measurement.....229

9 Touch Probe Cycles: Automatic Tool Measurement.....263

10 Cycles: Special Functions..... 287

11 Tables of Cycles..... 291







<b>1</b>	<b>Fundamentals.....</b>	<b>19</b>
<b>1.1</b>	<b>About this manual.....</b>	<b>20</b>
<b>1.2</b>	<b>Control model, software, and features.....</b>	<b>22</b>
	Software options.....	23
	New or modified cycle functions of software 81760x-08.....	28



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



<b>3</b>	<b>Using Touch Probe Cycles.....</b>	<b>37</b>
<b>3.1</b>	<b>General information about touch probe cycles.....</b>	<b>38</b>
	Method of function.....	38
	Consideration of a basic rotation in the Manual operation mode.....	38
	Touch probe cycles in the Manual Operation and Electronic Handwheel operating modes.....	38
	Touch probe cycles for automatic operation.....	39
<b>3.2</b>	<b>Before You Start Working with Touch Probe Cycles.....</b>	<b>41</b>
	Maximum traverse to touch point: DIST in touch probe table.....	41
	Set-up clearance to touch point: SET_UP in touch probe table.....	41
	Orient the infrared touch probe to the programmed probe direction: TRACK in touch probe table.....	41
	Touch trigger probe, probing feed rate: F in touch probe table.....	42
	Touch trigger probe, rapid traverse for positioning: FMAX.....	42
	Touch trigger probe, rapid traverse for positioning: F_PREPOS in touch probe table.....	42
	Executing touch probe cycles.....	43
<b>3.3</b>	<b>Program defaults for cycles.....</b>	<b>45</b>
	Overview.....	45
	Entering GLOBAL DEF definitions.....	46
	Using GLOBAL DEF information.....	46
	Global data valid everywhere.....	47
	Global data for probing functions.....	47
<b>3.4</b>	<b>Touch-probe table.....</b>	<b>48</b>
	General information.....	48
	Editing touch probe tables.....	48
	Touch probe data.....	49



<b>4</b>	<b>Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment.....</b>	<b>51</b>
<b>4.1</b>	<b>Overview.....</b>	<b>52</b>
<b>4.2</b>	<b>Touch probe cycles 14xx: fundamentals.....</b>	<b>53</b>
	Functionalities common to touch probe cycles 14xx for measuring rotations.....	53
	Semi-automatic mode.....	55
	Evaluation of tolerances.....	60
	Transferring the actual position.....	61
<b>4.3</b>	<b>PROBING IN PLANE (Cycle 1420, DIN/ISO: G1420, option 17).....</b>	<b>62</b>
	Application.....	62
	Please note while programming!.....	63
	Cycle parameters.....	64
<b>4.4</b>	<b>PROBING ON EDGE (Cycle 1410, DIN/ISO: G1410, option 17).....</b>	<b>67</b>
	Application.....	67
	Please note while programming!.....	69
	Cycle parameters.....	70
<b>4.5</b>	<b>PROBING TWO CIRCLES (Cycle 1411, DIN/ISO: G1411, option 17).....</b>	<b>73</b>
	Application.....	73
	Please note while programming!.....	75
	Cycle parameters.....	76
<b>4.6</b>	<b>Touch probe cycles 4xx: fundamentals.....</b>	<b>79</b>
	Characteristics common to all touch probe cycles for measuring workpiece misalignment.....	79
<b>4.7</b>	<b>BASIC ROTATION (Cycle 400, DIN/ISO: G400, option 17).....</b>	<b>80</b>
	Application.....	80
	Please note while programming:.....	80
	Cycle parameters.....	81
<b>4.8</b>	<b>ROT. OF 2 HOLES (Cycle 401, DIN/ISO: G401, option 17).....</b>	<b>83</b>
	Application.....	83
	Please note while programming:.....	83
	Cycle parameters.....	84
<b>4.9</b>	<b>ROT. OF 2 STUDS (Cycle 402, DIN/ISO: G402, option 17).....</b>	<b>87</b>
	Application.....	87
	Please note while programming:.....	88
	Cycle parameters.....	89
<b>4.10</b>	<b>ROT. IN ROTARY AXIS (Cycle 403, DIN/ISO: G403, option 17).....</b>	<b>92</b>
	Application.....	92
	Please note while programming:.....	93
	Cycle parameters.....	94



<b>4.11</b>	<b>ROT. IN C AXIS (Cycle 405, DIN/ISO: G405, option 17)</b>	<b>97</b>
	Application	97
	Please note while programming:	98
	Cycle parameters	99
<b>4.12</b>	<b>SET BASIC ROTATION (Cycle 404, DIN/ISO: G404, option 17)</b>	<b>101</b>
	Application	101
	Cycle parameters	101
<b>4.13</b>	<b>Example: Determining a basic rotation from two holes</b>	<b>102</b>



<b>5</b>	<b>Touch Probe Cycles: Automatic Presetting.....</b>	<b>103</b>
<b>5.1</b>	<b>Fundamentals.....</b>	<b>104</b>
	Overview.....	104
	Characteristics common to all touch probe cycles for presetting.....	106
<b>5.2</b>	<b>PRESET INSIDE RECTANGLE (Cycle 410, DIN/ISO: G410, option 17).....</b>	<b>107</b>
	Application.....	107
	Please note while programming:.....	108
	Cycle parameters.....	109
<b>5.3</b>	<b>PRESET OUTS. RECTAN (Cycle 411, DIN/ISO: G411, option 17).....</b>	<b>111</b>
	Application.....	111
	Please note while programming:.....	112
	Cycle parameters.....	113
<b>5.4</b>	<b>PRESET INSIDE CIRCLE (Cycle 412, DIN/ISO: G412, option 17).....</b>	<b>115</b>
	Application.....	115
	Please note while programming:.....	116
	Cycle parameters.....	117
<b>5.5</b>	<b>PRESET OUTS. CIRCLE (Cycle 413, DIN/ISO: G413, option 17).....</b>	<b>120</b>
	Application.....	120
	Please note while programming:.....	121
	Cycle parameters.....	122
<b>5.6</b>	<b>PRESET OUTSIDE CORNER (Cycle 414, DIN/ISO: G414, option 17).....</b>	<b>125</b>
	Application.....	125
	Please note while programming:.....	126
	Cycle parameters.....	127
<b>5.7</b>	<b>PRESET INSIDE CORNER (Cycle 415, DIN/ISO: G415, option 17).....</b>	<b>130</b>
	Application.....	130
	Please note while programming:.....	131
	Cycle parameters.....	132
<b>5.8</b>	<b>PRESET CIRCLE CENTER (Cycle 416, DIN/ISO: G416, option 17).....</b>	<b>135</b>
	Application.....	135
	Please note while programming:.....	136
	Cycle parameters.....	137
<b>5.9</b>	<b>PRESET IN TS AXIS (Cycle 417, DIN/ISO: G417, option 17).....</b>	<b>140</b>
	Application.....	140
	Please note while programming:.....	140
	Cycle parameters.....	141



<b>5.10</b>	<b>PRESET FROM 4 HOLES (Cycle 418, DIN/ISO: G418, option 17)</b>	<b>142</b>
	Application	142
	Please note while programming:	143
	Cycle parameters	144
<b>5.11</b>	<b>PRESET IN ONE AXIS (Cycle 419, DIN/ISO: G419, option 17)</b>	<b>147</b>
	Application	147
	Please note while programming:	147
	Cycle parameters	148
<b>5.12</b>	<b>SLOT CENTER PRESET (Cycle 408, DIN/ISO: G408, option 17)</b>	<b>150</b>
	Application	150
	Please note while programming:	151
	Cycle parameters	152
<b>5.13</b>	<b>RIDGE CENTER PRESET (Cycle 409, DIN/ISO: G409, option 17)</b>	<b>154</b>
	Application	154
	Please note while programming:	155
	Cycle parameters	156
<b>5.14</b>	<b>Example: Presetting in center of a circular segment and on top surface of workpiece</b>	<b>158</b>
<b>5.15</b>	<b>Example: Presetting on top surface of workpiece and in center of a bolt hole circle</b>	<b>159</b>



<b>6</b>	<b>Touch Probe Cycles: Automatic Workpiece Inspection.....</b>	<b>161</b>
<b>6.1</b>	<b>Fundamentals.....</b>	<b>162</b>
	Overview.....	162
	Recording the results of measurement.....	163
	Measurement results in Q parameters.....	165
	Classification of results.....	165
	Tolerance monitoring.....	165
	Tool monitoring.....	166
	Reference system for measurement results.....	167
<b>6.2</b>	<b>REFERENCE PLANE (Cycle 0, DIN/ISO: G55, option 17).....</b>	<b>168</b>
	Application.....	168
	Please note while programming:.....	168
	Cycle parameters.....	168
<b>6.3</b>	<b>POLAR PRESET (Cycle 1, option 17).....</b>	<b>169</b>
	Application.....	169
	Please note while programming:.....	169
	Cycle parameters.....	169
<b>6.4</b>	<b>MEASURE ANGLE (Cycle 420, DIN/ISO: G420, option 17).....</b>	<b>170</b>
	Application.....	170
	Please note while programming:.....	170
	Cycle parameters.....	171
<b>6.5</b>	<b>MEASURE HOLE (Circle 421, DIN/ISO: G421, option 17).....</b>	<b>173</b>
	Application.....	173
	Please note while programming:.....	173
	Cycle parameters.....	174
<b>6.6</b>	<b>MEASURE CIRCLE OUTSIDE (Cycle 422, DIN/ISO: G422, option 17).....</b>	<b>177</b>
	Application.....	177
	Please note while programming:.....	177
	Cycle parameters.....	178
<b>6.7</b>	<b>MEASURE RECTANGLE INSIDE (Cycle 423, DIN/ISO: G423, option 17).....</b>	<b>181</b>
	Application.....	181
	Please note while programming:.....	182
	Cycle parameters.....	183
<b>6.8</b>	<b>MEASURE RECTANGLE OUTSIDE (Cycle 424, DIN/ISO: G424, option 17).....</b>	<b>185</b>
	Application.....	185
	Please note while programming:.....	185
	Cycle parameters.....	186



<b>6.9</b>	<b>MEASURE INSIDE WIDTH (Cycle 425, DIN/ISO: G425, option 17)</b>	<b>188</b>
	Application	188
	Please note while programming	188
	Cycle parameters	189
<b>6.10</b>	<b>MEASURE RIDGE WIDTH (Cycle 426, DIN/ISO: G426, option 17)</b>	<b>191</b>
	Application	191
	Please note while programming	191
	Cycle parameters	192
<b>6.11</b>	<b>MEASURE COORDINATE (Cycle 427, DIN/ISO: G427, option 17)</b>	<b>194</b>
	Application	194
	Please note while programming	194
	Cycle parameters	195
<b>6.12</b>	<b>MEAS. BOLT HOLE (Cycle 430, DIN/ISO: G430, option 17)</b>	<b>197</b>
	Application	197
	Please note while programming	197
	Cycle parameters	198
<b>6.13</b>	<b>MEASURE PLANE (Cycle 431, DIN/ISO: G431, option 17)</b>	<b>200</b>
	Application	200
	Please note while programming	201
	Cycle parameters	201
<b>6.14</b>	<b>Programming Examples</b>	<b>203</b>
	Example: Measuring and reworking a rectangular stud	203
	Example: Measuring a rectangular pocket and recording the results	205



<b>7</b>	<b>Touch Probe Cycles: Special Functions.....</b>	<b>207</b>
<b>7.1</b>	<b>Fundamentals.....</b>	<b>208</b>
	Overview.....	208
<b>7.2</b>	<b>MEASURING (Cycle 3, option 17).....</b>	<b>209</b>
	Application.....	209
	Please note while programming:.....	209
	Cycle parameters.....	210
<b>7.3</b>	<b>MEASURING IN 3-D (Cycle 4, option 17).....</b>	<b>211</b>
	Application.....	211
	Please note while programming:.....	211
	Cycle parameters.....	212
<b>7.4</b>	<b>FAST PROBING (Cycle 441, DIN/ISO: G441, option 17).....</b>	<b>213</b>
	Application.....	213
	Please note while programming:.....	213
	Cycle parameters.....	214
<b>7.5</b>	<b>Calibrating a touch trigger probe.....</b>	<b>215</b>
<b>7.6</b>	<b>Displaying calibration values.....</b>	<b>216</b>
<b>7.7</b>	<b>TS CALIBRATION OF TOOL LENGTH (Cycle 461, DIN/ISO: G461, option 17).....</b>	<b>217</b>
<b>7.8</b>	<b>CALIBRATION OF A TS IN A RING (Cycle 462, DIN/ISO: G462, option 17).....</b>	<b>219</b>
<b>7.9</b>	<b>TS CALIBRATION ON STUD (Cycle 463, DIN/ISO: G463, option 17).....</b>	<b>222</b>
<b>7.10</b>	<b>CALIBRATION OF TS ON A SPHERE (Cycle 460, DIN/ISO: G460, option 17).....</b>	<b>225</b>



<b>8</b>	<b>Touch Probe Cycles: Automatic Kinematics Measurement.....</b>	<b>229</b>
<b>8.1</b>	<b>Kinematics measurement with TS touch probes (option 48).....</b>	<b>230</b>
	Fundamentals.....	230
	Overview.....	231
<b>8.2</b>	<b>Prerequisites.....</b>	<b>232</b>
	Please note while programming:.....	233
<b>8.3</b>	<b>SAVE KINEMATICS (Cycle 450, DIN/ISO: G450, option 48).....</b>	<b>234</b>
	Application.....	234
	Please note while programming:.....	234
	Cycle parameters.....	235
	Log function.....	235
	Notes on data management.....	236
<b>8.4</b>	<b>MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option 48).....</b>	<b>237</b>
	Application.....	237
	Positioning direction.....	239
	Machines with Hirth-coupled axes.....	240
	Example calculation of measuring positions for an A axis:.....	240
	Choice of number of measuring points.....	241
	Choice of the calibration sphere position on the machine table.....	242
	Notes on the accuracy.....	242
	Notes on various calibration methods.....	243
	Backlash.....	244
	Please note while programming:.....	245
	Cycle parameters.....	246
	Various modes (Q406).....	249
	Log function.....	250
<b>8.5</b>	<b>PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option 48).....</b>	<b>251</b>
	Application.....	251
	Please note while programming:.....	253
	Cycle parameters.....	254
	Adjustment of interchangeable heads.....	257
	Drift compensation.....	259
	Log function.....	261



<b>9</b>	<b>Touch Probe Cycles: Automatic Tool Measurement.....</b>	<b>263</b>
<b>9.1</b>	<b>Fundamentals.....</b>	<b>264</b>
	Overview.....	264
	Differences between Cycles 30 to 33 and Cycles 480 to 483.....	265
	Setting machine parameters.....	266
	Entries in the tool table for milling tools.....	268
<b>9.2</b>	<b>CALIBRATE TT (Cycle 30 or 480, ISO: G480, option 17).....</b>	<b>269</b>
	Application.....	269
	Please note while programming:.....	270
	Cycle parameters.....	271
<b>9.3</b>	<b>Measuring tool length (Cycle 31 or 481, DIN/ISO: G481, option 17).....</b>	<b>272</b>
	Application.....	272
	Please note while programming:.....	273
	Cycle parameters.....	274
<b>9.4</b>	<b>Measuring a tool radius (Cycle 32 or 482, ISO: G482, option 17).....</b>	<b>276</b>
	Application.....	276
	Please note while programming:.....	277
	Cycle parameters.....	278
<b>9.5</b>	<b>Measuring tool length and radius (Cycle 33 or 483, ISO: G483, option 17).....</b>	<b>280</b>
	Application.....	280
	Please note while programming:.....	281
	Cycle parameters.....	282
<b>9.6</b>	<b>CALIBRATE IR TT (Cycle 484, ISO: G484, option 17).....</b>	<b>284</b>
	Application.....	284
	Cycle run.....	284
	Please note while programming:.....	286
	Cycle parameters.....	286



<b>10</b>	<b>Cycles: Special Functions.....</b>	<b>287</b>
<b>10.1</b>	<b>Fundamentals.....</b>	<b>288</b>
	Overview.....	288
<b>10.2</b>	<b>SPINDLE ORIENTATION (Cycle 13, ISO: G36).....</b>	<b>289</b>
	Application.....	289
	Please note while programming:.....	289
	Cycle parameters.....	289



<b>11</b>	<b>Tables of Cycles.....</b>	<b>291</b>
<b>11.1</b>	<b>Table of cycles.....</b>	<b>292</b>
	Touch probe cycles.....	292



# 1

## Fundamentals



## 1.1 About this manual

### Safety precautions

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

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

#### **DANGER**

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

#### **WARNING**

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

#### **CAUTION**

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

#### **NOTICE**

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

### Sequence of information in precautionary statements

All precautionary statements comprise the following four sections:

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



### Informational notes

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

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



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



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



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

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

We continuously strive to improve our documentation for you.

Please help us by sending your suggestions to the following e-mail address:

**tnc-userdoc@heidenhain.de**



## 1.2 Control model, software, and features

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

Control model	NC software number
TNC 620	817600-08
TNC 620 E	817601-08
TNC 620 Programming station	817605-08

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

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

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

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

- Tool measurement with the TT

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

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



### Operating instructions:

All cycle functions not connected to the measuring cycles 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: 1303427-xx



### User's Manual:

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

Conversational Programming User's Manual ID: 1096883-xx

ISO Programming User's Manual ID: 1096887-xx

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



## Software options

The TNC 620 features various software options, each of which can be enabled separately by your machine tool builder. 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>	<b>Machining with rotary tables</b> <ul style="list-style-type: none"> <li>■ Cylindrical contours as if in two axes</li> <li>■ Feed rate in distance per minute</li> </ul> <b>Coordinate conversions:</b> Tilting the working plane
-----------------------------------	--

---

### Advanced Function Set 2 (option 9)

---

<b>Advanced functions (set 2)</b> Export license required	<b>3-D machining:</b> <ul style="list-style-type: none"> <li>■ 3-D tool compensation through surface-normal vectors</li> <li>■ Using the electronic handwheel to change the angle of the swivel head during program run; the position of the tool point remains unchanged (TCPM = Tool Center Point Management)</li> <li>■ Keeping the tool normal to the contour</li> <li>■ Tool radius compensation normal to the tool direction</li> <li>■ Manual traverse in the active tool-axis system</li> </ul> <b>Interpolation:</b> Linear in > 4 axes (export license required)
--	---

---

### Touch Probe Functions (option 17)

---

<b>Touch probe functions</b>	<b>Touch probe cycles:</b> <ul style="list-style-type: none"> <li>■ Compensation of tool misalignment in automatic mode</li> <li>■ Set the preset in the <b>Manual operation</b> mode of operation</li> <li>■ Presetting in automatic mode</li> <li>■ Automatically measuring workpieces</li> <li>■ Tools can be measured automatically</li> </ul>
------------------------------	--

---

### HEIDENHAIN DNC (option 18)

---

Communication with external PC applications over COM component

---

### Advanced Programming Features (option 19)

---

<b>Expanded programming functions</b>	<b>FK free contour programming:</b> Programming in HEIDENHAIN conversational format with graphic support for workpiece drawings not dimensioned for NC
---------------------------------------	---

---



---

**Advanced Programming Features (option 19)**


---

**Fixed cycles:**

- Peck drilling, reaming, boring, counterboring, centering
  - Milling internal and external threads
  - Milling of rectangular and circular pockets and studs
  - Clearing level and oblique surfaces
  - Milling of straight and circular slots
  - Circular and linear point patterns
  - Contour train, contour pocket, trochoidal contour slot
  - Engraving
  - OEM cycles (special cycles developed by the machine tool builder) can be integrated
- 

**Advanced Graphic Features (option 20)**


---

**Expanded graphic functions****Program-verification graphics, program-run graphics**

- Plan view
  - Projection in three planes
  - 3-D view
- 

**Advanced Function Set 3 (option 21)**


---

**Expanded functions Group 3****Tool compensation:**

M120: Radius-compensated contour look ahead for up to 99 NC blocks (LOOK AHEAD)

**3-D machining:**

M118: Superimpose handwheel positioning during program run

---

**Pallet Management (option 22)**


---

**Pallet management**

Processing workpieces in any sequence

---

**CAD Import (option 42)**


---

**CAD import**

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

**KinematicsOpt (option 48)**


---

**Optimizing the machine kinematics**

- Backup/restore active kinematics
  - Test active kinematics
  - Optimize active kinematics
- 

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


---

**Standardized interface**

The OPC UA NC Server provides a standardized interface (OPC UA) for external access to the control's data and functions

These software options allow you to create up to six parallel client connections



**Extended Tool Management (option 93)**

<b>Extended tool management</b>	Python-based
---------------------------------	--------------

**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>
--	--

**State Reporting Interface – SRI (option 137)**

<b>HTTP accesses to the control status</b>	<ul style="list-style-type: none"> <li>■ Reading out the times of status changes</li> <li>■ Reading out the active NC programs</li> </ul>
--	---

**Cross Talk Compensation – CTC (option 141)**

<b>Compensation of axis couplings</b>	<ul style="list-style-type: none"> <li>■ Determination of dynamically caused position deviation through axis acceleration</li> <li>■ Compensation of the TCP (<b>T</b>ool <b>C</b>enter <b>P</b>oint)</li> </ul>
---------------------------------------	--

**Position Adaptive Control – PAC (option 142)**

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

**Load Adaptive Control – LAC (option 143)**

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

**Active Chatter Control – ACC (option 145)**

<b>Active chatter control</b>	Fully automatic function for chatter control during machining
-------------------------------	---

**Machine Vibration Control – MVC (option 146)**

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

**Batch Process Manager (option 154)**

<b>Batch process manager</b>	Planning of production orders
------------------------------	-------------------------------

**Component Monitoring (option 155)**

<b>Component monitoring without external sensors</b>	Monitoring configured machine components for overload
--	---

**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 tool builder. This includes functional safety (FS), for example.

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

ID: 827222-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 tool builder or HEIDENHAIN.

### Intended place of operation

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

### Legal information

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

Further information is available on the control as follows:

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

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

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



## Optional parameters

The comprehensive cycle package is continuously further developed by HEIDENHAIN. Every new software version thus may also introduce new Q parameters for cycles. These new Q parameters are optional parameters, which were not all available in some older software versions. Within a cycle, they are always provided at the end of the cycle definition. The section "New or modified cycle functions of software 81760x-08" 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 (as of TNC 150 B) can be run with the new software version of the TNC 620. Even if new optional parameters ("Optional parameters") have been added to existing cycles, you will generally be able to run your NC programs as usual. This is achieved because the stored default value will be used. The other way round, if you want to run an NC program created with a new software version on an older control, you can delete the respective optional Q parameters from the cycle definition with the NO ENT key. In this way you can ensure that the NC program is backward compatible. If NC blocks contain invalid elements, the control will mark them as ERROR blocks when the file is opened.



## New or modified cycle functions of software 81760x-08



### 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: 1322094-xx

### User's Manual for Programming of Machining Cycles:

#### New functions:

- 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 **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 forms.
- Cycle **1282 OCM CIRCLE BOUNDARY** (ISO: **G1282**, option 167)  
This cycle allows you to define a circular boundary for islands or open pockets that you previously programmed with the standard OCM forms.
- The control offers an **OCM cutting data calculator** with which you can determine the optimum cutting data for the **272 OCM ROUGHING** cycle (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



**Modified functions:**

- Cycle **225 ENGRAVING** (ISO: **G225**) allows you to use system variables to engrave the number of the current week.
- Cycles **202 BORING** (ISO: **G202**) and **204 BACK BORING** (ISO: **G204**, option 19) restore the spindle status after machining to that which was active before the cycle.
- The threads of cycles **206 TAPPING** (ISO: **G206**), **207 RIGID TAPPING** (ISO: **G207**), **209 TAPPING W/ CHIP BRKG** (ISO: **G209**, option 19), and **18 THREAD CUTTING** (ISO: **G18**) are shown shaded in Test Run mode.
- 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**, option 19)
  - Cycle 23 **FLOOR FINISHING** (ISO: **G123**, option 19)
  - Cycle 24 **SIDE FINISHING** (ISO: **G124**, option 19)
  - Cycle 233 **FACE MILLING** (ISO: **G233**, option 19)
  - 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)
- Cycles **251 RECTANGULAR POCKET** (ISO: **G251**), **252 CIRCULAR POCKET** (ISO: **G252**, option 19), and **272 OCM ROUGHING** (ISO: **G272**, option 167) take into consideration a cutting width defined in the **RCUTS** column when calculating the plunging path.
- Cycles **208 BORE MILLING** (ISO: **G208**), **253 SLOT MILLING** (ISO: **G208**), and **254 CIRCULAR SLOT** (ISO: **G254**, option 19) monitor a cutting width defined in the **RCUTS** column 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.
- The machine manufacturer can hide Cycle **238 MEASURE MACHINE STATUS** (ISO: **G238**, option 155).
- An input value of 2 has been added to parameter **Q569 OPEN BOUNDARY** in Cycle **271 OCM CONTOUR DATA** (ISO: **G271**, option 167). The control uses this value to interpret the first contour within the **CONTOUR DEF** function as a boundary block of a pocket.
- Cycle **272 OCM ROUGHING** (ISO: **G272**, option 167) was expanded:
  - The parameter **Q576 SPINDLE SPEED** allows you to define a spindle speed for the roughing tool.
  - The parameter **Q579 PLUNGING FACTOR S** allows you to define a factor for the spindle speed during plunging.
  - Use the parameter **Q575 INFEEED STRATEGY** to define whether the control machines the contour from top to bottom or vice versa.



- The maximum input range of parameter **Q370 TOOL PATH OVERLAP** has been changed. Old: from 0.01 to 1. New: from 0.04 to 1.99.
- If helical plunging is not possible, the control attempts a reciprocating plunging motion with the tool.
- Cycle **273 OCM FINISHING FLOOR** (ISO: **G273**, option 167) was expanded.

The following parameters have been added:

- **Q595 STRATEGY**: machining with equal path distances or constant tool angle
- **Q577 APPROACH RADIUS FACTOR**: factor for the tool radius in order to adapt the approach radius

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

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

#### Changed Functions

- Cycles **480 CALIBRATE TT** (ISO: **G480**) and **484 CALIBRATE IR TT** (ISO: **G484**, option 17) can be used to calibrate a tool touch probe with cuboid probe contacts.  
**Further information:** "CALIBRATE TT (Cycle 30 or 480, ISO: G480, option 17)", Page 269  
**Further information:** "CALIBRATE IR TT (Cycle 484, ISO: G484, option 17)", Page 284
- For rotating tools, Cycle **483 MEASURE TOOL** (ISO: **G483**, option 17) first measures the tool length and then the tool radius.  
**Further information:** "Measuring tool length and radius (Cycle 33 or 483, ISO: G483, option 17)", Page 280
- By default, Cycles **1410 PROBING ON EDGE** (ISO: **G1410**) and **1411 PROBING TWO CIRCLES** (ISO: **G1411**, option 17) 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:** "PROBING ON EDGE (Cycle 1410, DIN/ISO: G1410, option 17)", Page 67  
**Further information:** "PROBING TWO CIRCLES (Cycle 1411, DIN/ISO: G1411, option 17)", Page 73







# 2

**Fundamentals /  
Overviews**



## 2.1 Introduction

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

### NOTICE

#### Danger of collision!

Cycles execute extensive operations. Danger of collision!

- You should run a program test before machining



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

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

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

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



## 2.2 Available cycle groups

### Overview of fixed cycles



- 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, determining the load,	<b>Further information:</b> User's Manual for Programming of Machining Cycles
	► If required, switch to machine-specific machining cycles  The machine tool builder can integrate these types of machining cycles.	



## Overview of touch probe cycles



- Press the **TOUCH PROBE** key.

Soft key	Cycle group	Page
	Cycles for automatic measurement and compensation of workpiece misalignment	52
	Cycles for automatic workpiece presetting	104
	Cycles for automatic workpiece inspection	162
	Special cycles	208
	Touch probe calibration	215
	Cycles for automatic kinematics measurement	231
	Cycles for automatic tool measurement (enabled by the machine manufacturer)	264
	► Switch to machine-specific touch probe cycles, if available; these touch probe cycles can be integrated by the machine tool builder	



# 3

**Using Touch Probe  
Cycles**



### 3.1 General information about touch probe cycles



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

The touch probe cycles are available only with option 17. If you are using a HEIDENHAIN touch probe, this option is automatically available.



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

#### Method of function

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

**Further information:** "Before You Start Working with Touch Probe Cycles", Page 41

When the probe stylus contacts the workpiece,

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

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

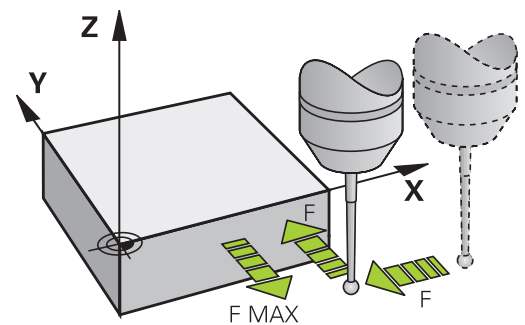
#### Consideration of a basic rotation in the Manual operation mode

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

#### Touch probe cycles in the Manual Operation and Electronic Handwheel operating modes

In the **Manual operation** and **Electronic handwheel** modes, the control provides touch probe cycles that allow you to:

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





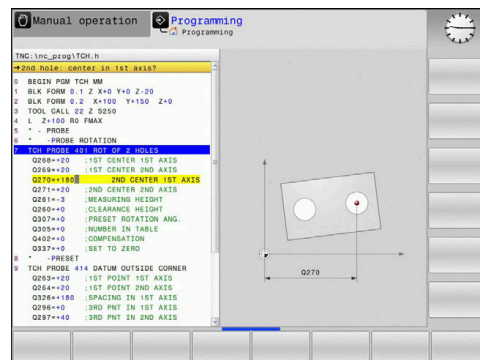
## Touch probe cycles for automatic operation

Besides the touch probe cycles, which you can use in the Manual Operation and Electronic handwheel modes, the control provides numerous cycles for a wide variety of applications in automatic mode:

- Calibrating a touch trigger probe
- Compensating workpiece misalignment
- Presetting
- Automatic workpiece inspection
- Automatic tool measurement

You can program the touch probe cycles in the **Programming** operating mode via the **TOUCH PROBE** key. Like the most recent 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 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.

Soft key	Measuring cycle group	Page
	Cycles for automatic measurement and compensation of workpiece misalignment	52
	Cycles for automatic workpiece presetting	104
	Cycles for automatic workpiece inspection	162
	Special cycles	208
	Calibrate TS	215
	Kinematics	231
	Cycles for automatic tool measurement (enabled by the machine manufacturer)	264

### NC blocks

5 TCH PROBE 410 PRESET INSIDE RECTAN.
Q321=+50 ;CENTER IN 1ST AXIS
Q322=+50 ;CENTER IN 2ND AXIS
Q323=60 ;FIRST SIDE LENGTH
Q324=20 ;2ND SIDE LENGTH
Q261=-5 ;MEASURING HEIGHT
Q320=0 ;SET-UP CLEARANCE
Q260=+20 ;CLEARANCE HEIGHT
Q301=0 ;MOVE TO CLEARANCE
Q305=10 ;NUMBER IN TABLE
Q331=+0 ;PRESET
Q332=+0 ;PRESET
Q303=+1 ;MEAS. VALUE TRANSFER
Q381=1 ;PROBE IN TS AXIS
Q382=+85 ;1ST CO. FOR TS AXIS
Q383=+50 ;2ND CO. FOR TS AXIS
Q384=+0 ;3RD CO. FOR TS AXIS
Q333=+0 ;PRESET

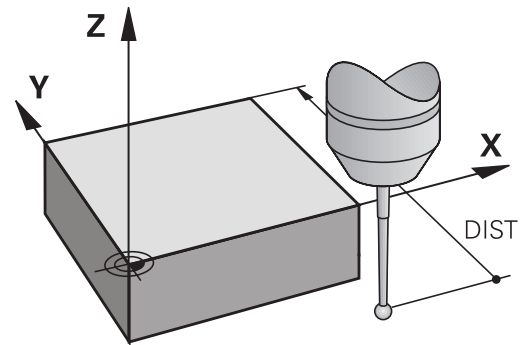


### 3.2 Before You Start Working with Touch Probe Cycles

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

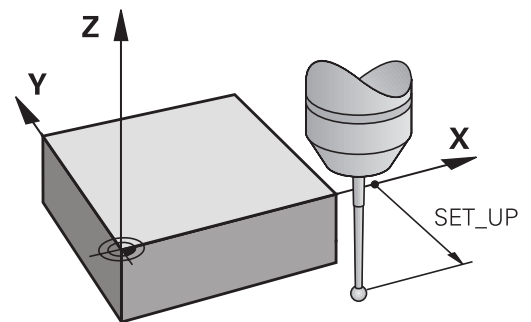
#### Maximum traverse to touch point: **DIST** in touch probe table

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



#### Set-up clearance to touch point: **SET\_UP** in touch probe table

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



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

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



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



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

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

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

The feed-rate potentiometer 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.

### NOTICE

#### Danger of collision!

When running touch probe cycles **400 to 499**, no cycles for coordinate transformation must be active.

- ▶ 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 running touch probe cycles **1400 to 1499**, no cycles for coordinate transformation must be active.

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



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



- Please note that the units of measure from **Q113** in the measurement report and return parameters depend on the main program.
- Touch probe cycles **408 to 419** and **1400 to 1499** may be run even if a basic rotation is active. Make sure, however, that the basic rotation angle does not change when you use Cycle **7 DATUM SHIFT** after the touch probe cycle.



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

- If the current coordinate of the south pole of the stylus is less than the coordinate of the clearance height (as defined in the cycle), the control first retracts the touch probe in the touch probe axis to clearance height and then positions it in the working plane to the first touch point.
- If the current coordinate of the stylus south pole is greater than the coordinate of the clearance height, then the control first positions the touch probe to the first probe point in the working plane, and then in the touch-probe axis directly to the measuring height.



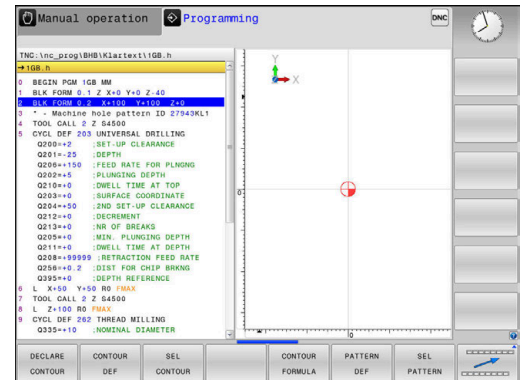
### 3.3 Program defaults for cycles

#### 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** you can define these cycle parameters at the beginning of the program, so that they are effective globally for all cycles used in the NC program. In the respective cycle you then simply reference the value defined at the beginning of the program.

The following GLOBAL DEF functions are available:






Soft key	Machining patterns	Page
100 GLOBAL DEF GENERAL	GLOBAL DEF COMMON Definition of generally valid cycle parameters	47
105 GLOBAL DEF DRILLING	GLOBAL DEF DRILLING Definition of specific drilling cycle parameters	<b>Further information:</b> User's Manual for Programming of Machining Cycles
110 GLOBAL DEF POCKET MILLING	GLOBAL DEF POCKET MILLING Definition of specific pocket-milling cycle parameters	<b>Further information:</b> User's Manual for Programming of Machining Cycles
111 GLOBAL DEF CNTR MILLING	GLOBAL DEF CONTOUR MILLING Definition of specific contour milling cycle parameters	<b>Further information:</b> User's Manual for Programming of Machining Cycles
125 GLOBAL DEF POSITIONING.	GLOBAL DEF POSITIONING Definition of the positioning behavior for <b>CYCL CALL PAT</b>	<b>Further information:</b> User's Manual for Programming of Machining Cycles
120 GLOBAL DEF PROBING	GLOBAL DEF PROBING Definition of specific touch probe cycle parameters	47

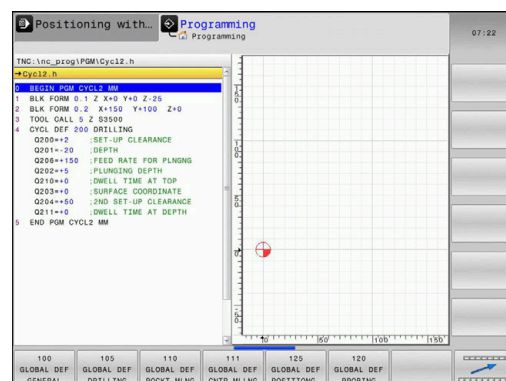




## Entering GLOBAL DEF definitions

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

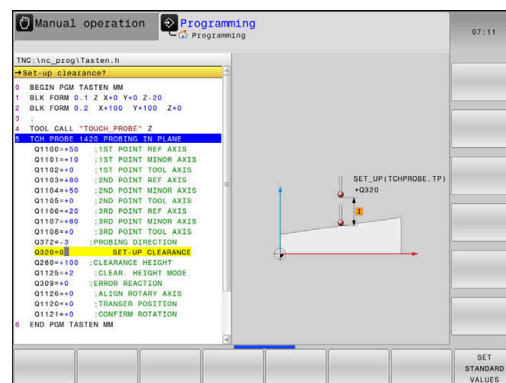


## Using GLOBAL DEF information

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.

- ▶ 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 machining cycles **2xx** and touch probe cycles **451, 452**

- ▶ **Q200 Set-up clearance?** (incremental): Distance between tool tip and workpiece surface. Enter a positive value.  
Input range: 0 to 99999.9999
- ▶ **Q204 2nd set-up clearance?** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: 0 to 99999.9999
- ▶ **Q253 Feed rate for pre-positioning?:** Feed rate at which the control traverses the tool within a cycle.  
Input range: 0 to 99999.999; alternatively **FMAX**, **FAUTO**
- ▶ **Q208 Feed rate for retraction?:** Feed rate at which the control retracts the tool. Input range: 0 to 99999.999; alternatively **FMAX**, **FAUTO**

### Example

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

## Global data for probing functions

The parameters are valid for all touch probe cycles **4xx** and **14xx** as well as for Cycles **271, 1271, 1272, 1273, 1278**

- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:  
**0**: Move at measuring height between measuring points  
**1**: Move at clearance height between measuring points

### Example

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



## 3.4 Touch-probe table

### General information

Various data are stored in the touch probe table that defines the probe behavior during the probing process. If you use several touch probes on your machine tool, you can save separate data for each touch probe.



You can also view and edit the data of the touch probe table in the Tool Management.

### Editing touch probe tables

Proceed as follows:



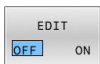
- ▶ Press the **Manual operation** key



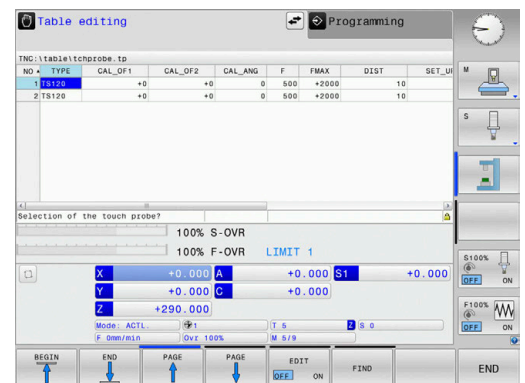
- ▶ Press the **TOUCH PROBE** soft key
- > The control displays additional soft keys.



- ▶ Press the **TCH PROBE TABLE** soft key



- ▶ Set the **EDIT** soft key to **ON**.
- ▶ Using the arrow keys, select the desired setting.
- ▶ Perform desired changes.
- ▶ Exit the touch probe table: Press the **END** soft key.





## Touch probe data

Abbr.	Inputs	Dialog
<b>NO</b>	Number of the touch probe: Enter this number in the tool table (column: <b>TP_NO</b> ) under the appropriate tool entries	–
<b>TYPE</b>	Selection of the touch probe used	<b>Selection of the touch probe?</b>
<b>CAL_OF1</b>	Offset of the touch probe axis to the spindle axis in the principal axis	<b>TS center misalignmt. ref. axis? [mm]</b>
<b>CAL_OF2</b>	Offset of the touch probe axis to the spindle axis in the secondary axis	<b>TS center misalignmt. aux. axis? [mm]</b>
<b>CAL_ANG</b>	Prior to calibrating or probing the control aligns the touch probe with the spindle angle (if spindle orientation is possible)	<b>Spindle angle for calibration?</b>
<b>F</b>	Feed rate at which the control will probe the workpiece <b>F</b> can never be greater than set in the optional <b>maxTouch-Feed</b> machine parameter (no. 122602).	<b>Probing feed rate? [mm/min]</b>
<b>FMAX</b>	Feed rate at which the touch probe is pre-positioned and is positioned between the measuring points	<b>Rapid traverse in probing cycle? [mm/min]</b>
<b>DIST</b>	If the stylus is not deflected within this defined value, the control will issue an error message.	<b>Maximum measuring range? [mm]</b>
<b>SET_UP</b>	In <b>SET_UP</b> 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 <b>SET_UP</b>	<b>Set-up clearance? [mm]</b>
<b>F_PREPOS</b>	Define the speed for pre-positioning: <ul style="list-style-type: none"> <li>■ Pre-positioning with speed from <b>FMAX</b>: <b>FMAX_PROBE</b></li> <li>■ Pre-positioning with machine rapid traverse: <b>FMAX_MACHINE</b></li> </ul>	<b>Pre-position at rapid? ENT/NOENT</b>
<b>TRACK</b>	To increase measuring accuracy, you can use <b>TRACK = ON</b> to have an infrared touch probe oriented in the programmed probe direction before every probing process. In this way, the stylus is always deflected in the same direction: <ul style="list-style-type: none"> <li>■ <b>ON</b>: Perform spindle tracking</li> <li>■ <b>OFF</b>: Do not perform spindle tracking</li> </ul>	<b>Probe oriented? Yes=ENT/No=NOENT</b>



Abbr.	Inputs	Dialog
<b>SERIAL</b>	You need not make an entry in this column. The TNC automatically enters the serial number of the touch probe if the touch probe has an EnDat interface.	<b>Serial number?</b>
<b>REACTION</b>	<p>As soon as touch probes with a collision protection adapter detect a collision, they react by resetting the ready signal. The entry defines how the control is to react to the resetting of the ready signal</p> <ul style="list-style-type: none"> <li>■ <b>NCSTOP</b>: The NC program will be interrupted</li> <li>■ <b>EMERGSTOP</b>: Emergency stop, quick braking of the axes</li> </ul>	<b>Reaction? EMERGS- TOP=ENT/NCSTOP=NOENT</b>



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

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



# 4










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



## 4.1 Overview



The control must be specifically prepared by the machine tool builder for the use of a 3-D touch probe. HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

Soft key	Cycle	Page
	PROBING IN PLANE (Cycle 1420, DIN/ISO: G1420, option 17) <ul style="list-style-type: none"> <li>■ Automatic measurement using three points</li> <li>■ Compensation via basic rotation or rotary table rotation</li> </ul>	62
	PROBING ON EDGE (Cycle 1410, DIN/ISO: G1410, option 17) <ul style="list-style-type: none"> <li>■ Automatic measurement using two points</li> <li>■ Compensation via basic rotation or rotary table rotation</li> </ul>	67
	PROBING TWO CIRCLES (Cycle 1411, DIN/ISO: G1411, option 17) <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>	73
	BASIC ROTATION (Cycle 400, DIN/ISO: G400, option 17) <ul style="list-style-type: none"> <li>■ Automatic measurement using two points</li> <li>■ Compensation via basic rotation</li> </ul>	80
	ROT. OF 2 HOLES (Cycle 401, DIN/ISO: G401, option 17) <ul style="list-style-type: none"> <li>■ Automatic measurement using two holes</li> <li>■ Compensation via basic rotation</li> </ul>	83
	ROT. OF 2 STUDS (Cycle 402, DIN/ISO: G402, option 17) <ul style="list-style-type: none"> <li>■ Automatic measurement using two studs</li> <li>■ Compensation via basic rotation</li> </ul>	87
	ROT. IN ROTARY AXIS (Cycle 403, DIN/ISO: G403, option 17) <ul style="list-style-type: none"> <li>■ Automatic measurement using two points</li> <li>■ Compensation via rotary table rotation</li> </ul>	92
	ROT. IN C AXIS (Cycle 405, DIN/ISO: G405, option 17) <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>	97
	SET BASIC ROTATION (Cycle 404, DIN/ISO: G404, option 17) <ul style="list-style-type: none"> <li>■ Setting any basic rotation</li> </ul>	101



## 4.2 Touch probe cycles 14xx: fundamentals

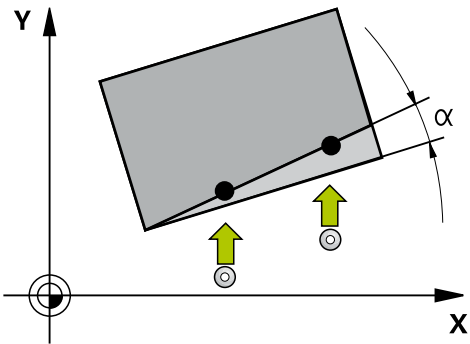
### Functionalities common to touch probe cycles 14xx for measuring rotations

Rotations can be determined using the following three cycles:

- 1410 PROBING ON EDGE
- 1411 PROBING TWO CIRCLES
- 1420 PROBING IN PLANE

These cycles include the following functionality:

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



Programming 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.

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

**Logging:**

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



## Semi-automatic mode

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

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

### Cycle run:

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

Proceed as follows:

- ▶ Use the axis-direction keys to pre-position the touch probe at the desired point

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:

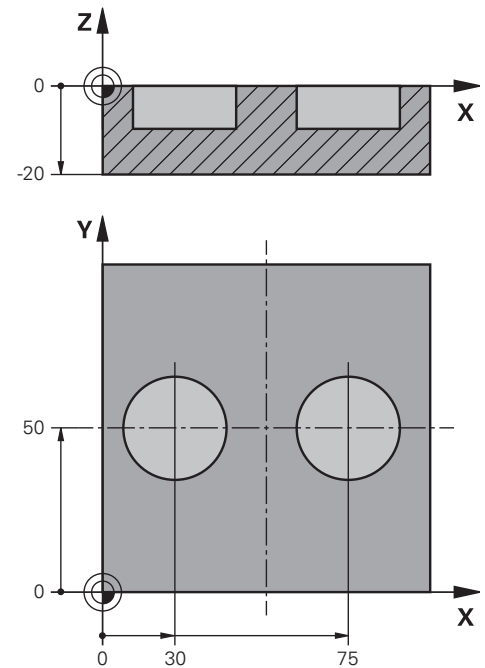
- See the drawing for these nominal positions.
- Semi-automatic mode can only be executed in the machine operating modes, i.e. not in the Test run operating mode.
- If you did not define a nominal position for a touch point in any direction, the control generates an error message.
- If you did not define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



### Examples

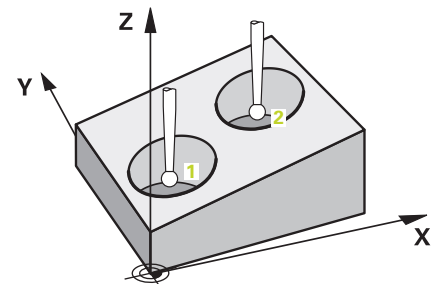
**Important:** Specify the **nominal positions** from the drawing!

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



### Hole

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

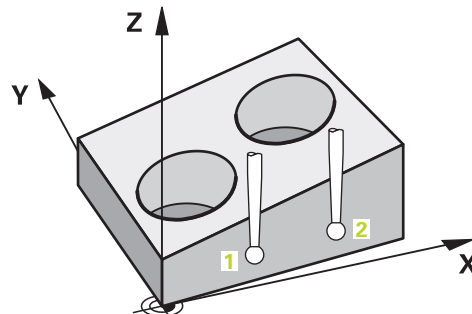


5 TCH PROBE 1411 PROBING TWO CIRCLES		Define the cycle
QS1100= "?30"	;1ST POINT REF AXIS	Nominal position 1 of principal axis exists, position of the workpiece unknown
QS1101= "?50"	;1ST POINT MINOR AXIS	Nominal position 1 of secondary axis exists, position of workpiece unknown
QS1102= "?"	;1ST POINT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



### Edge

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

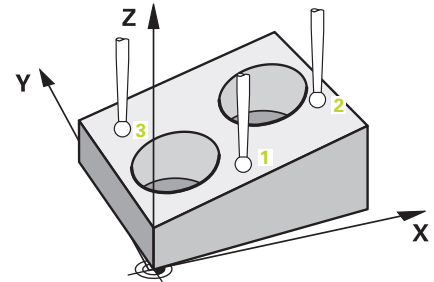


5 TCH PROBE 1410 PROBING ON EDGE		Define the cycle
QS1100= "?"	;1ST POINT REF AXIS	Nominal position 1 of principal axis unknown
QS1101= "?0"	;1ST POINT MINOR AXIS	Nominal position 1 of secondary axis exists, position of workpiece unknown
QS1102= "?"	;1ST POINT TOOL AXIS	Nominal position 1 of tool axis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



### Plane

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



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



## Evaluation of tolerances

Optionally, the control can monitor the cycles for tolerances. This includes monitoring the position and size of an object.

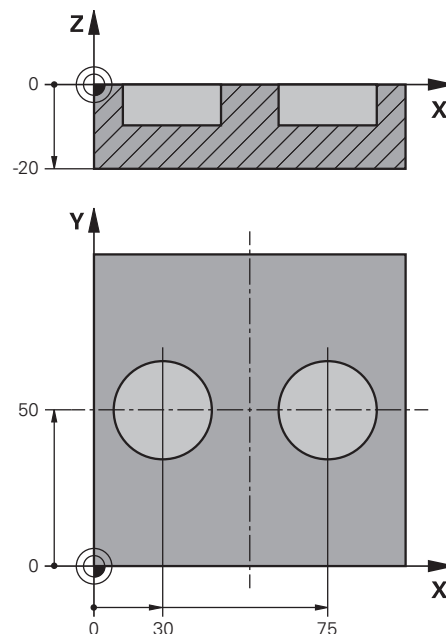
If you added a tolerance to a dimension, the control will monitor it and set an error status in the **Q183** return parameter if the tolerance is not met. Tolerance monitoring and the status always refer to the situation during probing. The preset will be corrected only afterwards, if required.

### Cycle run:

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



Please keep in mind that the touch probe cycles return the deviations with respect to the mean tolerance value in Q parameters **Q98x** and **Q99x**. Thus, these values equal the compensation values used by the cycle if input parameters **Q1120** and **Q1121** have been set accordingly. If no automatic evaluation has been programmed, the control will save the values with respect to the mean tolerance in the programmed Q parameters. You can further process them.



5 TCH PROBE 1411 PROBING TWO CIRCLES		Define the cycle
<b>Q1100=+30</b>	<b>;1ST POINT REF AXIS</b>	Nominal position 1 of reference axis
<b>Q1101= +50</b>	<b>;1ST POINT MINOR AXIS</b>	Nominal position 1 of minor axis
<b>Q1102= -5</b>	<b>;1ST POINT TOOL AXIS</b>	Nominal position 1 of tool axis
<b>QS1116="+10-1-0,5"</b>	<b>DIAMETER 1</b>	Diameter 1, tolerance specified
<b>Q1103= +75</b>	<b>;2ND POINT REF AXIS</b>	Nominal position 2 of reference axis
<b>Q1104=+50</b>	<b>;2ND POINT MINOR AXIS</b>	Nominal position 2 of minor axis
<b>QS1105= -5</b>	<b>;2ND POINT TOOL AXIS</b>	Nominal position 2 of tool axis
<b>QS1117="+10-1-0,5"</b>	<b>DIAMETER 2</b>	Diameter 2, tolerance specified
...	;	
<b>Q309=2</b>	<b>;ERROR REACTION</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: principal axis, secondary axis, and tool axis. If you define only one axis with its actual position, an error message will be generated.
- Actual positions can also be defined with Q parameters **Q1900-Q1999**.

### Example:

This feature allows you to do the following:

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

<b>5 TCH PROBE 1410 PROBING ON EDGE</b>	
<b>QS1100="10+0.02@10.0123"</b>	
<b>;1ST POINT REF AXIS</b>	Nominal position 1 of the principal axis with tolerance monitoring and actual position
<b>QS1101="50@50.0321"</b>	
<b>;1ST POINT MINOR AXIS</b>	Nominal position 1 of the secondary axis with tolerance monitoring and actual position
<b>QS1102=" -10-0.2+0.02@Q1900"</b>	
<b>;1ST POINT TOOL AXIS</b>	Nominal position 1 of the tool axis with tolerance monitoring and actual position
<b>...</b>	<b>;</b>



### 4.3 PROBING IN PLANE (Cycle 1420, DIN/ISO: G1420, option 17)

#### Application

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

Furthermore, you can perform the following operations with Cycle **1420**:

- If the probing position relative to the current datum is unknown, you can execute the cycle in semi-automatic mode

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

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

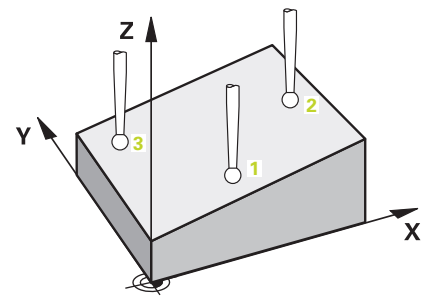
**Further information:** "Evaluation of tolerances", Page 60

- If you determine the actual position in advance, you can then define it as the actual position for the cycle

**Further information:** "Transferring the actual position", Page 61

#### Cycle run

- 1 The control positions the touch probe at the feed rate (depending on **Q1125**), using positioning logic "Executing touch probe cycles", to the programmed touch point **1**. There the control measures the first point in the plane. The control offsets the touch probe by the safety clearance in the direction opposite to the direction of probing.
- 2 If you programmed Traverse to clearance height, the touch probe returns to clearance height (depending on the setting in **Q1125**). It then moves in the working plane to touch point **2** to measure there the actual value of the second touch point in the plane.
- 3 The touch probe returns to the clearance height (depends on **Q1125**), then moves in the working plane to touch point **3** and measures the actual position of the third point of the plane.
- 4 Finally the control retracts the touch probe to the clearance height (depends on **Q1125**) and saves the measured values in the following Q parameters:





Parameter number	Meaning
<b>Q950 to Q952</b>	Measured position 1 in the principal, secondary, and tool axes
<b>Q953 to Q955</b>	Measured position 2 in the principal, secondary, and tool axes
<b>Q956 to Q958</b>	Measured position 3 in the principal, secondary, and tool axes
<b>Q961 to Q963</b>	Measured spatial angle SPA, SPB, and SPC in the W-CS
<b>Q980 to Q982</b>	Deviation 1 of the positions
<b>Q983 to Q985</b>	Deviation 2 of the positions
<b>Q986 to Q988</b>	Deviation 3 of the positions
<b>Q183</b>	Workpiece status (-1=not defined / 0=OK / 1=Rework / 2=Scrap)

### Please note while programming!

#### NOTICE

##### Danger of collision!

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

- Always move to clearance height between objects or touch points

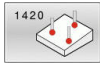
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- HEIDENHAIN recommends avoiding the use of axis angles in this cycle!
- The control can only calculate the angular values if the three touch points are not positioned on a straight line.
- The nominal spatial angle results from the defined nominal positions. The control saves the calculated spatial angle in parameters **Q961** to **Q963**. The control transfers the difference between the measured spatial angle and the nominal spatial angle to the basic rotation in 3-D of the preset table.

#### Aligning the rotary table axes:

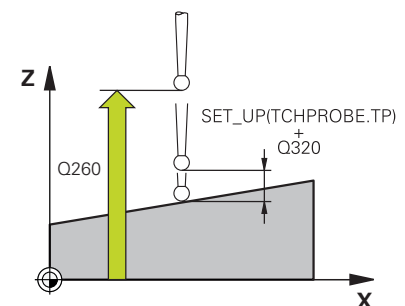
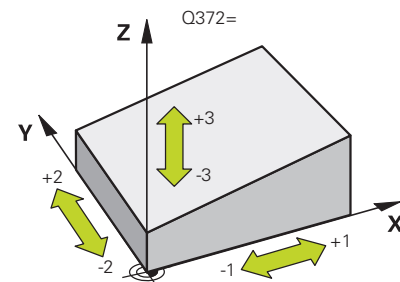
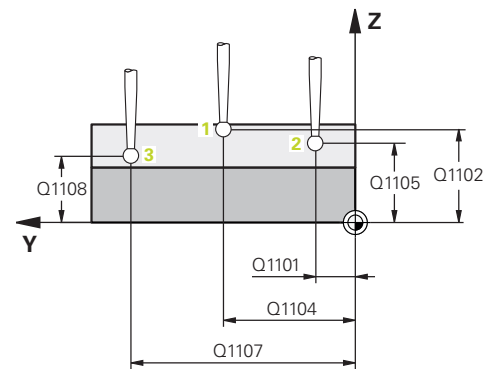
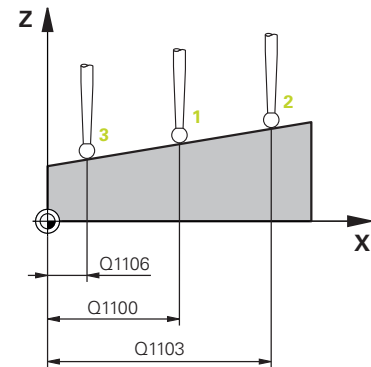
- Alignment with rotary table axes is only possible if two rotary table axes have been defined in the kinematics.
- To align the rotary table axes, (**Q1126** not equal to 0), the rotation must be saved (**Q1121** not equal to 0). Otherwise, an error message will be displayed because it is not possible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



## Cycle parameters



- ▶ **Q1100 1st noml. position of ref. axis** (absolute):  
Nominal position of the first touch point in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1101 1st noml. position of minor axis** (absolute):  
Nominal position of the first touch point in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1102 1st nominal position tool axis?** (absolute):  
Nominal position of the first touch point in the tool axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1103 2nd noml. position of ref axis?** (absolute):  
Nominal position of the second touch point in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1104 2nd noml. position of minor axis** (absolute):  
Nominal position of the second touch point in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1105 2nd nominal pos. of tool axis?** (absolute):  
nominal position of the second touch point in the tool axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1106 3rd noml. position of ref axis?** (absolute):  
Nominal position of the third touch point in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1107 3rd noml. position minor axis?** (absolute):  
Nominal position of the third touch point in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1108 3rd nominal position tool axis?** (absolute):  
Nominal position of the third touch point in the tool axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q372 Probe direction (-3 to +3)?**: Specify the axis in which probing is to be performed. With the algebraic sign, you define the positive or negative direction of traverse of the probing axis.  
Input range: -3 to +3
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999





- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1125 Traverse to clearance height?:** Definition of how the touch probe is to move between the probing points:  
  - 1: Do not move to clearance height. Pre-positioning occurs at **FMAX\_PROBE**
  - 0: Move to clearance height before and after the cycle. Pre-positioning occurs at **FMAX\_PROBE**
  - 1: Move to clearance height before and after each object. Pre-positioning occurs at **FMAX\_PROBE**
  - 2: Move to clearance height before and after each probing point. Pre-positioning occurs at a feed rate of **F2000**
- ▶ **Q309 Reaction to tolerance error?:** Specify whether the control is to interrupt the program run and display a message if a deviation is detected:  
  - 0: If the tolerance is exceeded, do not interrupt the program run, do not display an error message
  - 1: If the tolerance is exceeded, interrupt the program run and display an error message
  - 2: If the determined actual position indicates that the workpiece is scrap, the control displays a message and interrupts the program run.

In contrast, there will be no error reaction if the determined value is in a range where the workpiece can be reworked.

#### Example

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



- ▶ **Q1126 Align rotary axes?:** positioning the tilting axes for inclined machining:
  - 0:** maintain the current tilting axis position
  - 1:** position the tilting axis automatically and orient the probe tip (MOVE). The relative position between workpiece and the touch probe remains unchanged. The control performs a compensating movement with the linear axes
  - 2:** position the tilting axis automatically without orienting the probe tip (TURN)
- ▶ **Q1120 Transfer position?:** Define which touch point will be used to correct the current preset:
  - 0:** No correction
  - 1:** Correction based on the 1st touch point
  - 2:** Correction based on the 2nd touch point
  - 3:** Correction based on the 3rd touch point
  - 4:** Correction based on the averaged touch point position
- ▶ **Q1121 Confirm basic rotation?:** Define whether the control is to transfer the determined inclination as the basic rotation:
  - 0:** No basic rotation
  - 1:** Set basic rotation: The control saves the basic rotation.



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

### Application

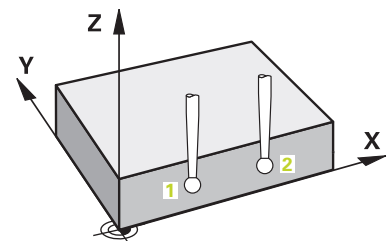
Touch probe cycle **1410** determines the misalignment of a workpiece by measuring two points located on an edge. The cycle determines the rotation based on the difference between the measured angle and the nominal angle.

Furthermore, you can perform the following operations with Cycle **1410**:

- If the probing position relative to the current datum is unknown, you can execute the cycle in semi-automatic mode  
**Further information:** "Semi-automatic mode", Page 55
- Optionally, the cycle can monitor the tolerances. That way you can monitor the position and size of an object  
**Further information:** "Evaluation of tolerances", Page 60
- If you determine the actual position in advance, you can then define it as the actual position for the cycle  
**Further information:** "Transferring the actual position", Page 61

### Cycle run

- 1 The control positions the touch probe at the feed rate (depending on **Q1125**), using positioning logic "Executing touch probe cycles", to the programmed touch point **1**. The sum of **Q320**, **SET\_UP** and the ball-tip radius is taken into account when probing in any probing direction. The control offsets the touch probe in the direction opposite to the direction of probing
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 The touch probe then moves to the next touch point **2** and probes again.
- 4 Finally, the control retracts the touch probe to the clearance height (depends on **Q1125**) and saves the measured values in the following Q parameters:





Parameter number	Meaning
<b>Q950 to Q952</b>	Measured position 1 in the principal, secondary, and tool axes
<b>Q953 to Q955</b>	Measured position 2 in the principal, secondary, and tool axes
<b>Q964</b>	Measured angle of rotation
<b>Q965</b>	Measured angle of rotation in the coordinate system of the rotary table
<b>Q980 to Q982</b>	Deviation 1 of the positions
<b>Q983 to Q985</b>	Deviation 2 of the positions
<b>Q994</b>	Measured angle deviation
<b>Q995</b>	Measured angle deviation in the coordinate system of the rotary table
<b>Q183</b>	Workpiece status (-1=not defined / 0=OK / 1=Rework / 2=Scrap)



## Please note while programming!

### NOTICE

#### Danger of collision!

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

- Always move to clearance height between objects or touch points

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

#### To determine the basic rotation in an actively tilted working plane, please remember:

- If the current coordinates of the rotary axes and the defined tilt angles (3-D ROT menu) match, the working plane is consistent. Thus, the basic rotation is calculated in the input coordinate system (I-CS), based on the tool axis.
- If the current coordinates of the rotary axes and the defined tilt angles (3-D ROT menu) do not match, the working plane is inconsistent. Thus, the basic rotation is calculated in the workpiece coordinate system (W-CS), based on the tool axis.



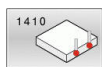
If no checking is configured in **chkTiltingAxes** (no. 204601), the cycle assumes that the working plane is consistent. When calculating the basic rotation, the system then uses the I#CS system.

#### Aligning the rotary table axes:

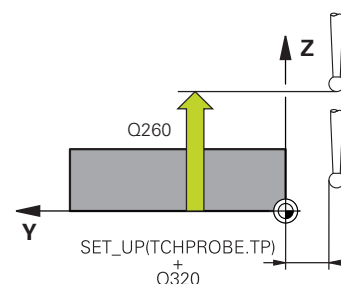
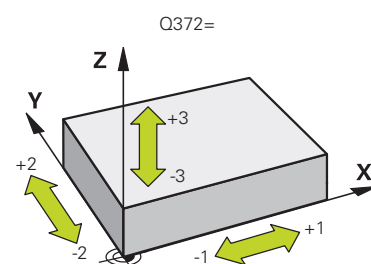
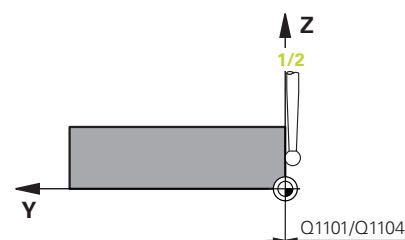
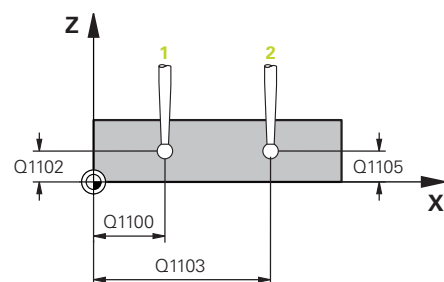
- Alignment with rotary table axes is only possible if the measured rotation can be compensated using a rotary table axis. This must be the first rotary table axis, seen from the workpiece.
- To align the rotary table axes, (**Q1126** not equal to 0), the rotation must be saved (**Q1121** not equal to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



## Cycle parameters



- ▶ **Q1100 1st noml. position of ref. axis** (absolute):  
Nominal position of the first touch point in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1101 1st noml. position of minor axis** (absolute): Nominal position of the first touch point in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1102 1st nominal position tool axis?** (absolute):  
Nominal position of the first touch point in the tool axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1103 2nd noml. position of ref axis?** (absolute):  
Nominal position of the second touch point in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1104 2nd noml. position of minor axis** (absolute): Nominal position of the second touch point in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1105 2nd nominal pos. of tool axis?** (absolute):  
nominal position of the second touch point in the tool axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q372 Probe direction (-3 to +3)?**: Specify the axis in which probing is to be performed. With the algebraic sign, you define the positive or negative direction of traverse of the probing axis.  
Input range: -3 to +3
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999





- **Q1125 Traverse to clearance height?:** Definition of how the touch probe is to move between the probing points:
  - 1: Do not move to clearance height. Pre-positioning occurs at **FMAX\_PROBE**
  - 0: Move to clearance height before and after the cycle. Pre-positioning occurs at **FMAX\_PROBE**
  - 1: Move to clearance height before and after each object. Pre-positioning occurs at **FMAX\_PROBE**
  - 2: Move to clearance height before and after each probing point. Pre-positioning occurs at a feed rate of **F2000**
- **Q309 Reaction to tolerance error?:** Specify whether the control is to interrupt the program run and display a message if a deviation is detected:
  - 0: If the tolerance is exceeded, do not interrupt the program run, do not display an error message
  - 1: If the tolerance is exceeded, interrupt the program run and display an error message
  - 2: If the determined actual position indicates that the workpiece is scrap, the control displays a message and interrupts the program run. In contrast, there will be no error reaction if the determined value is in a range where the workpiece can be reworked.

#### Example

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



- ▶ **Q1126 Align rotary axes?:** positioning the tilting axes for inclined machining:
  - 0:** maintain the current tilting axis position
  - 1:** position the tilting axis automatically and orient the probe tip (MOVE). The relative position between workpiece and the touch probe remains unchanged. The control performs a compensating movement with the linear axes
  - 2:** position the tilting axis automatically without orienting the probe tip (TURN)
- ▶ **Q1120 Transfer position?:** Define which touch point will be used to correct the current preset:
  - 0:** No correction
  - 1:** Correction based on the 1st touch point
  - 2:** Correction based on the 2nd touch point
  - 3:** Correction based on the averaged touch point position
- ▶ **Q1121 CONFIRM ROTATION?:** definition of whether the control is to set the determined misalignment as a basic rotation:
  - 0:** no basic rotation
  - 1:** set basic rotation: the control saves the basic rotation
  - 2:** rotary table rotation: the control will make an entry in the corresponding **Offset** column of the preset table



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

### Application

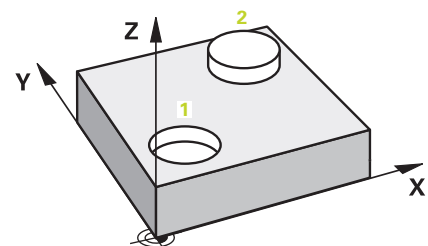
Touch probe cycle **1411** captures the center points of two holes or cylindrical studs and calculates a straight line connecting these center points. The cycle determines the rotation in the working plane based on the difference between the measured angle and the nominal angle.

Furthermore, you can perform the following operations with Cycle **1411**:

- If the probing position relative to the current datum is unknown, you can execute the cycle in semi-automatic mode  
**Further information:** "Semi-automatic mode", Page 55
- Optionally, the cycle can monitor the tolerances. That way you can monitor the position and size of an object  
**Further information:** "Evaluation of tolerances", Page 60
- If you determine the actual position in advance, you can then define it as the actual position for the cycle  
**Further information:** "Transferring the actual position", Page 61

### Cycle run

- 1 The control positions the touch probe at the feed rate (depending on **Q1125**), using positioning logic "Executing touch probe cycles", to the programmed center point **1**. The sum of **Q320**, **SET\_UP** and the ball-tip radius is taken into account when probing in any probing direction. The control offsets the touch probe by the safety clearance in the direction opposite to the direction of probing.
- 2 Then the probe moves to the entered measuring height and probes (depending on the number of probes in **Q423**) the first hole or stud center point.
- 3 The touch probe returns to the clearance height and then to the position entered as center of the second hole or second stud **2**.
- 4 Then the control moves the probe to the entered measuring height and probes (depending on the number of probes in **Q423**) the second hole or stud center point.
- 5 Finally, the control retracts the touch probe to the clearance height (depends on **Q1125**) and saves the measured values in the following Q parameters:





Parameter number	Meaning
<b>Q950 to Q952</b>	Measured position 1 in the principal, secondary, and tool axes
<b>Q953 to Q955</b>	Measured position 2 in the principal, secondary, and tool axes
<b>Q964</b>	Measured angle of rotation
<b>Q965</b>	Measured angle of rotation in the coordinate system of the rotary table
<b>Q966 to Q967</b>	Measured first and second diameters
<b>Q980 to Q982</b>	Deviation 1 of the positions
<b>Q983 to Q985</b>	Deviation 2 of the positions
<b>Q994</b>	Measured angle deviation
<b>Q995</b>	Measured angle deviation in the coordinate system of the rotary table
<b>Q996 to Q997</b>	Measurement error of the first and second diameters
<b>Q183</b>	Workpiece status (-1=not defined / 0=OK / 1=Rework / 2=Scrap)



## Please note while programming!

### NOTICE

#### Danger of collision!

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

- Always move to clearance height between objects or touch points

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

#### To determine the basic rotation in an actively tilted working plane, please remember:

- If the current coordinates of the rotary axes and the defined tilt angles (3-D ROT menu) match, the working plane is consistent. Thus, the basic rotation is calculated in the input coordinate system (I-CS), based on the tool axis.
- If the current coordinates of the rotary axes and the defined tilt angles (3-D ROT menu) do not match, the working plane is inconsistent. Thus, the basic rotation is calculated in the workpiece coordinate system (W-CS), based on the tool axis.



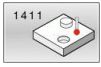
If no checking is configured in **chkTiltingAxes** (no. 204601), the cycle assumes that the working plane is consistent. When calculating the basic rotation, the system then uses the I#CS system.

#### Aligning the rotary table axes:

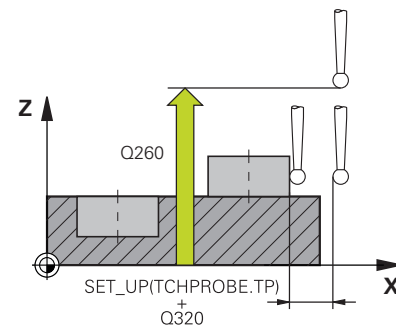
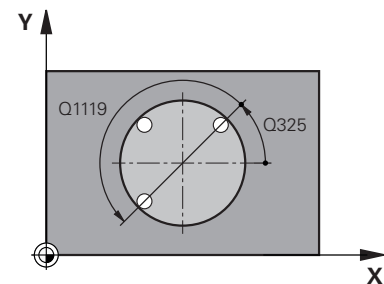
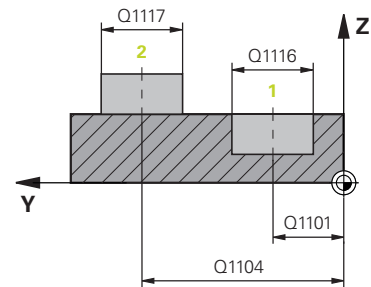
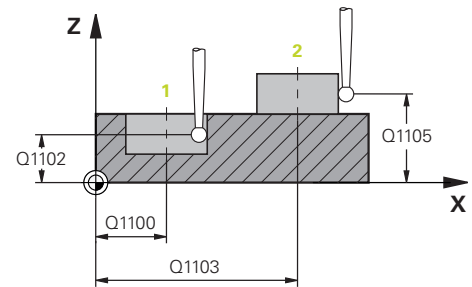
- Alignment with rotary table axes is only possible if the measured rotation can be compensated using a rotary table axis. This must be the first rotary table axis, seen from the workpiece.
- To align the rotary table axes, (**Q1126** not equal to 0), the rotation must be saved (**Q1121** not equal to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



## Cycle parameters



- ▶ **Q1100 1st noml. position of ref. axis** (absolute): Nominal position of the first touch point in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1101 1st noml. position of minor axis** (absolute): Nominal position of the first touch point in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1102 1st nominal position tool axis?** (absolute): Nominal position of the first touch point in the tool axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1116 Diameter of 1st position?**: Diameter of the first hole or stud.  
Input range: 0 to 9999.9999
- ▶ **Q1103 2nd noml. position of ref axis?** (absolute): Nominal position of the second touch point in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1104 2nd noml. position of minor axis** (absolute): Nominal position of the second touch point in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1105 2nd nominal pos. of tool axis?** (absolute): nominal position of the second touch point in the tool axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1117 Diameter of 2nd position?**: Diameter of the second hole or stud.  
Input range: 0 to 9999.9999
- ▶ **Q1115 Geometry type (0-3)?**: Specify the geometry of the objects  
**0**: position 1 = hole and position 2 = hole  
**1**: position 1 = stud and position 2 = stud  
**2**: position 1 = hole and position 2 = stud  
**3**: position 1 = stud and position 2 = hole
- ▶ **Q423 Number of probes?** (absolute): Number of touch points on the diameter.  
Input range: 3 to 8
- ▶ **Q325 Starting angle?** (absolute): Angle between the principal axis of the working plane and the first touch point.  
Input range: -360.000 to 360.000





- ▶ **Q1119 Arc angular length?:** Angular range in which the touch points are distributed.  
Input range: -359.999 to +360.000
- ▶ **Q320 Set-up clearance?** (incremental): Additional distance between the touch point and ball tip.  
**Q320** is added to **SET\_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q1125 Traverse to clearance height?:** Definition of how the touch probe is to move between the probing points:
  - 1: Do not move to clearance height. Pre-positioning occurs at **FMAX\_PROBE**
  - 0: Move to clearance height before and after the cycle. Pre-positioning occurs at **FMAX\_PROBE**
  - 1: Move to clearance height before and after each object. Pre-positioning occurs at **FMAX\_PROBE**
  - 2: Move to clearance height before and after each probing point. Pre-positioning occurs at a feed rate of **F2000**
- ▶ **Q309 Reaction to tolerance error?:** Specify whether the control is to interrupt the program run and display a message if a deviation is detected:
  - 0: If the tolerance is exceeded, do not interrupt the program run, do not display an error message
  - 1: If the tolerance is exceeded, interrupt the program run and display an error message
  - 2: If the determined actual position indicates that the workpiece is scrap, the control displays a message and interrupts the program run.

In contrast, there will be no error reaction if the determined value is in a range where the workpiece can be reworked.

#### Example

5 TCH PROBE 1410 PROBING TWO CIRCLES	
Q1100=+0	;1ST POINT REF AXIS
Q1101=+0	;1ST POINT MINOR AXIS
Q1102=+0	;1ST POINT TOOL AXIS
Q1116=0	;DIAMETER 1
Q1103=+0	;2ND POINT REF AXIS
Q1104=+0	;2ND POINT MINOR AXIS
Q1105=+0	;2ND POINT TOOL AXIS
Q1117=+0	;DIAMETER 2
Q1115=0	;GEOMETRY TYPE
Q423=4	;NO. OF PROBE POINTS
Q325=+0	;STARTING ANGLE
Q1119=+360	;ANGULAR LENGTH
Q320=+0	;SET-UP CLEARANCE
Q260=+100	;CLEARANCE HEIGHT
Q1125=+2	;CLEAR. HEIGHT MODE
Q309=+0	;ERROR REACTION
Q1126=+0	;ALIGN ROTARY AXIS
Q1120=+0	;TRANSER POSITION
Q1121=+0	;CONFIRM ROTATION



- ▶ **Q1126 Align rotary axes?:** positioning the tilting axes for inclined machining:
  - 0:** maintain the current tilting axis position
  - 1:** position the tilting axis automatically and orient the probe tip (MOVE). The relative position between workpiece and the touch probe remains unchanged. The control performs a compensating movement with the linear axes
  - 2:** position the tilting axis automatically without orienting the probe tip (TURN)
- ▶ **Q1120 Transfer position?:** Define which touch point will be used to correct the current preset:
  - 0:** No correction
  - 1:** Correction based on the 1st touch point
  - 2:** Correction based on the 2nd touch point
  - 3:** Correction based on the averaged touch point position
- ▶ **Q1121 CONFIRM ROTATION?:** definition of whether the control is to set the determined misalignment as a basic rotation:
  - 0:** no basic rotation
  - 1:** set basic rotation: the control saves the basic rotation
  - 2:** rotary table rotation: the control will make an entry in the corresponding **Offset** column of the preset table



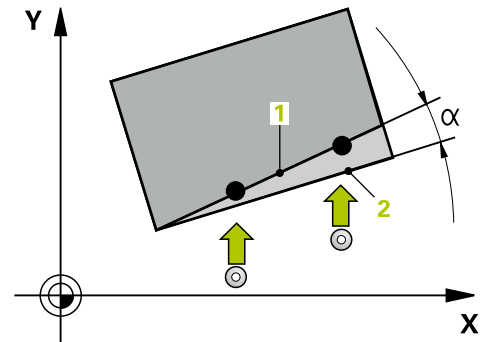
## 4.6 Touch probe cycles 4xx: fundamentals

### Characteristics common to all touch probe cycles for measuring workpiece misalignment

For Cycles **400**, **401**, and **402** you can define through parameter **Q307 Preset value for rotation angle** whether the measurement result is to be corrected by a known angle  $\alpha$  (see figure at right). This enables you to measure the basic rotation against any straight line **1** of the workpiece and to establish the reference to the actual  $0^\circ$  direction **2**.



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





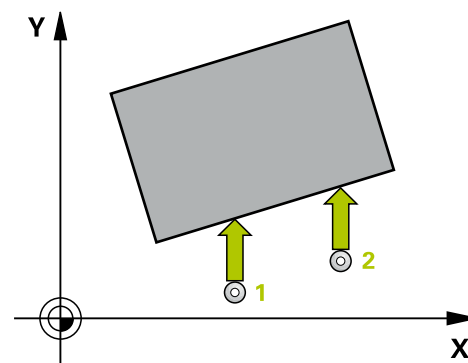
## 4.7 BASIC ROTATION (Cycle 400, DIN/ISO: G400, option 17)

### 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.

### Cycle run

- 1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), at the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the defined traverse direction
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 The touch probe then moves to the next touch point **2** and probes again.
- 4 The control returns the touch probe to the clearance height and performs the basic rotation it determined.



### Please note while programming:

#### NOTICE

##### Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- 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.

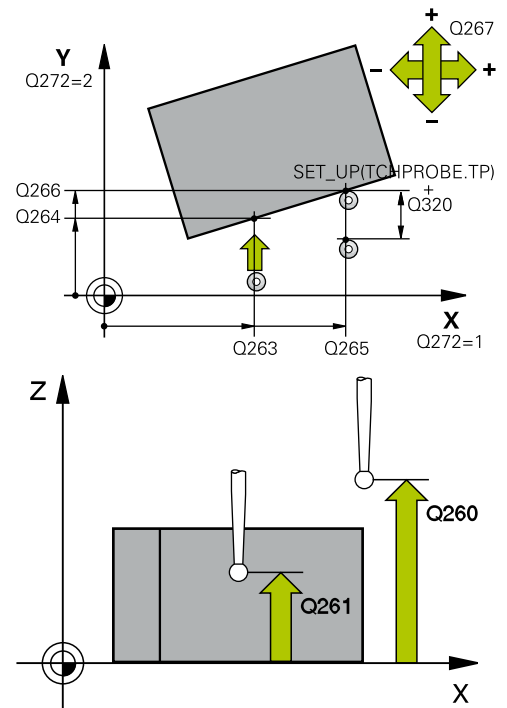
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The control will reset an active basic rotation at the beginning of the cycle.



## Cycle parameters



- ▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:  
1: Principal axis = measuring axis  
2: Secondary axis = measuring axis
- ▶ **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:  
-1: Negative traverse direction  
+1: Positive traverse direction
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:  
0: Move at measuring height between measuring points  
1: Move at clearance height between measuring points



### Example

5 TCH PROBE 400 BASIC ROTATION	
Q263=+10	;1ST POINT 1ST AXIS
Q264=+3,5	;1ST POINT 2ND AXIS
Q265=+25	;2ND POINT 1ST AXIS
Q266=+2	;2ND PNT IN 2ND AXIS
Q272=+2	;MEASURING AXIS
Q267=+1	;TRAVERSE DIRECTION
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q307=0	;PRESET ROTATION ANG.
Q305=0	;NUMBER IN TABLE



- ▶ **Q307 Preset value for rotation angle** (absolute):  
If the misalignment is to be measured against any straight line rather than the principal axis, enter the angle of this reference line. The control will then calculate the difference between the value measured and the angle of the reference line for the basic rotation.  
Input range: -360.000 to 360.000
- ▶ **Q305 Preset number in table?**: Enter the number of the preset in the preset table in which the control is to save the determined basic rotation. If you enter **Q305=0**, the control automatically stores the determined basic rotation in the ROT menu of the Manual Operation mode.  
Input range: 0 to 99999



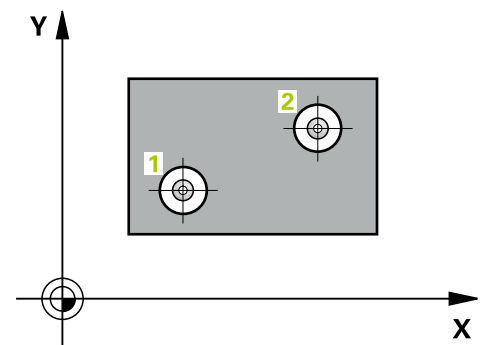
## 4.8 ROT. OF 2 HOLES (Cycle 401, DIN/ISO: G401, option 17)

### Application

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

### Cycle run

- 1 The control positions the touch probe at rapid traverse speed (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), to the programmed center point of the first hole **1**.
- 2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
- 3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
- 4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
- 5 Then the control returns the touch probe to the clearance height and performs the basic rotation it determined.



### Please note while programming:

#### NOTICE

##### Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- 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.

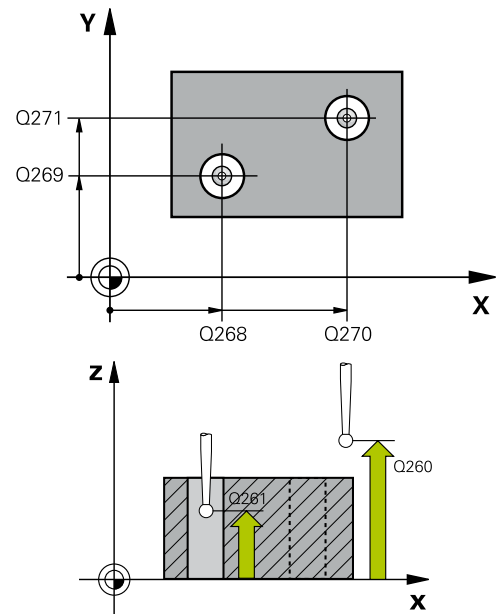
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The control will reset an active basic rotation at the beginning of the cycle.
- If you want to compensate the misalignment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



## Cycle parameters



- ▶ **Q268 1st hole: center in 1st axis?** (absolute):  
Center of the first hole in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q269 1st hole: center in 2nd axis?** (absolute):  
Center of the first hole in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q270 2nd hole: center in 1st axis?** (absolute):  
Center of the second hole in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q271 2nd hole: center in 2nd axis?** (absolute):  
Center of the second hole in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q261 Measuring height in probe axis?** (absolute):  
Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999



### Example

**5 TCH PROBE 401 ROT OF 2 HOLES**

**Q268=-37 ;1ST CENTER 1ST AXIS**

**Q269=+12 ;1ST CENTER 2ND AXIS**



- ▶ **Q307 Preset value for rotation angle** (absolute):  
If the misalignment is to be measured against any straight line rather than the principal axis, enter the angle of this reference line. The control will then calculate the difference between the value measured and the angle of the reference line for the basic rotation.  
Input range: -360.000 to 360.000
- ▶ **Q305 Number in table?** Enter the number of a row in the preset table. The control will enter the value in this row:  
**Q305 = 0:** The rotary axis will be zeroed in row 0 of the preset table. The control will make an entry in the **OFFSET** column. (Example: For tool axis Z, the entry is made in **C\_OFFS**). In addition, all other values (X, Y, Z, etc.) of the currently active preset will be transferred to row 0 of the preset table. In addition, the control activates the preset from row 0.  
**Q305 > 0:** The rotary axis will be zeroed in the preset table row specified here. The control will make an entry in the corresponding **OFFSET** column of the preset table. (Example: For tool axis Z, the entry is made in **C\_OFFS**).  
**Q305 depends on the following parameters:**  
**Q337 = 0** and, at the same time, **Q402 = 0:** A basic rotation will be set in the row specified in **Q305**. (Example: For tool axis Z, an entry is made in column **SPC**)  
**Q337 = 0** and, at the same time, **Q402 = 1:** Parameter **Q305** is not effective  
**Q337 = 1:** Parameter **Q305** is in effect as described above  
 Input range: 0 to 99 999

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



- ▶ **Q402 Basic rotation/alignment (0/1):** Define here whether the control is to set the determined misalignment as a basic rotation or whether to compensate it by a rotation of the rotary table:  
**0:** Set basic rotation: the control saves the basic rotation (example: for tool axis Z, the control uses the **SPC** column)  
**1:** Rotate the rotary table: an entry will be made in the **Offset** column of the preset table (example: for tool axis Z, the control uses the **C\_Offs** column), in addition, the corresponding axis will be rotated
- ▶ **Q337 Set to zero after alignment?:** Define whether the control is to set the position display of the corresponding rotary axis to 0 after the alignment:  
**0:** The position display is not set to 0 after the alignment  
**1:** After the alignment, the position display is set to 0, provided you have defined **Q402=1**



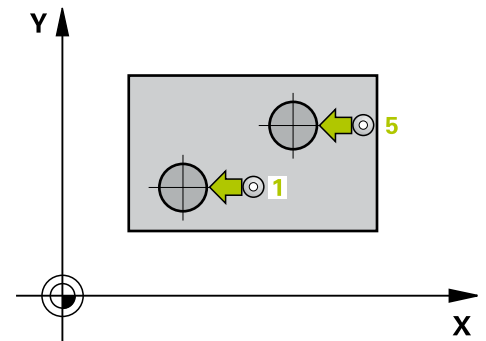
## 4.9 ROT. OF 2 STUDS (Cycle 402, DIN/ISO: G402, option 17)

### Application

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

### Cycle run

- 1 The control positions the touch probe at rapid traverse speed (value from FMAX column), using positioning logic (see "Executing touch probe cycles", Page 43), to the programmed center point of the first stud **1**.
- 2 Then the probe moves to the entered **measuring height 1** and probes four points to find the center of the first stud. The touch probe moves on a circular arc between the touch points, each of which is offset by 90°.
- 3 The touch probe returns to the clearance height and then positions the probe to starting point **5** of the second stud.
- 4 The control moves the touch probe to the entered **measuring height 2** and probes four points to determine the second stud center point.
- 5 Then the control returns the touch probe to the clearance height and performs the basic rotation it determined.





**Please note while programming:**

**NOTICE**

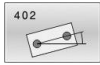
**Danger of collision!**

When running touch probe cycles **400 to 499**, no cycles for coordinate transformation must be active.

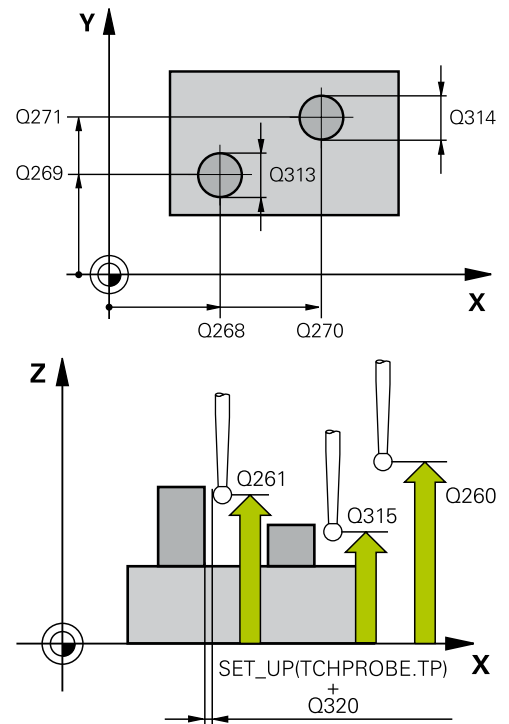
- ▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
  - ▶ Reset any coordinate transformations beforehand.
- This cycle can only be executed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



## Cycle parameters



- ▶ **Q268 1st stud: center in 1st axis?** (absolute):  
Center of the first stud in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q269 1st stud: center in 2nd axis?** (absolute):  
Center of the first stud in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q313 Diameter of stud 1?**: Approximate diameter of the first stud. Enter a value that is more likely to be too large than too small.  
Input range: 0 to 99999.9999
- ▶ **Q261 Meas. height stud 1 in TS axis?** (absolute):  
coordinate of the ball tip center (= touch point) in the touch probe axis at which stud 1 is to be measured.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q270 2nd stud: center in 1st axis?** (absolute):  
Center of the second stud in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q271 2nd stud: center in 2nd axis?** (absolute):  
Center of the second stud in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q313 Diameter of stud 2?**: Approximate diameter of the second stud. Enter a value that is more likely to be too large than too small.  
Input range: 0 to 99999.9999
- ▶ **Q315 Meas. height stud 2 in TS axis?** (absolute):  
coordinate of the ball tip center (= touch point) in the touch probe axis at which stud 2 is to be measured.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:  
  - 0**: Move at measuring height between measuring points
  - 1**: Move at clearance height between measuring points



### Example

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



► **Q307 Preset value for rotation angle** (absolute):

If the misalignment is to be measured against any straight line rather than the principal axis, enter the angle of this reference line. The control will then calculate the difference between the value measured and the angle of the reference line for the basic rotation.

Input range: -360.000 to 360.000

► **Q305 Number in table?** Enter the number of a row in the preset table. The control will enter the value in this row:

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

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

**Q305 depends on the following parameters:**

**Q337 = 0** and, at the same time, **Q402 = 0:** A basic rotation will be set in the row specified in **Q305**. (Example: For tool axis Z, an entry is made in column **SPC**)

**Q337 = 0** and, at the same time, **Q402 = 1:** Parameter **Q305** is not effective

**Q337 = 1:** Parameter **Q305** is in effect as described above

Input range: 0 to 99 999



- ▶ **Q402 Basic rotation/alignment (0/1):** Define here whether the control is to set the determined misalignment as a basic rotation or whether to compensate it by a rotation of the rotary table:  
**0:** Set basic rotation: the control saves the basic rotation (example: for tool axis Z, the control uses the **SPC** column)  
**1:** Rotate the rotary table: an entry will be made in the **Offset** column of the preset table (example: for tool axis Z, the control uses the **C\_Offs** column), in addition, the corresponding axis will be rotated
- ▶ **Q337 Set to zero after alignment?:** Define whether the control is to set the position display of the corresponding rotary axis to 0 after the alignment:  
**0:** The position display is not set to 0 after the alignment  
**1:** After the alignment, the position display is set to 0, provided you have defined **Q402=1**



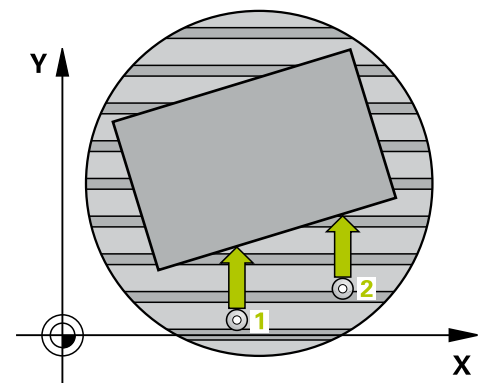
## 4.10 ROT. IN ROTARY AXIS (Cycle 403, DIN/ISO: G403, option 17)

### Application

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

### Cycle run

- 1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), at the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the defined traverse direction
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 The touch probe then moves to the next touch point **2** and probes again.
- 4 The control returns the touch probe to the clearance height and rotates the rotary axis, which was defined in the cycle, by the measured value. Optionally, you can specify whether the control is to set the determined rotation angle to 0 in the preset table or in the datum table.





**Please note while programming:**

**NOTICE**

**Danger of collision!**

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

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

**NOTICE**

**Danger of collision!**

If you set parameter **Q312** Axis for compensating movement? to 0, then the cycle will automatically determine the rotary axis to be aligned (recommended setting). When doing so, it determines an angle that depends on the sequence of the touch points.

The measured angle goes from the first to the second touch point. If you select the A, B or C axis as compensation axis in parameter **Q312**, the cycle determines the angle, regardless of the sequence of the probing points. The calculated angle lies in the range from  $-90^\circ$  to  $+90^\circ$ .

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

**NOTICE**

**Danger of collision!**

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

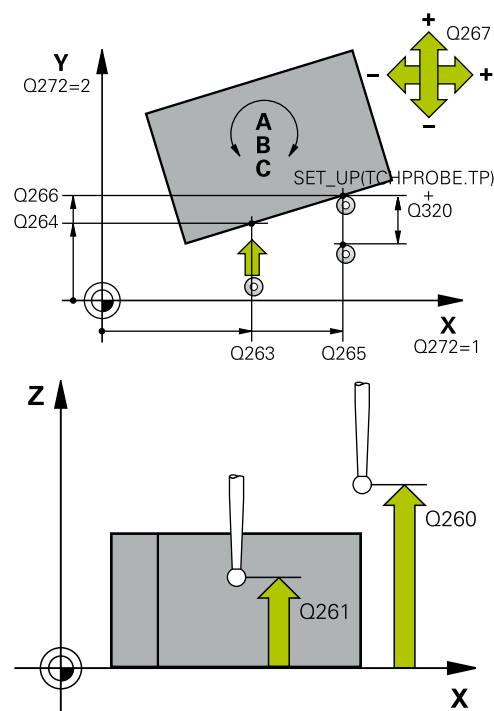
- ▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- ▶ Reset any coordinate transformations beforehand.

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



403

- ▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working plane.  
Input range -99999.9999 to 99999.9999
- ▶ **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q272 Meas. axis (1/2/3, 1=ref. axis)?**: Axis in which the measurement is to be made:  
**1**: Principal axis = measuring axis  
**2**: Secondary axis = measuring axis  
**3**: Touch probe axis = measuring axis
- ▶ **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:  
**-1**: Negative traverse direction  
**+1**: Positive traverse direction
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:  
**0**: Move at measuring height between measuring points  
**1**: Move at clearance height between measuring points



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



- ▶ **Q312 Axis for compensating movement?:**  
Specify the rotary axis in which the control is to compensate the measured misalignment:  
**0:** Automatic mode—the control uses the active kinematics to determine the rotary axis to be aligned. In Automatic mode the first rotary axis of the table (as viewed from the workpiece) is used as compensation axis. This is the recommended setting!  
**4:** Compensate misalignment with rotary axis A  
**5:** Compensate misalignment with rotary axis B  
**6:** Compensate misalignment with rotary axis C
- ▶ **Q337 Set to zero after alignment?:** Define whether the control should set the angle of the aligned rotary axis to 0 in the preset table or in the datum table after the alignment.  
**0:** Do not set the angle of the rotary axis to 0 in the table after alignment  
**1:** Set the angle of the rotary axis to 0 in the table after alignment
- ▶ **Q305 Number in table?** Specify the number of the row in the preset table in which the control is to enter the basic rotation.  
**Q305 = 0:** The rotary axis will be zeroed in 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 is to zero the rotary axis. The control will make an entry in the **OFFSET** column of the preset table.  
**Q305 depends on the following parameters:**  
**Q337 = 0:** Parameter **Q305** is not effective  
**Q337 = 1:** Parameter **Q305** is effective as described above  
**Q312 = 0:** Parameter **Q305** is effective as described above  
**Q312 > 0:** The entry in **Q305** is ignored. The control will make an entry in the **OFFSET** column, in the row of the preset table that was active when the cycle was called  
 Input range: 0 to 99999



- ▶ **Q303 Meas. value transfer (0,1)?:** Specify whether the determined preset is to be saved in the preset table or in the datum table:  
**0:** Write the measured preset as a datum shift to the active datum table. The reference system is the active workpiece coordinate system  
**1:** Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- ▶ **Q380 Ref. angle in ref. axis?:** Angle to which the control is to align the probed straight line. Only effective if the rotary axis is in automatic mode or if C is selected (**Q312** = 0 or 6).  
Input range: 0 to 360.000



## 4.11 ROT. IN C AXIS (Cycle 405, DIN/ISO: G405, option 17)

### Application

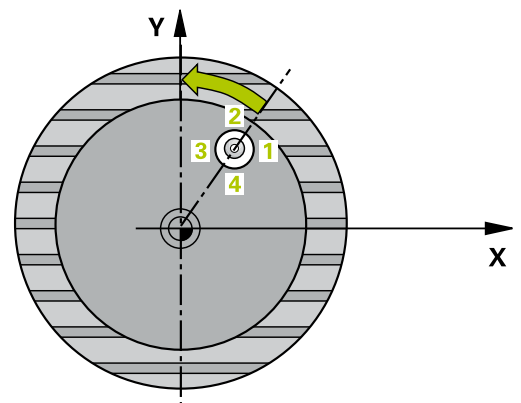
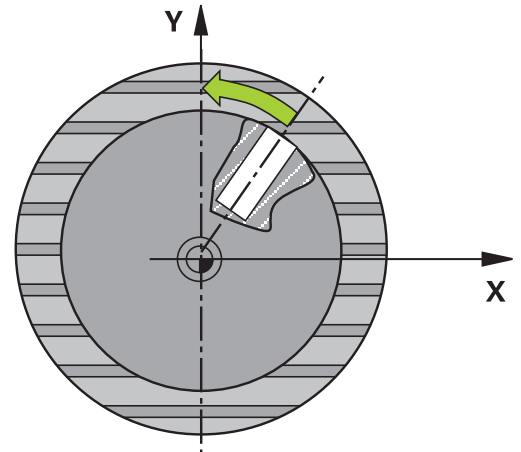
With touch probe cycle **405**, you can measure

- 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 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.

### Cycle run

- 1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic "Executing touch probe cycles", at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET\_UP** column of the touch probe table.
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
- 3 Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times and then positions the touch probe on the hole center measured.
- 5 Finally the control returns the touch probe to the clearance height and aligns the workpiece by rotating the rotary table. The control rotates the rotary table in such a way that the hole center after compensation lies in the direction of the positive Y axis or at the nominal position of the hole center point—both with a vertical and a horizontal touch probe axis. The measured angular offset is also available in parameter **Q150**.





**Please note while programming:****NOTICE****Danger of collision!**

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

- ▶ Make sure that there is no material in the pocket/hole
- ▶ To prevent a collision between the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole).

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

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- ▶ 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.

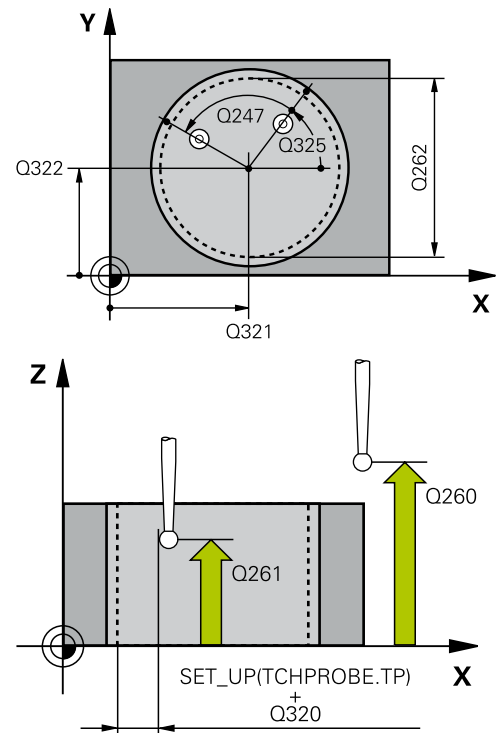
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The smaller the stepping angle, the less accurately the control can calculate the circle center point. Minimum input value: 5°.



## Cycle parameters



- ▶ **Q321 Center in 1st axis?** (absolute): Center of the hole in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q322 Center in 2nd axis?** (absolute): Center of the hole in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point with the positive Y axis. If you program **Q322** not equal to zero 0, then the control aligns the hole center point with the nominal position (angle resulting from the position of the hole center).  
Input range: -99999.9999 to 99999.9999
- ▶ **Q262 Nominal diameter?**: Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large.  
Input range: 0 to 99999.9999
- ▶ **Q325 Starting angle?** (absolute): Angle between the principal axis of the working plane and the first touch point.  
Input range: -360.000 to 360.000
- ▶ **Q247 Intermediate stepping angle?** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range: -120.000 to 120.000
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999



### Example

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



- ▶ **Q301 Move to clearance height (0/1)?:** Definition of how the touch probe is to move between the measuring points:
    - 0:** Move at measuring height between measuring points
    - 1:** Move at clearance height between measuring points
  - ▶ **Q337 Set to zero after alignment?:**
    - 0:** Set the display of the C axis to 0 and write value to **C\_Offset** of the active row of the datum table
    - >0:** Write the measured angular offset to the datum table. Row number = value in **Q337**. If a C-axis shift is registered in the datum table, the control adds the measured angular offset with the correct sign, positive or negative
- Input range: 0 to 2999



## 4.12 SET BASIC ROTATION (Cycle 404, DIN/ISO: G404, option 17)

### 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.

### NOTICE

#### Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- ▶ 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.

### Example

**5 TCH PROBE 404 SET BASIC ROTATION**

**Q307=+0 ;PRESET ROTATION ANG.**

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

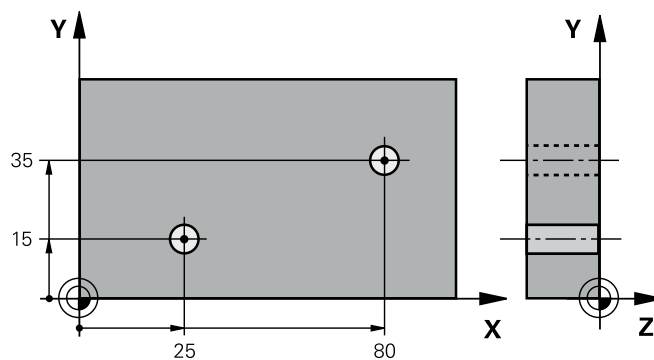
### Cycle parameters



- ▶ **Q307 Preset value for rotation angle:** Angular value to which the basic rotation is to be set. Input range: -360.000 to 360.000
- ▶ **Q305 Preset number in table?:** Enter the number of the preset in the preset table in which the control is to save the determined basic rotation. Input range: -1 to 99999. If you enter **Q305=0** or **Q305=-1**, the control additionally saves the determined basic rotation in the basic rotation menu (**Probing rot**) of the **Manual Operation** mode.
  - 1 = Overwrite and activate the active preset
  - 0 = Copy the active preset to preset row 0, write basic rotation to preset row 0 and activate preset 0
  - >1 = Save the basic rotation to the specified preset. The preset is not activated
 Input range: -1 to +99999



### 4.13 Example: Determining a basic rotation from two holes



0 BEGIN P GM CYC401 MM			
1 TOOL CALL 69 Z			
2 TCH PROBE 401 ROT OF 2 HOLES			
Q268=+25	;1ST CENTER 1ST AXIS	Center of the 1st hole: X coordinate	
Q269=+15	;1ST CENTER 2ND AXIS	Center of the 1st hole: Y coordinate	
Q270=+80	;2ND CENTER 1ST AXIS	Center of the 2nd hole: X coordinate	
Q271=+35	;2ND CENTER 2ND AXIS	Center of the 2nd hole: Y coordinate	
Q261=-5	;MEASURING HEIGHT	Coordinate in the touch probe axis in which the measurement is made	
Q260=+20	;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision	
Q307=+0	;PRESET ROTATION ANG.	Angle of the reference line	
Q305=0	;NUMBER IN TABLE		
Q402=1	;COMPENSATION	Compensate misalignment by rotating the rotary table	
Q337=1	;SET TO ZERO	Set the display to zero after the alignment	
3 CALL PGM 35K47			Call part program
4 END PGM CYC401 MM			



# 5

**Touch Probe  
Cycles: Automatic  
Presetting**



## 5.1 Fundamentals









### Overview

The control offers twelve cycles for automatically finding presets and handling them as follows:





- Setting the determined values directly as display values
- Writing the determined values to the preset table
- Writing the determined values to a datum table



The control must be specifically prepared by the machine tool builder for the use of a 3-D touch probe. HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used. Depending on the setting of the optional **CfgPresetSettings** machine parameter (no. 204600), the control will check during probing whether the position of the rotary axis matches the tilting angles **3-D ROTATION**. If that is not the case, the control displays an error message.

Soft key	Cycle	Page
	PRESET INSIDE RECTANGLE (Cycle 410, DIN/ISO: G410, option 17) <ul style="list-style-type: none"> <li>■ Measuring the inside length and width of a rectangle</li> <li>■ and defining the center as preset</li> </ul>	107
	PRESET OUTS. RECTAN (Cycle 411, DIN/ISO: G411, option 17) <ul style="list-style-type: none"> <li>■ Measuring the outside length and width of a rectangle</li> <li>■ and defining the center as preset</li> </ul>	111
	PRESET INSIDE CIRCLE (Cycle 412, DIN/ISO: G412, option 17) <ul style="list-style-type: none"> <li>■ Measuring any four points on the circle inside</li> <li>■ and defining the center as preset</li> </ul>	115
	PRESET OUTS. CIRCLE (Cycle 413, DIN/ISO: G413, option 17) <ul style="list-style-type: none"> <li>■ Measuring any four points on circle outside</li> <li>■ and defining the center as preset</li> </ul>	120
	PRESET OUTSIDE CORNER (Cycle 414, DIN/ISO: G414, option 17) <ul style="list-style-type: none"> <li>■ Measuring two straight lines on the outside</li> <li>■ and defining the intersection of the lines as preset</li> </ul>	125
	PRESET INSIDE CORNER (Cycle 415, DIN/ISO: G415, option 17) <ul style="list-style-type: none"> <li>■ Measuring two straight lines on the inside</li> <li>■ and defining the intersection of the lines as preset</li> </ul>	130
	PRESET CIRCLE CENTER (Cycle 416, DIN/ISO: G416, option 17) <ul style="list-style-type: none"> <li>■ Measuring any three holes on a circular hole pattern</li> <li>■ and defining the circle center as preset</li> </ul>	135
	PRESET IN TS AXIS (Cycle 417, DIN/ISO: G417, option 17) <ul style="list-style-type: none"> <li>■ Measuring any position in the touch-probe axis</li> <li>■ and defining any position as preset</li> </ul>	140



Soft key	Cycle	Page
	<p>PRESET FROM 4 HOLES (Cycle 418, DIN/ISO: G418, option 17)</p> <ul style="list-style-type: none"> <li>■ Measuring 2 holes on each line crosswise</li> <li>■ and defining the intersection of the lines as preset</li> </ul>	142
	<p>PRESET IN ONE AXIS (Cycle 419, DIN/ISO: G419, option 17)</p> <ul style="list-style-type: none"> <li>■ Measuring any position in a selectable axis</li> <li>■ and defining any position in a selectable axis as preset</li> </ul>	147
	<p>SLOT CENTER PRESET (Cycle 408, DIN/ISO: G408, option 17)</p> <ul style="list-style-type: none"> <li>■ Measuring the width of an inside slot</li> <li>■ and defining slot center as datum</li> </ul>	150
	<p>RIDGE CENTER PRESET (Cycle 409, DIN/ISO: G409, option 17)</p> <ul style="list-style-type: none"> <li>■ Measuring the width of an outside ridge</li> <li>■ and defining the ridge center as preset</li> </ul>	154



## Characteristics common to all touch probe cycles for presetting



You can also run touch probe cycles **408** to **419** during an active rotation (basic rotation or Cycle **10**).

### Preset and touch probe axis

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

Active touch probe axis	Set preset in
Z	X and Y
Y	Z and X
X	Y and Z

### Saving the calculated preset

In all cycles for presetting, you can use input parameters **Q303** and **Q305** to define how the control is to save the calculated preset:

- **Q305 = 0, Q303 = 1:**  
The control copies the active preset to row 0 and activates row 0, deleting simple transformations.
- **Q305 not equal to 0, Q303 = 0:**  
The result is written to the datum table, row **Q305**; **activate the datum with Cycle 7 in the NC program**
- **Q305 not equal to 0, Q303 = 1:**  
The result is written to the preset table, row **Q305**, the machine coordinate system is the reference system (REF coordinates); **activate the preset using Cycle 247 in the NC program**
- **Q305 not equal to 0, Q303 = -1**



This combination can only occur if you

- read in NC programs (containing Cycles **410** to **418**) 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.



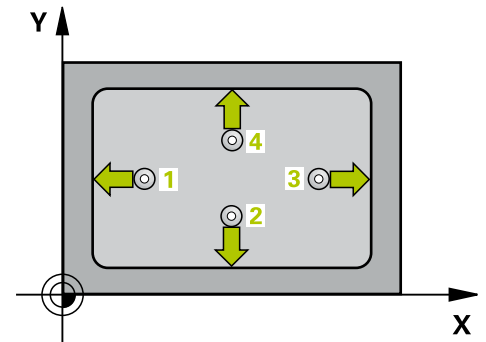
## 5.2 PRESET INSIDE RECTANGLE (Cycle 410, DIN/ISO: G410, option 17)

### 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 at rapid traverse (value from **FMAX** column), using positioning logic "Executing touch probe cycles", at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET\_UP** column of the touch probe table.
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 106)
- 6 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation and saves the actual values in the following Q parameters.



Parameter number	Meaning
<b>Q151</b>	Actual value of center in reference axis
<b>Q152</b>	Actual value of center in minor axis
<b>Q154</b>	Actual value of side length in the reference axis
<b>Q155</b>	Actual value of side length in the minor axis



**Please note while programming:****NOTICE****Danger of collision!**

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- ▶ 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 **low** estimates for the lengths of the first and second sides. If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points.

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

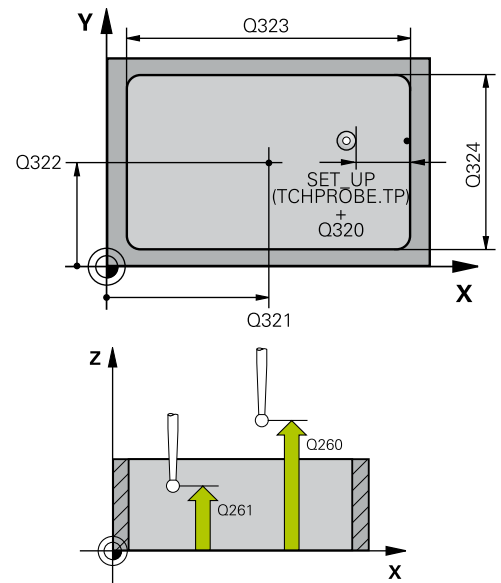
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.



## Cycle parameters



- ▶ **Q321 Center in 1st axis?** (absolute): Center of the pocket in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q322 Center in 2nd axis?** (absolute): Center of the pocket in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q323 First side length?** (incremental): Pocket length, parallel to the principal axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q324 Second side length?** (incremental): Pocket length, parallel to the secondary axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:  
**0**: Move at measuring height between measuring points  
**1**: Move at clearance height between measuring points
- ▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table:  
 If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation  
 If **Q303 = 0**, the control writes the data to the datum table. The datum is not activated automatically  
 Input range: 0 to 9999



### Example

5 TCH PROBE 410 PRESET INSIDE RECTAN	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q323=60	;FIRST SIDE LENGTH
Q324=20	;2ND SIDE LENGTH
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q305=10	;NUMBER IN TABLE
Q331=+0	;PRESET
Q332=+0	;PRESET
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;PRESET



- ▶ **Q331 New preset in reference axis?** (absolute):  
Coordinate in the principal axis at which the control should set the pocket center. Default setting = 0.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q332 New preset in minor axis?** (absolute):  
Coordinate in the secondary axis at which the control should set the pocket center. Default setting = 0.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:  
  - 1**: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 106)
  - 0**: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
  - 1**: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- ▶ **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:  
  - 0**: Do not set the preset in the touch probe axis
  - 1**: Set the preset in the touch probe axis
- ▶ **Q382 Probe TS axis: Coord. 1st axis?** (absolute):  
Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q383 Probe TS axis: Coord. 2nd axis?** (absolute):  
Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q384 Probe TS axis: Coord. 3rd axis?** (absolute):  
Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q333 New preset in TS axis?** (absolute):  
Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0.  
Input range: -99999.9999 to 99999.9999



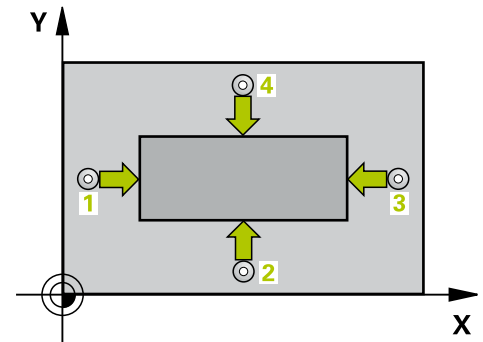
### 5.3 PRESET OUTS. RECTAN (Cycle 411, DIN/ISO: G411, option 17)

#### 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 at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET\_UP** column of the touch probe table.
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 106)
- 6 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation and saves the actual values in the following Q parameters.



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



**Please note while programming:****NOTICE****Danger of collision!**

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

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

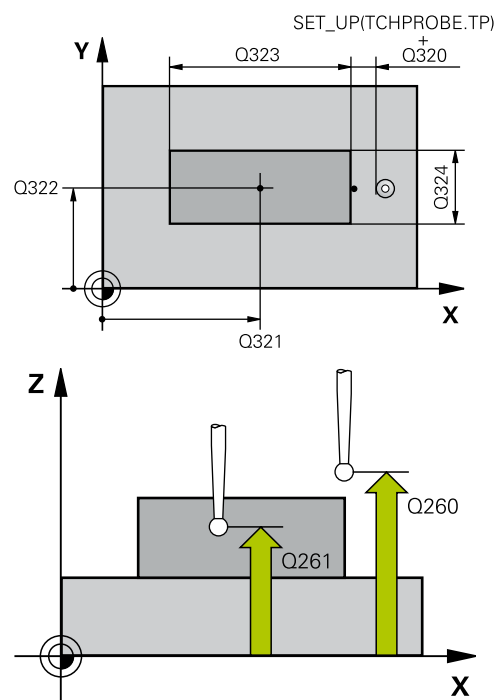
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.



## Cycle parameters



- ▶ **Q321 Center in 1st axis?** (absolute): Center of the stud in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q322 Center in 2nd axis?** (absolute): Center of the stud in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q323 First side length?** (incremental): Stud length, parallel to the principal axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q324 Second side length?** (incremental): Stud length, parallel to the secondary axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:  
**0**: Move at measuring height between measuring points  
**1**: Move at clearance height between measuring points
- ▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table:  
 If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation  
 If **Q303 = 0**, the control writes the data to the datum table. The datum is not activated automatically  
 Input range: 0 to 9999



### Example

#### 5 TCH PROBE 411 PRESET OUTS. RECTAN

Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q323=60	;FIRST SIDE LENGTH
Q324=20	;2ND SIDE LENGTH
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q305=0	;NUMBER IN TABLE
Q331=+0	;PRESET
Q332=+0	;PRESET
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;PRESET



- ▶ **Q331 New preset in reference axis?** (absolute):  
Coordinate in the principal axis at which the control should set the stud center. Default setting = 0.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q332 New preset in minor axis?** (absolute):  
Coordinate in the secondary axis at which the control should set the stud center. Default setting = 0.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:  
  - 1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 106)
  - 0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
  - 1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- ▶ **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:  
  - 0: Do not set the preset in the touch probe axis
  - 1: Set the preset in the touch probe axis
- ▶ **Q382 Probe TS axis: Coord. 1st axis?** (absolute):  
Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q383 Probe TS axis: Coord. 2nd axis?** (absolute):  
Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q384 Probe TS axis: Coord. 3rd axis?** (absolute):  
Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q333 New preset in TS axis?** (absolute):  
Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0.  
Input range: -99999.9999 to 99999.9999



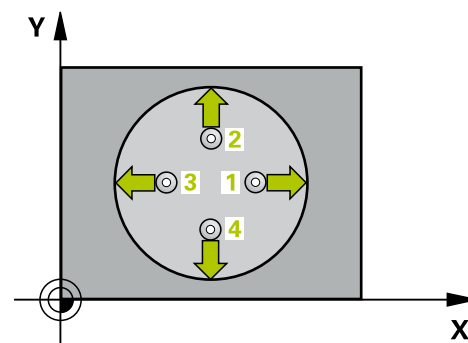
## 5.4 PRESET INSIDE CIRCLE (Cycle 412, DIN/ISO: G412, option 17)

### 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.

### Cycle run

- 1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET\_UP** column of the touch probe table.
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
- 3 Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 Finally, the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 106), and saves the actual values in the Q parameters listed below.
- 6 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.



Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter



**Please note while programming:****NOTICE****Danger of collision!**

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- ▶ 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 the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole). If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points.

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

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The smaller the stepping angle **Q247**, the less accurately the control can calculate the preset. Minimum input value: 5°



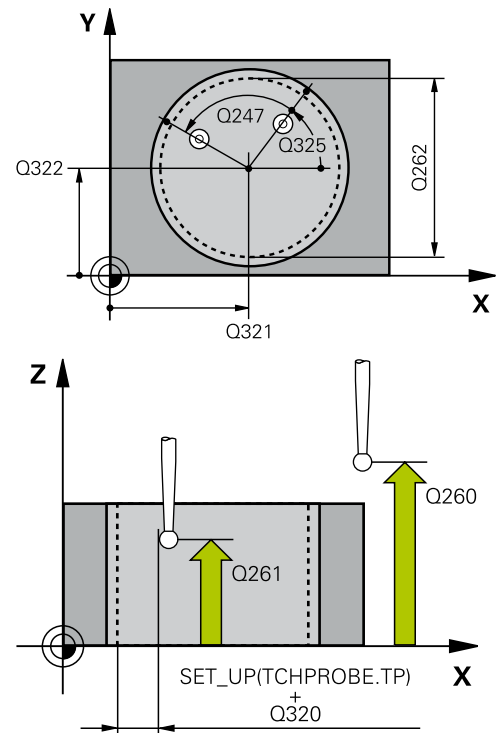
Program the stepping angle to be less than 90°



## Cycle parameters



- ▶ **Q321 Center in 1st axis?** (absolute): Center of the pocket in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q322 Center in 2nd axis?** (absolute): Center of the pocket in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point to the positive Y axis. If you program **Q322** not equal to 0, then the control aligns the hole center point to the nominal position.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q262 Nominal diameter?**: Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large.  
Input range: 0 to 99999.9999
- ▶ **Q325 Starting angle?** (absolute): Angle between the principal axis of the working plane and the first touch point.  
Input range: -360.000 to 360.000
- ▶ **Q247 Intermediate stepping angle?** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°.  
Input range: -120.000 to 120.000
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999



### Example

#### 5 TCH PROBE 412 PRESET INSIDE CIRCLE

Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q262=75	;NOMINAL DIAMETER
Q325=+0	;STARTING ANGLE
Q247=+60	;STEPPING ANGLE
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q305=12	;NUMBER IN TABLE
Q331=+0	;PRESET
Q332=+0	;PRESET



- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:  
**0**: Move at measuring height between measuring points  
**1**: Move at clearance height between measuring points
- ▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table:  
 If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation  
 If **Q303 = 0**, the control writes the data to the datum table. The datum is not activated automatically  
 Input range: 0 to 9999
- ▶ **Q331 New preset in reference axis?** (absolute):  
 Coordinate in the principal axis at which the control should set the pocket center. Default setting = 0.  
 Input range: -99999.9999 to 99999.9999
- ▶ **Q332 New preset in minor axis?** (absolute):  
 Coordinate in the secondary axis at which the control should set the pocket center. Default setting = 0.  
 Input range: -99999.9999 to 99999.9999
- ▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:  
**-1**: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 106)  
**0**: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system  
**1**: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).

<b>Q303=+1</b>	<b>;MEAS. VALUE TRANSFER</b>
<b>Q381=1</b>	<b>;PROBE IN TS AXIS</b>
<b>Q382=+85</b>	<b>;1ST CO. FOR TS AXIS</b>
<b>Q383=+50</b>	<b>;2ND CO. FOR TS AXIS</b>
<b>Q384=+0</b>	<b>;3RD CO. FOR TS AXIS</b>
<b>Q333=+1</b>	<b>;PRESET</b>
<b>Q423=4</b>	<b>;NO. OF PROBE POINTS</b>
<b>Q365=1</b>	<b>;TYPE OF TRAVERSE</b>



- ▶ **Q381 Probe in TS axis? (0/1):** Specify whether the control should also set the preset in the touch probe axis:  
**0:** Do not set the preset in the touch probe axis  
**1:** Set the preset in the touch probe axis
- ▶ **Q382 Probe TS axis: Coord. 1st axis? (absolute):**  
 Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381 = 1**.  
 Input range: -99999.9999 to 99999.9999
- ▶ **Q383 Probe TS axis: Coord. 2nd axis? (absolute):**  
 Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381 = 1**.  
 Input range: -99999.9999 to 99999.9999
- ▶ **Q384 Probe TS axis: Coord. 3rd axis? (absolute):**  
 Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381 = 1**.  
 Input range: -99999.9999 to 99999.9999
- ▶ **Q333 New preset in TS axis? (absolute):**  
 Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0.  
 Input range: -99999.9999 to 99999.9999
- ▶ **Q423 No. probe points in plane (4/3)?:** Specify whether the control should measure the circle with 4 or 3 touch points:  
**4:** Use 4 measuring points (default setting)  
**3:** Use 3 measuring points
- ▶ **Q365 Type of traverse? Line=0/arc=1:** Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" is active (**Q301=1**):  
**0:** Move in a straight line between machining operations  
**1:** Move in a circular arc on the pitch circle diameter between machining operations



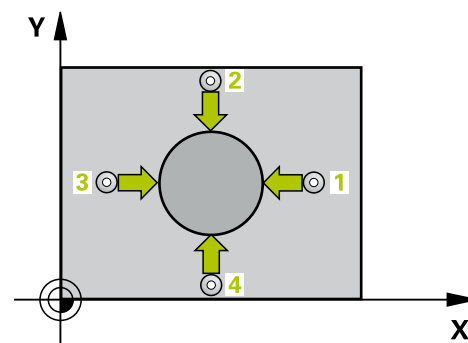
## 5.5 PRESET OUTS. CIRCLE (Cycle 413, DIN/ISO: G413, option 17)

### 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.

### Cycle run

- 1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET\_UP** column of the touch probe table.
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
- 3 Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 Finally, the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 106), and saves the actual values in the Q parameters listed below.
- 6 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.



Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter



**Please note while programming:****NOTICE****Danger of collision!**

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- ▶ 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.
- 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°



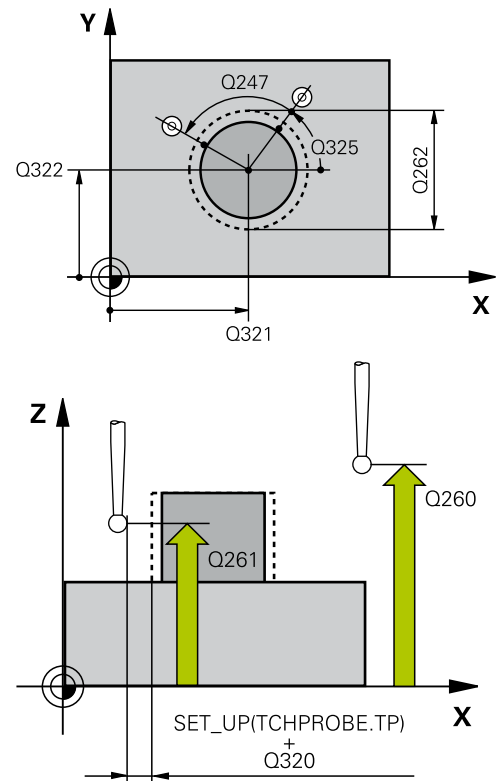
Program the stepping angle to be less than 90°



## Cycle parameters



- ▶ **Q321 Center in 1st axis?** (absolute): Center of the stud in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q322 Center in 2nd axis?** (absolute): Center of the stud in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point to the positive Y axis. If you program **Q322** not equal to 0, then the control aligns the hole center point to the nominal position.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q262 Nominal diameter?**: Approximate diameter of the stud. Enter a value that is more likely to be too large than too small.  
Input range: 0 to 99999.9999
- ▶ **Q325 Starting angle?** (absolute): Angle between the principal axis of the working plane and the first touch point.  
Input range: -360.000 to 360.000
- ▶ **Q247 Intermediate stepping angle?** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range: -120.000 to 120.000
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:  
**0**: Move at measuring height between measuring points  
**1**: Move at clearance height between measuring points



### Example

5 TCH PROBE 413 PRESET OUTS. CIRCLE	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q262=75	;NOMINAL DIAMETER
Q325=+0	;STARTING ANGLE
Q247=+60	;STEPPING ANGLE
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q305=15	;NUMBER IN TABLE
Q331=+0	;PRESET
Q332=+0	;PRESET
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;PRESET
Q423=4	;NO. OF PROBE POINTS
Q365=1	;TYPE OF TRAVERSE



- ▶ **Q305 Number in table?:** Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table:  
If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation  
If **Q303 = 0**, the control writes the data to the datum table. The datum is not activated automatically  
Input range: 0 to 9999
- ▶ **Q331 New preset in reference axis? (absolute):**  
Coordinate in the principal axis at which the control should set the stud center. Default setting = 0.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q332 New preset in minor axis? (absolute):**  
Coordinate in the secondary axis at which the control should set the stud center. Default setting = 0.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q303 Meas. value transfer (0,1)?:** Specify whether the determined preset is to be saved in the datum table or in the preset table:  
-1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 106)  
0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system  
1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- ▶ **Q381 Probe in TS axis? (0/1):** Specify whether the control should also set the preset in the touch probe axis:  
0: Do not set the preset in the touch probe axis  
1: Set the preset in the touch probe axis



- ▶ **Q382 Probe TS axis: Coord. 1st axis?** (absolute):  
Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381 = 1**.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q383 Probe TS axis: Coord. 2nd axis?** (absolute):  
Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381 = 1**.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q384 Probe TS axis: Coord. 3rd axis?** (absolute):  
Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381 = 1**.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q333 New preset in TS axis?** (absolute):  
Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q423 No. probe points in plane (4/3)?**: Specify whether the control should measure the circle with 4 or 3 touch points:  
**4**: Use 4 measuring points (default setting)  
**3**: Use 3 measuring points
- ▶ **Q365 Type of traverse? Line=0/arc=1**: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" is active (**Q301=1**):  
**0**: Move in a straight line between machining operations  
**1**: Move in a circular arc on the pitch circle diameter between machining operations



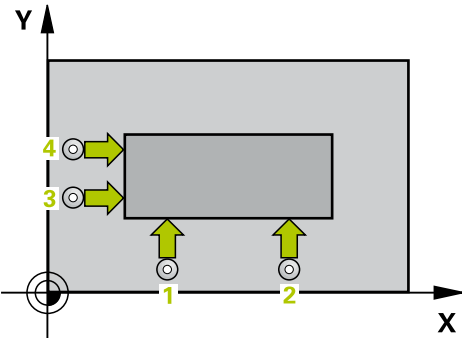
## 5.6 PRESET OUTSIDE CORNER (Cycle 414, DIN/ISO: G414, option 17)

### 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.

### Cycle run

- 1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), at touch point **1** (see figure at right). The control offsets the touch probe by the set-up clearance in the direction opposite the respective traverse direction.
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the 3rd measuring point.
- 3 The touch probe then moves to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 Finally the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 106), and saves the coordinates of the determined corner in the Q parameters listed below.
- 6 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.



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

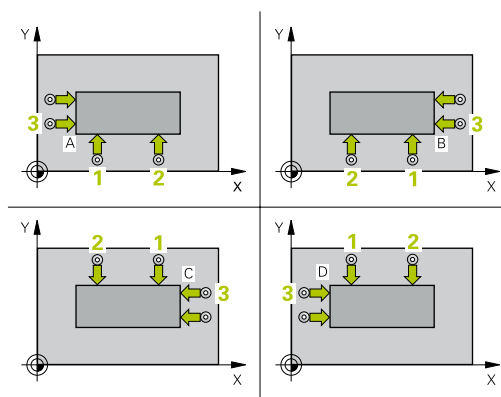
Parameter number	Meaning
Q151	Actual value of corner in reference axis
Q152	Actual value of corner in minor axis



**Please note while programming:****NOTICE****Danger of collision!**

When running touch probe cycles **400 to 499**, no cycles for coordinate transformation must be active.

- ▶ 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.
  - Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
  - By defining the positions of the measuring points **1** and **3**, you also determine the corner at which the control sets the preset (see figure at the right and table below).



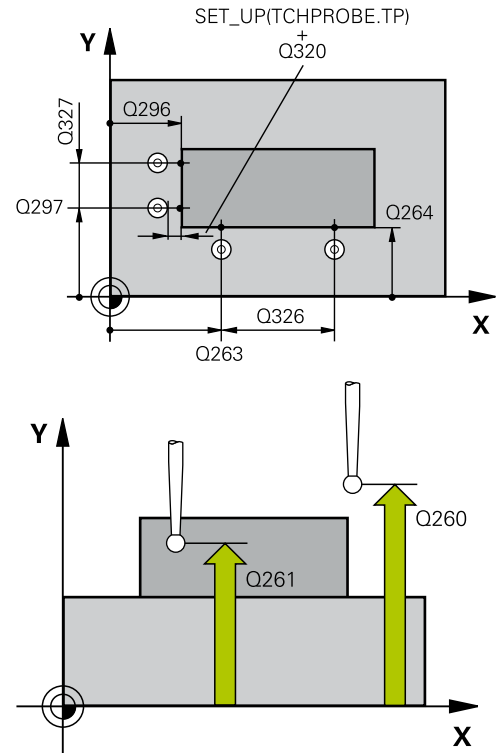
Corner	X coordinate	Y coordinate
A	Point <b>1</b> greater than point <b>3</b>	Point <b>1</b> less than point <b>3</b>
B	Point <b>1</b> less than point <b>3</b>	Point <b>1</b> less than point <b>3</b>
C	Point <b>1</b> less than point <b>3</b>	Point <b>1</b> greater than point <b>3</b>
D	Point <b>1</b> greater than point <b>3</b>	Point <b>1</b> greater than point <b>3</b>



## Cycle parameters



- ▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q326 Spacing in 1st axis?** (incremental): Distance between the first and second measuring points in the principal axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q296 3rd measuring point in 1st axis?** (absolute): Coordinate of the third touch point in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q297 3rd measuring point in 2nd axis?** (absolute): Coordinate of the third touch point in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q327 Spacing in 2nd axis?** (incremental): Distance between third and fourth measuring points in the secondary axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:  
  - 0**: Move at measuring height between measuring points
  - 1**: Move at clearance height between measuring points



### Example

5 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



- ▶ **Q304 Execute basic rotation (0/1)?**: Definition of whether the control should compensate workpiece misalignment with a basic rotation:  
**0**: Do not execute basic rotation  
**1**: Execute basic rotation
- ▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the coordinates of the corner. Depending on **Q303**, the control writes the entry to the preset table or datum table:  
 If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation  
 If **Q303 = 0**, the control writes the data to the datum table. The datum is not activated automatically  
 Input range: 0 to 9999
- ▶ **Q331 New preset in reference axis?** (absolute):  
 Coordinate in the principal axis at which the control should set the corner. Default setting = 0.  
 Input range: -99999.9999 to 99999.9999
- ▶ **Q332 New preset in minor axis?** (absolute):  
 Coordinate in the secondary axis at which the control should set the corner. Default setting = 0.  
 Input range: -99999.9999 to 99999.9999
- ▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:  
**-1**: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 106)  
**0**: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system  
**1**: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- ▶ **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:  
**0**: Do not set the preset in the touch probe axis  
**1**: Set the preset in the touch probe axis

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



- ▶ **Q382 Probe TS axis: Coord. 1st axis?** (absolute):  
Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q383 Probe TS axis: Coord. 2nd axis?** (absolute):  
Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q384 Probe TS axis: Coord. 3rd axis?** (absolute):  
Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q333 New preset in TS axis?** (absolute):  
Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0.  
Input range: -99999.9999 to 99999.9999



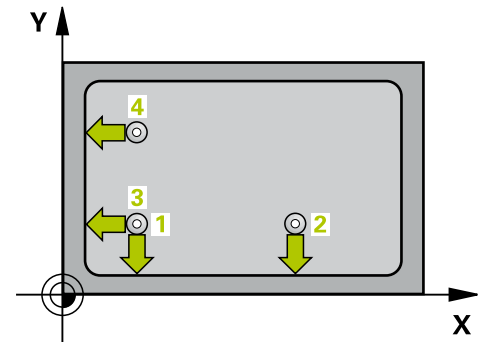
## 5.7 PRESET INSIDE CORNER (Cycle 415, DIN/ISO: G415, option 17)

### 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.

### Cycle run

- 1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic "Executing touch probe cycles", at touch point **1** (see figure at right). The control offsets the touch probe in the principal and secondary axes by the set-up clearance **Q320** + **SET\_UP** + ball-tip radius (in the direction opposite the respective traverse direction)
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The probing direction is derived from the number by which you identify the corner.
- 3 The touch probe moves to the next touch point **2**; the control offsets the touch probe in the secondary axis by the set-up clearance **Q320** + **SET\_UP** + ball-tip radius and then performs the second probing operation
- 4 The control positions the touch probe at touch point **3** (same positioning logic as for the first touch point) and performs the probing operation there
- 5 The touch probe then moves to touch point **4**. The control offsets the touch probe in the principal axis by the set-up clearance **Q320** + **SET\_UP** + ball-tip radius and then performs the fourth probing operation
- 6 Finally, the control retracts the touch probe to the clearance height. It processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 106), and saves the coordinates of the determined corner in the Q parameters listed below.
- 7 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.



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

Parameter number	Meaning
Q151	Actual value of corner in reference axis
Q152	Actual value of corner in minor axis



**Please note while programming:****NOTICE****Danger of collision!**

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

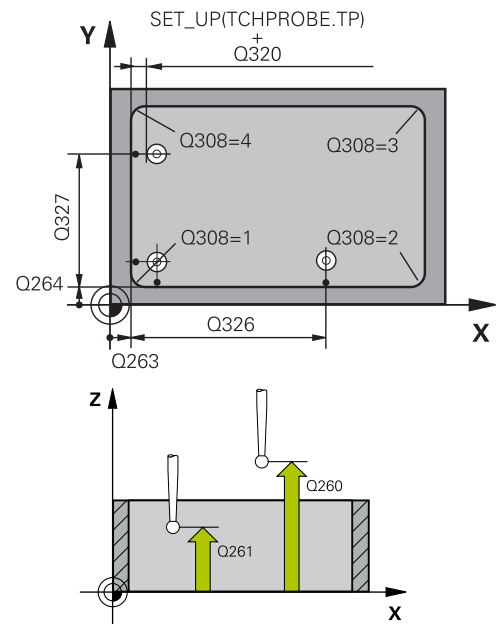
- ▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
  - ▶ Reset any coordinate transformations beforehand.
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
  - Before defining this cycle, you must have programmed a tool call to define the touch probe axis.



## Cycle parameters



- ▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the corner in the principal axis of the working plane  
Input range: -99999.9999 to 99999.9999
- ▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the corner in the secondary axis of the working plane  
Input range: -99999.9999 to 99999.9999
- ▶ **Q326 Spacing in 1st axis?** (incremental): Distance between the corner and second measuring point in the principal axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q327 Spacing in 2nd axis?** (incremental): Distance between the corner and fourth measuring point in the secondary axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q308 Corner? (1/2/3/4):** Number identifying the corner which the control is to set as the preset.  
Input range: 1 to 4
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:  
  - 0**: Move at measuring height between measuring points
  - 1**: Move at clearance height between measuring points
- ▶ **Q304 Execute basic rotation (0/1)?**: Definition of whether the control should compensate workpiece misalignment with a basic rotation:  
  - 0**: Do not execute basic rotation
  - 1**: Execute basic rotation



### Example

5 TCH PROBE 415 PRESET 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



- ▶ **Q305 Number in table?:** Indicate the number of the row in the preset table / datum table in which the control saves the coordinates of the corner. Depending on **Q303**, the control writes the entry to the preset table or datum table:  
If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation  
If **Q303 = 0**, the control writes the data to the datum table. The datum is not activated automatically  
Input range: 0 to 9999
- ▶ **Q331 New preset in reference axis? (absolute):**  
Coordinate in the principal axis at which the control should set the corner. Default setting = 0.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q332 New preset in minor axis? (absolute):**  
Coordinate in the secondary axis at which the control should set the corner. Default setting = 0.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q303 Meas. value transfer (0,1)?:** Specify whether the determined preset is to be saved in the datum table or in the preset table:  
-1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 106)  
0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system  
1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- ▶ **Q381 Probe in TS axis? (0/1):** Specify whether the control should also set the preset in the touch probe axis:  
0: Do not set the preset in the touch probe axis  
1: Set the preset in the touch probe axis



- ▶ **Q382 Probe TS axis: Coord. 1st axis?** (absolute):  
Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q383 Probe TS axis: Coord. 2nd axis?** (absolute):  
Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q384 Probe TS axis: Coord. 3rd axis?** (absolute):  
Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q333 New preset in TS axis?** (absolute):  
Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0.  
Input range: -99999.9999 to 99999.9999



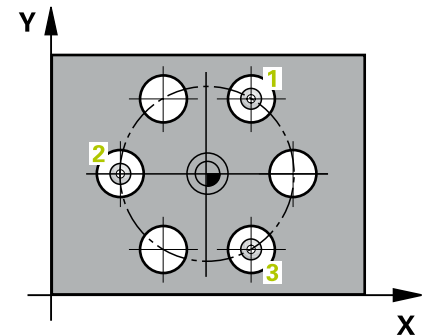
## 5.8 PRESET CIRCLE CENTER (Cycle 416, DIN/ISO: G416, option 17)

### 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 rapid traverse speed (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), to the programmed center point of the first hole **1**.
- 2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
- 3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
- 4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
- 5 The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
- 6 The control moves the touch probe to the entered measuring height and probes four points to determine the third hole center point.
- 7 Finally, the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 106), and saves the actual values in the Q parameters listed below.
- 8 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.



Parameter number	Meaning
<b>Q151</b>	Actual value of center in reference axis
<b>Q152</b>	Actual value of center in minor axis
<b>Q153</b>	Actual value of bolt hole circle diameter



**Please note while programming:****NOTICE****Danger of collision!**

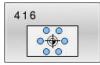
When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- ▶ 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.

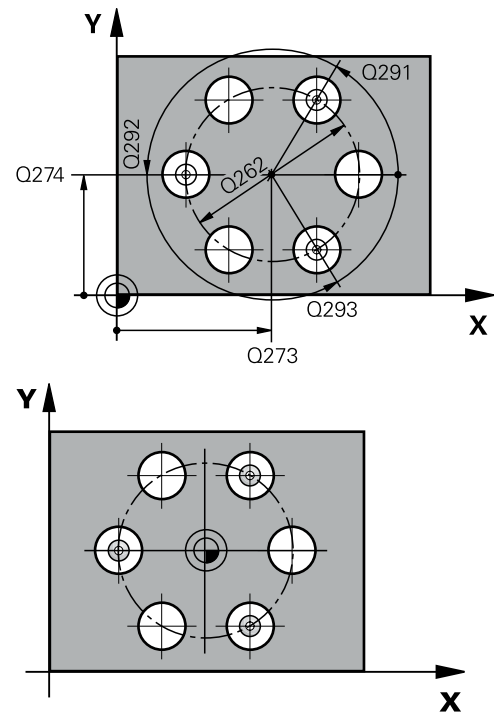
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.



## Cycle parameters



- ▶ **Q273 Center in 1st axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q274 Center in 2nd axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q262 Nominal diameter?**: Enter the approximate bolt hole circle diameter. The smaller the hole diameter, the more exact the nominal diameter must be.  
Input range: -0 to 99999.9999
- ▶ **Q291 Polar coord. angle of 1st hole?** (absolute): Polar coordinate angle of the first hole center in the working plane.  
Input range: -360.0000 to 360.0000
- ▶ **Q292 Polar coord. angle of 2nd hole?** (absolute): Polar coordinate angle of the second hole center in the working plane.  
Input range: -360.0000 to 360.0000
- ▶ **Q293 Polar coord. angle of 3rd hole?** (absolute): Polar coordinate angle of the third hole center in the working plane.  
Input range: -360.0000 to 360.0000
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table:  
If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation  
If **Q303 = 0**, the control writes the data to the datum table. The datum is not activated automatically  
Input range: 0 to 9999
- ▶ **Q331 New preset in reference axis?** (absolute): Coordinate in the principal axis at which the control should set the bolt-hole circle center.  
Default setting = 0.  
Input range: -99999.9999 to 99999.9999



### Example

5 TCH PROBE 416 PRESET CIRCLE CENTER
Q273=+50 ;CENTER IN 1ST AXIS
Q274=+50 ;CENTER IN 2ND AXIS
Q262=90 ;NOMINAL DIAMETER
Q291=+34 ;ANGLE OF 1ST HOLE
Q292=+70 ;ANGLE OF 2ND HOLE
Q293=+210 ;ANGLE OF 3RD HOLE
Q261=-5 ;MEASURING HEIGHT
Q260=+20 ;CLEARANCE HEIGHT
Q305=12 ;NUMBER IN TABLE
Q331=+0 ;PRESET
Q332=+0 ;PRESET
Q303=+1 ;MEAS. VALUE TRANSFER
Q381=1 ;PROBE IN TS AXIS
Q382=+85 ;1ST CO. FOR TS AXIS
Q383=+50 ;2ND CO. FOR TS AXIS
Q384=+0 ;3RD CO. FOR TS AXIS
Q333=+1 ;PRESET
Q320=0 ;SET-UP CLEARANCE



- ▶ **Q332 New preset in minor axis?** (absolute):  
Coordinate in the secondary axis at which the control should set the bolt-hole circle center.  
Default setting = 0.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:  
-1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 106)  
0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system  
1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- ▶ **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:  
0: Do not set the preset in the touch probe axis  
1: Set the preset in the touch probe axis
- ▶ **Q382 Probe TS axis: Coord. 1st axis?** (absolute):  
Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q383 Probe TS axis: Coord. 2nd axis?** (absolute):  
Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
Input range: -99999.9999 to 99999.9999



- ▶ **Q384 Probe TS axis: Coord. 3rd axis?** (absolute):  
Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q333 New preset in TS axis?** (absolute):  
Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Additional distance between the touch point and ball tip.  
**Q320** is added to **SET\_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis.  
Input range: 0 to 99999.9999



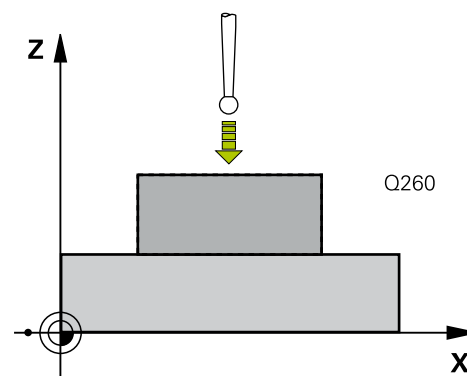
## 5.9 PRESET IN TS AXIS (Cycle 417, DIN/ISO: G417, option 17)

### 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.

### Cycle run

- 1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), at the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the positive direction of the touch probe axis
- 2 Then the touch probe moves in its own axis to the coordinate entered as touch point **1** and measures the actual position with a simple probing movement
- 3 Finally, the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 106), and saves the actual values in the Q parameters listed below.



Parameter number	Meaning
<b>Q160</b>	Actual value of measured point

### Please note while programming:

#### NOTICE

##### Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- 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.

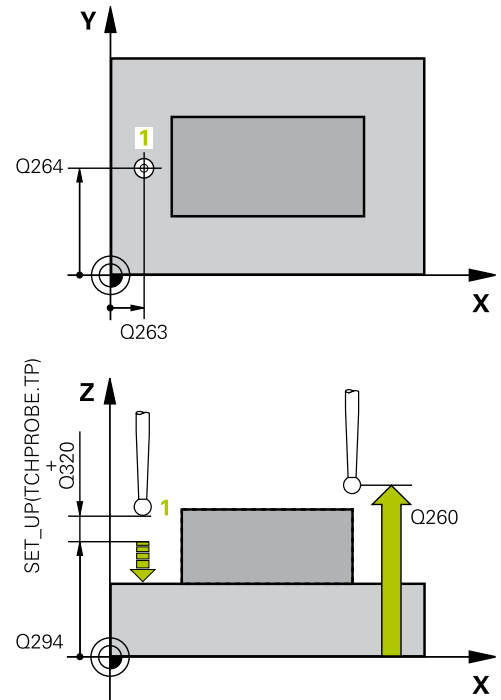
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The control then sets the preset in this axis.



## Cycle parameters



- ▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q294 1st measuring point in 3rd axis?** (absolute): Coordinate of the first touch point in the touch probe axis.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the coordinates.  
If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation  
If **Q303 = 0**, the control writes the data to the datum table. The datum is not activated automatically  
Input range: 0 to 9999
- ▶ **Q333 New preset in TS axis?** (absolute): Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the datum table or in the preset table:  
  - 1**: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 106)
  - 0**: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system
  - 1**: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).



### Example

5 TCH PROBE 417 PRESET IN TS AXIS	
Q263=+25	;1ST POINT 1ST AXIS
Q264=+25	;1ST POINT 2ND AXIS
Q294=+25	;1ST POINT 3RD AXIS
Q320=0	;SET-UP CLEARANCE
Q260=+50	;CLEARANCE HEIGHT
Q305=0	;NUMBER IN TABLE
Q333=+0	;PRESET
Q303=+1	;MEAS. VALUE TRANSFER



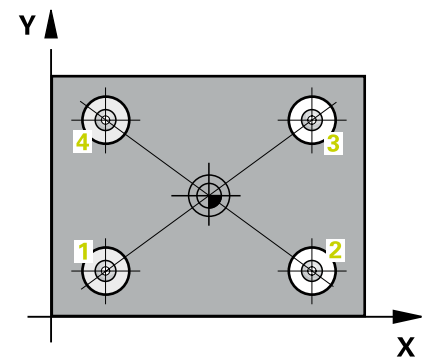
## 5.10 PRESET FROM 4 HOLES (Cycle 418, DIN/ISO: G418, option 17)

### 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 rapid traverse speed (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), to the center point of the first hole **1**.
- 2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
- 3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
- 4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
- 5 The control repeats this step for holes **3** and **4**.
- 6 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 106). The control calculates the preset as the intersection of the lines connecting the centers of holes **1/3** and **2/4** and saves the actual values in the Q parameters listed below.
- 7 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.



Parameter number	Meaning
<b>Q151</b>	Actual value of intersection point in reference axis
<b>Q152</b>	Actual value of intersection point in minor axis



**Please note while programming:****NOTICE****Danger of collision!**

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- ▶ 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.

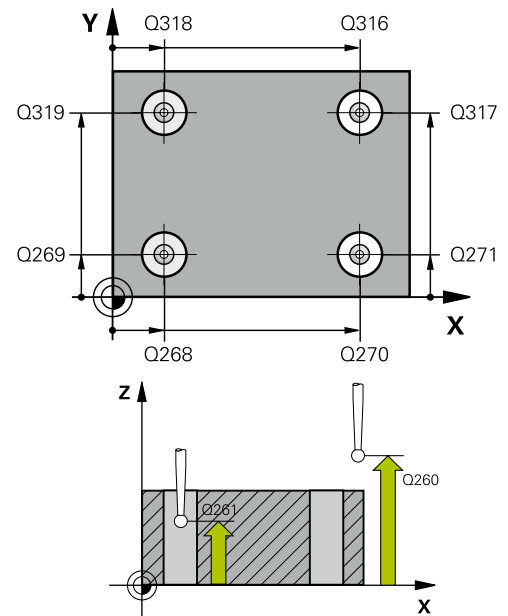
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.



## Cycle parameters



- ▶ **Q268 1st hole: center in 1st axis?** (absolute):  
Center of the first hole in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q269 1st hole: center in 2nd axis?** (absolute):  
Center of the first hole in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q270 2nd hole: center in 1st axis?** (absolute):  
Center of the second hole in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q271 2nd hole: center in 2nd axis?** (absolute):  
Center of the second hole in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q316 3rd hole: Center in 1st axis?** (absolute):  
Center of the third hole in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q317 3rd hole: Center in 2nd axis?** (absolute):  
Center of the third hole in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q318 4th hole: Center in 1st axis?** (absolute):  
Center of the fourth hole in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q319 4th hole: Center in 2nd axis?** (absolute):  
Center of the fourth hole in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q261 Measuring height in probe axis?** (absolute):  
Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999



### Example

5 TCH PROBE 418 PRESET FROM 4 HOLES	
Q268=+20	;1ST CENTER 1ST AXIS
Q269=+25	;1ST CENTER 2ND AXIS
Q270=+150	;2ND CENTER 1ST AXIS
Q271=+25	;2ND CENTER 2ND AXIS
Q316=+150	;3RD CENTER 1ST AXIS
Q317=+85	;3RD CENTER 2ND AXIS
Q318=+22	;4TH CENTER 1ST AXIS
Q319=+80	;4TH CENTER 2ND AXIS
Q261=-5	;MEASURING HEIGHT
Q260=+10	;CLEARANCE HEIGHT
Q305=12	;NUMBER IN TABLE
Q331=+0	;PRESET
Q332=+0	;PRESET
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+0	;PRESET



- ▶ **Q305 Number in table?:** Indicate the number of the row in the preset table / datum table in which the control saves the coordinates of the point of intersection of the connecting lines.  
If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation  
If **Q303 = 0**, the control writes the data to the datum table. The datum is not activated automatically  
Input range: 0 to 9999
- ▶ **Q331 New preset in reference axis? (absolute):**  
Coordinate in the principal axis at which the control should set the intersection of the connecting lines. Default setting = 0.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q332 New preset in minor axis? (absolute):**  
Coordinate in the secondary axis at which the control should set the intersection of the connecting lines. Default setting = 0.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q303 Meas. value transfer (0,1)?:** Specify whether the determined preset is to be saved in the datum table or in the preset table:  
-1: Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 106)  
0: Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system  
1: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- ▶ **Q381 Probe in TS axis? (0/1):** Specify whether the control should also set the preset in the touch probe axis:  
0: Do not set the preset in the touch probe axis  
1: Set the preset in the touch probe axis



- ▶ **Q382 Probe TS axis: Coord. 1st axis?** (absolute):  
Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q383 Probe TS axis: Coord. 2nd axis?** (absolute):  
Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q384 Probe TS axis: Coord. 3rd axis?** (absolute):  
Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q333 New preset in TS axis?** (absolute):  
Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0.  
Input range: -99999.9999 to 99999.9999



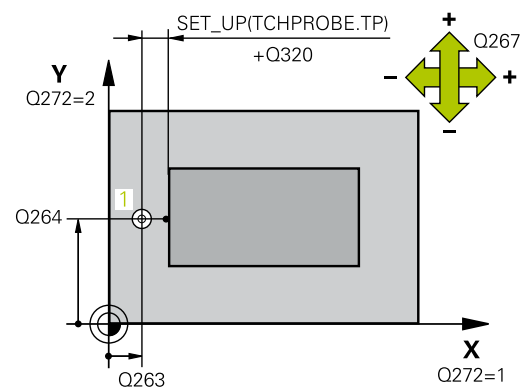
## 5.11 PRESET IN ONE AXIS (Cycle 419, DIN/ISO: G419, option 17)

### 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.

### Cycle run

- 1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), at the programmed touch point **1**. The control offsets the touch probe by the safety clearance in the direction opposite to the programmed direction of probing
- 2 Then the touch probe moves to the programmed measuring height and measures the actual position with a simple probing movement.
- 3 Finally, the control returns the touch probe to clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 106)



### Please note while programming:

#### NOTICE

##### Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- 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.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- If you want to save the preset in several axes in the preset table, you can use Cycle **419** several times in a row. However, you also have to reactivate the preset number after every run of Cycle **419**. If you work with preset 0 as active preset, this process is not required.







- ▶ **Q305 Number in table?:** Indicate the number of the row in the preset table / datum table in which the control saves the coordinates.  
If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation  
If **Q303 = 0**, the control writes the data to the datum table. The datum is not activated automatically  
Input range: 0 to 9999
- ▶ **Q333 New preset? (absolute):** Coordinate at which the control should set the preset. Default setting = 0.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q303 Meas. value transfer (0,1)?:** Specify whether the determined preset is to be saved in the datum table or in the preset table:  
**-1:** Do not use! Is entered by the control when old NC programs are read in (see "Characteristics common to all touch probe cycles for presetting", Page 106)  
**0:** Write the measured preset to the active datum table. The reference system is the active workpiece coordinate system  
**1:** Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).



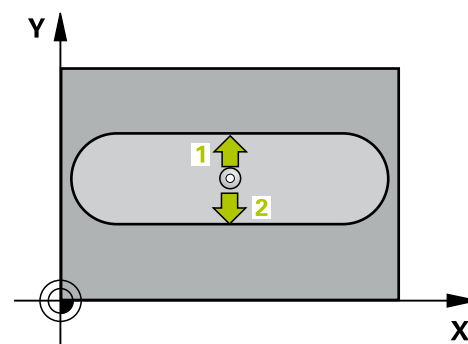
## 5.12 SLOT CENTER PRESET (Cycle 408, DIN/ISO: G408, option 17)

### 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.

### Cycle run

- 1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET\_UP** column of the touch probe table.
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 Finally, the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 106), and saves the actual values in the Q parameters listed below.
- 5 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.



Parameter number	Meaning
<b>Q166</b>	Actual value of measured slot width
<b>Q157</b>	Actual value of the centerline



**Please note while programming:****NOTICE****Danger of collision!**

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- ▶ 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 **low** estimate for the slot width. If the slot width and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the slot. In this case, the touch probe does not return to the clearance height between the two measuring points.

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

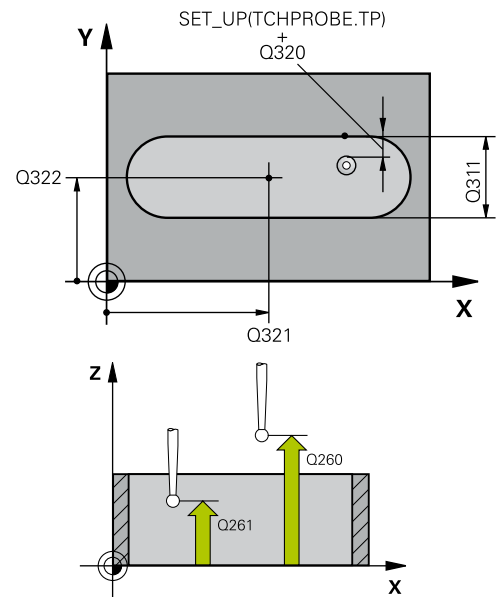
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.



## Cycle parameters



- ▶ **Q321 Center in 1st axis?** (absolute): Center of the slot in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q322 Center in 2nd axis?** (absolute): Center of the slot in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q311 Width of slot?** (incremental): Width of the slot, regardless of its position in the working plane.  
Input range 0 to 99999.9999
- ▶ **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:  
1: Principal axis = measuring axis  
2: Secondary axis = measuring axis
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:  
0: Move at measuring height between measuring points  
1: Move at clearance height between measuring points
- ▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table:  
If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation  
If **Q303 = 0**, the control writes the data to the datum table. The datum is not activated automatically  
Input range: 0 to 9999



### Example

5 TCH PROBE 408 SLOT CENTER PRESET	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q311=25	;SLOT WIDTH
Q272=1	;MEASURING AXIS
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q305=10	;NUMBER IN TABLE
Q405=+0	;PRESET
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;PRESET



- ▶ **Q405 New preset?** (absolute): Coordinate in the measuring axis at which the control should set the calculated slot center. Default setting = 0.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the preset table or in the datum table:  
**0**: Write the measured preset as a datum shift to the active datum table. The reference system is the active workpiece coordinate system  
**1**: Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- ▶ **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:  
**0**: Do not set the preset in the touch probe axis  
**1**: Set the preset in the touch probe axis
- ▶ **Q382 Probe TS axis: Coord. 1st axis?** (absolute):  
 Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
 Input range: -99999.9999 to 99999.9999
- ▶ **Q383 Probe TS axis: Coord. 2nd axis?** (absolute):  
 Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
 Input range: -99999.9999 to 99999.9999
- ▶ **Q384 Probe TS axis: Coord. 3rd axis?** (absolute):  
 Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.  
 Input range: -99999.9999 to 99999.9999
- ▶ **Q333 New preset in TS axis?** (absolute):  
 Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0.  
 Input range: -99999.9999 to 99999.9999



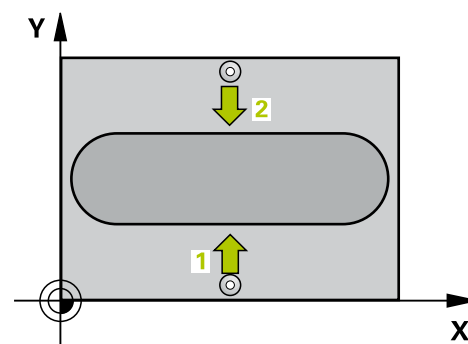
### 5.13 RIDGE CENTER PRESET (Cycle 409, DIN/ISO: G409, option 17)

#### 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.

#### Cycle run

- 1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET\_UP** column of the touch probe table.
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves at clearance height to the next touch point **2** and probes it.
- 4 Finally, the control returns the touch probe to the clearance height and processes the determined preset depending on cycle parameters **Q303** and **Q305** (see "Characteristics common to all touch probe cycles for presetting", Page 106), and saves the actual values in the Q parameters listed below.
- 5 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.



Parameter number	Meaning
<b>Q166</b>	Actual value of measured ridge width
<b>Q157</b>	Actual value of the centerline



**Please note while programming:****NOTICE****Danger of collision!**

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- ▶ 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.

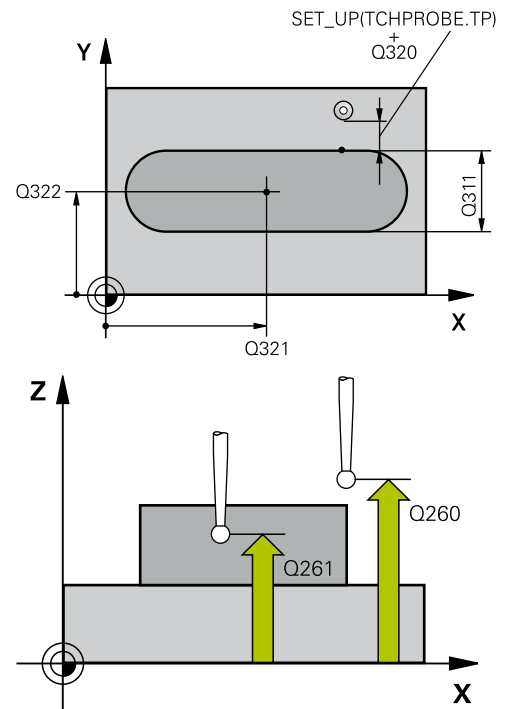
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.



## Cycle parameters



- ▶ **Q321 Center in 1st axis?** (absolute): Center of the ridge in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q322 Center in 2nd axis?** (absolute): Center of the ridge in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q311 Ridge width?** (incremental): Width of the ridge, regardless of its position in the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:  
1: Principal axis = measuring axis  
2: Secondary axis = measuring axis
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the center point coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table:  
If **Q303 = 1**, then the control writes to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the control writes the entry to the corresponding row of the preset table without automatic activation  
If **Q303 = 0**, the control writes the data to the datum table. The datum is not activated automatically  
Input range: 0 to 9999
- ▶ **Q405 New preset?** (absolute): Coordinate in the measuring axis at which the control should set the calculated ridge center. Default setting = 0.  
Input range: -99999.9999 to 99999.9999



### Example

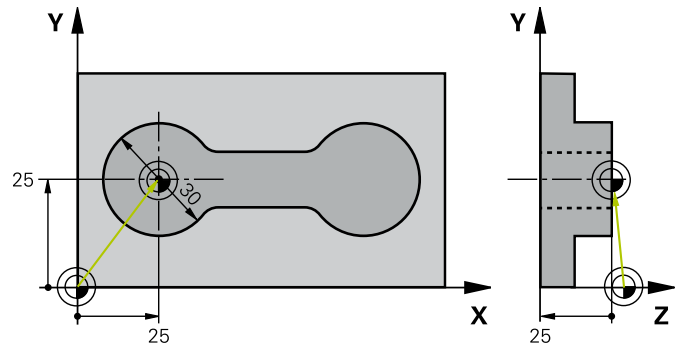
5 TCH PROBE 409 RIDGE CENTER PRESET	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q311=25	;RIDGE WIDTH
Q272=1	;MEASURING AXIS
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q305=10	;NUMBER IN TABLE
Q405=+0	;PRESET
Q303=+1	;MEAS. VALUE TRANSFER
Q381=1	;PROBE IN TS AXIS
Q382=+85	;1ST CO. FOR TS AXIS
Q383=+50	;2ND CO. FOR TS AXIS
Q384=+0	;3RD CO. FOR TS AXIS
Q333=+1	;PRESET



- ▶ **Q303 Meas. value transfer (0,1)?:** Specify whether the determined preset is to be saved in the preset table or in the datum table:  
**0:** Write the measured preset as a datum shift to the active datum table. The reference system is the active workpiece coordinate system  
**1:** Write the measured preset to the preset table. The reference system is the machine coordinate system (REF system).
- ▶ **Q381 Probe in TS axis? (0/1):** Specify whether the control should also set the preset in the touch probe axis:  
**0:** Do not set the preset in the touch probe axis  
**1:** Set the preset in the touch probe axis
- ▶ **Q382 Probe TS axis: Coord. 1st axis? (absolute):** Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381 = 1**.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q383 Probe TS axis: Coord. 2nd axis? (absolute):** Coordinate of the probe point in the secondary axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381 = 1**.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q384 Probe TS axis: Coord. 3rd axis? (absolute):** Coordinate of the touch point in the touch probe axis, at which point the preset is to be set in the touch probe axis. Only effective if **Q381 = 1**.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q333 New preset in TS axis? (absolute):** Coordinate in the touch probe axis at which the control should set the preset. Default setting = 0.  
Input range: -99999.9999 to 99999.9999



### 5.14 Example: Presetting in center of a circular segment and on top surface of workpiece

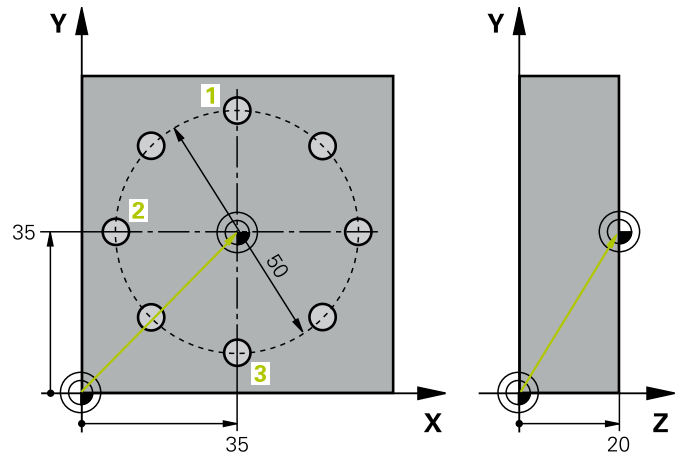


0 BEGIN PGM CYC413 MM		
1 TOOL CALL 69 Z		
2 TCH PROBE 413 PRESET OUTS. CIRCLE		
Q321=+25	;CENTER IN 1ST AXIS	Center of circle: X coordinate
Q322=+25	;CENTER IN 2ND AXIS	Center of circle: Y coordinate
Q262=30	;NOMINAL DIAMETER	Circle diameter
Q325=+90	;STARTING ANGLE	Polar coordinate angle for 1st touch point
Q247=+45	;STEPPING ANGLE	Stepping angle for calculating the starting points 2 to 4
Q261=-5	;MEASURING HEIGHT	Coordinate in the touch probe axis in which the measurement is made
Q320=2	;SET-UP CLEARANCE	Safety clearance in addition to SET_UP column
Q260=+10	;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q301=0	;MOVE TO CLEARANCE	Do not move to clearance height between measuring points
Q305=0	;NUMBER IN TABLE	Set display
Q331=+0	;PRESET	Set the display in X to 0
Q332=+10	;PRESET	Set the display in Y to 10
Q303=+0	;MEAS. VALUE TRANSFER	Without function, since display is to be set
Q381=1	;PROBE IN TS AXIS	Also set preset in the touch probe axis
Q382=+25	;1ST CO. FOR TS AXIS	X coordinate of touch point
Q383=+25	;2ND CO. FOR TS AXIS	Y coordinate of touch point
Q384=+25	;3RD CO. FOR TS AXIS	Z coordinate of touch point
Q333=+0	;PRESET	Set the display in Z to 0
Q423=4	;NO. OF PROBE POINTS	Measure circle with 4 probes
Q365=0	;TYPE OF TRAVERSE	Move on circular path between measuring points
3 CALL PGM 35K47		Call part program
4 END PGM CYC413 MM		



## 5.15 Example: Presetting on top surface of workpiece and in center of a bolt hole circle

The control is to write the measured bolt-hole circle center to the preset table so that it may be used at a later time.



<b>0 BEGIN PGM CYC416 MM</b>	
<b>1 TOOL CALL 69 Z</b>	
<b>2 TCH POBE 417 PRESET IN TS AXIS</b>	Cycle definition for presetting in the touch probe axis
Q263=+7.5 ;1ST POINT 1ST AXIS	Touch point: X coordinate
Q264=+7.5 ;1ST POINT 2ND AXIS	Touch point: Y coordinate
Q294=+25 ;1ST POINT 3RD AXIS	Touch point: Z coordinate
Q320=0 ;SET-UP CLEARANCE	Safety clearance in addition to SET_UP column
Q260=+50 ;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q305=1 ;NUMBER IN TABLE	Write Z coordinate in line 1
Q333=+0 ;PRESET	Set touch-probe axis to 0
Q303=+1 ;MEAS. VALUE TRANSFER	In the preset table PRESET.PR, save the calculated preset referenced to the machine-based coordinate system (REF system)
<b>3 TCH PROBE 416 PRESET CIRCLE CENTER</b>	
Q273=+35 ;CENTER IN 1ST AXIS	Center of the bolt hole circle: X coordinate
Q274=+35 ;CENTER IN 2ND AXIS	Center of the bolt hole circle: Y coordinate
Q262=50 ;NOMINAL DIAMETER	Diameter of the bolt hole circle
Q291=+90 ;ANGLE OF 1ST HOLE	Polar coordinate angle for 1st hole center <b>1</b>
Q292=+180 ;ANGLE OF 2ND HOLE	Polar coordinate angle for 2nd hole center <b>2</b>
Q293=+270 ;ANGLE OF 3RD HOLE	Polar coordinate angle for 3rd hole center <b>3</b>
Q261=+15 ;MEASURING HEIGHT	Coordinate in the touch probe axis in which the measurement is made
Q260=+10 ;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q305=1 ;NUMBER IN TABLE	Write center of bolt hole circle (X and Y) to row 1
Q331=+0 ;PRESET	
Q332=+0 ;PRESET	



<b>Q303=+1</b>	<b>;MEAS. VALUE TRANSFER</b>	In the preset table PRESET.PR, save the calculated preset referenced to the machine-based coordinate system (REF system)
<b>Q381=0</b>	<b>;PROBE IN TS AXIS</b>	Do not set a preset in the touch probe axis
<b>Q382=+0</b>	<b>;1ST CO. FOR TS AXIS</b>	No function
<b>Q383=+0</b>	<b>;2ND CO. FOR TS AXIS</b>	No function
<b>Q384=+0</b>	<b>;3RD CO. FOR TS AXIS</b>	No function
<b>Q333=+0</b>	<b>;PRESET</b>	No function
<b>Q320=0</b>	<b>;SET-UP CLEARANCE.</b>	Safety clearance in addition to SET_UP column
<b>4 CYCL DEF 247 PRESETTING</b>		Activate new preset with Cycle 247
<b>Q339=1</b>	<b>;PRESET NUMBER</b>	
<b>6 CALL PGM 35KLZ</b>		Call part program
<b>7 END PGM CYC416 MM</b>		



# 6

**Touch Probe  
Cycles: Automatic  
Workpiece  
Inspection**



## 6.1 Fundamentals

### Overview



The control must be specifically prepared by the machine tool builder for the use of a 3-D touch probe. HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.








### NOTICE

#### Danger of collision!






When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- ▶ 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
	REFERENCE PLANE (Cycle 0, DIN/ISO: G55, option 17) ■ Measuring a coordinate in a selectable axis	168
	POLAR PRESET (Cycle 1, option 17) ■ Measuring a point ■ Probing direction via angle	169
	MEASURE ANGLE (Cycle 420, DIN/ISO: G420, option 17) ■ Measuring an angle in the working plane	170
	MEASURE HOLE (Circle 421, DIN/ISO: G421, option 17) ■ Measuring the position of a hole ■ Measuring the diameter of a hole ■ Nominal-to-actual value comparison, if applicable	173
	MEASURE CIRCLE OUTSIDE (Cycle 422, DIN/ISO: G422, option 17) ■ Measuring the position of a circular stud ■ Measuring the diameter of a circular stud ■ Nominal-to-actual value comparison, if applicable	177
	MEASURE RECTANGLE INSIDE (Cycle 423, DIN/ISO: G423, option 17) ■ Measuring the position of a rectangular pocket ■ Measuring the length and width of a rectangular pocket ■ Nominal-to-actual value comparison, if applicable	181
	MEASURE RECTANGLE OUTSIDE (Cycle 424, DIN/ISO: G424, option 17) ■ Measuring the position of a rectangular stud ■ Measuring the length and width of a rectangular stud ■ Nominal-to-actual value comparison, if applicable	185



Soft key	Cycle	Page
	MEASURE INSIDE WIDTH (Cycle 425, DIN/ISO: G425, option 17) <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>	188
	MEASURE RIDGE WIDTH (Cycle 426, DIN/ISO: G426, option 17) <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>	191
	MEASURE COORDINATE (Cycle 427, DIN/ISO: G427, option 17) <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>	194
	MEAS. BOLT HOLE (Cycle 430, DIN/ISO: G430, option 17) <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>	197
	MEASURE PLANE (Cycle 431, DIN/ISO: G431, option 17) <ul style="list-style-type: none"> <li>■ Finding the angle of a plane by measuring three points</li> </ul>	200

## Recording the results of measurement

For all cycles in which you automatically measure workpieces (with the exception of Cycles **0** and **1**), you can have the control record the measurement results in a log. In the respective probing cycle you can define if the control is to

- Save the measuring log to a file
- Interrupt program run and display the measuring log on the screen
- Create no measuring log

If you want to save the measuring log to a file, the control by default saves the data as an ASCII file. The control will save the file in the directory that also contains the associated NC program.



Use the HEIDENHAIN data transfer software TNCRemo if you wish to output the measuring log over the data interface.



Example: Measuring log for touch probe cycle **421**:

### Measuring log for Probing Cycle 421 Hole Measuring

Date: 30-06-2005

Time: 6:55:04

Measuring program: TNC:\GEH35712\CHECK1.H

Nominal values:

Center in reference axis:	50.0000
Center in minor axis:	65.0000
Diameter:	12.0000

Given limit values:

Maximum limit for center in reference axis:	50.1000
Minimum limit for center in reference axis:	49.9000
Maximum limit for center in minor axis:	65.1000

Minimum limit for center in minor axis:	64.9000
Maximum dimension for hole:	12.0450
Minimum dimension for hole:	12.0000

Actual values:

Center in reference axis:	50.0810
Center in minor axis:	64.9530
Diameter:	12.0259

Deviations:

Center in reference axis:	0.0810
Center in minor axis:	-0.0470
Diameter:	0.0259

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

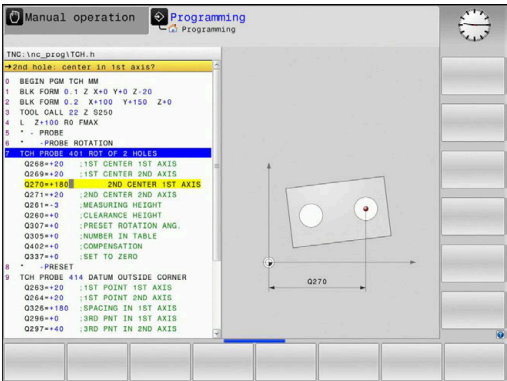
**End of measuring log**



Measurement results in Q parameters

The control saves the measurement results of the respective probing cycle in the globally effective Q parameters **Q150** to **Q160**. Deviations from the nominal values are saved in parameters **Q161** to **Q166**. Note the table of result parameters listed with every cycle description.

During cycle definition, the control also shows the result parameters for the respective cycle in a help graphic (see figure at right). The highlighted result parameter belongs to that input parameter.




Classification of results

For some cycles you can inquire the status of measuring results through the globally effective Q parameters **Q180** to **Q182**.

Class of results	Parameter value
Measurement results are within tolerance	<b>Q180</b> = 1
Rework is required	<b>Q181</b> = 1
Scrap	<b>Q182</b> = 1

The control sets the rework or scrap marker as soon as one of the measuring values is out of tolerance. To determine which of the measuring results is out of tolerance, check the measuring log, or compare the respective measuring results (**Q150** to **Q160**) with their limit values.

In Cycle **427** the control assumes by default that you are measuring an outside dimension (stud). However, you can correct the status of the measurement by entering the correct maximum and minimum dimension together with the probing direction.



The control also sets the status markers if you have not defined any tolerance values or maximum/minimum dimensions.

Tolerance monitoring

With most cycles for workpiece inspection, you can have the control perform tolerance monitoring. This requires that you define the necessary limit values during cycle definition. If you do not wish to monitor for tolerances, simply leave the default value 0 for this parameter set this parameter unchanged.



## Tool monitoring

With some cycles for workpiece inspection, you can have the control perform tool monitoring. The control then monitors whether

- the tool radius should be compensated due to the deviations from the nominal value (values in **Q16x**)
- the deviations from the nominal value (values in **Q16x**) are greater than the tool breakage tolerance.

## Tool compensation

### 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"
- **FN18: SYSREAD Q0 = ID990 NR10 IDX0**; specify the number of the **QS** parameter in **IDX**
- **Q0** = **Q0** + 0.2; add the index of the basic tool number
- In the cycle: **Q330** = **Q0**; use the indexed tool



### Tool breakage monitoring

#### 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.



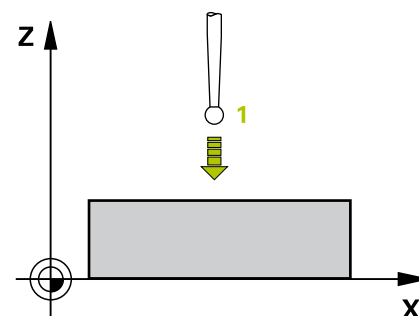
## 6.2 REFERENCE PLANE (Cycle 0, DIN/ISO: G55, option 17)

### Application

The touch probe cycle measures any position on the workpiece in a selectable axis direction.

### Cycle run

- 1 In a 3-D movement, the touch probe moves at rapid traverse (value from the **FMAX** column) to the pre-position **1** programmed in the cycle.
- 2 Next, the touch probe performs probing at the probing feed rate (**F** column). The probing direction must be defined in the cycle.
- 3 After the control has saved the position, the probe retracts to the starting point and saves the measured coordinate in a Q parameter. In addition, the control stores the coordinates of the position of the touch probe at the time of the triggering signal in parameters **Q115** to **Q119**. For the values in these parameters the control does not account for the stylus length and radius.



### Please note while programming:

#### NOTICE

#### Danger of collision!

The control moves the touch probe in a 3-D movement at rapid traverse to the pre-position programmed in the cycle. Depending on the previous position of the tool, there is danger of collision!

- Pre-position in order to avoid a collision when the programmed pre-positioning point is approached

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

### Cycle parameters



- **Parameter number for result?:** Enter the number of the Q parameter to which you want to assign the coordinate.  
Input range: 0 to 1999
- **Probing axis/probing direction?:** Select the probing axis with the axis key or the alphabetic keyboard, entering the algebraic sign for the probing direction. Confirm with the **ENT** key.  
Input range: All NC axes
- **Position value?:** Use the axis keys or the alphabetic keyboard to enter all coordinates for pre-positioning of the touch probe.  
Input range: -99999.9999 to 99999.9999
- To confirm your input, press the **ENT** key.

### Example

67 TCH PROBE 0.0 REF. PLANE Q5 X-

68 TCH PROBE 0.1 X+5 Y+0 Z-5



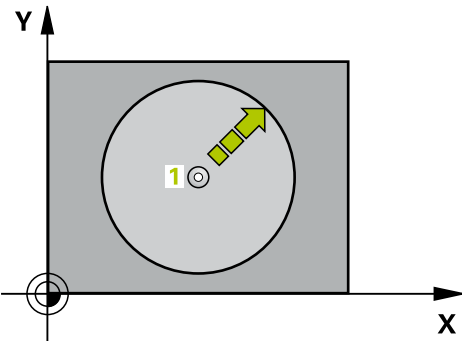
### 6.3 POLAR PRESET (Cycle 1, option 17)

#### Application

Touch probe cycle **1** measures any position on the workpiece in any probing direction.

#### Cycle run

- 1 In a 3-D movement, the touch probe moves at rapid traverse (value from the **FMAX** column) to the pre-position **1** programmed in the cycle.
- 2 Next, the touch probe performs probing at the probing feed rate (**F** column). During probing, the control moves the touch probe simultaneously in two axes (depending on the probing angle). Use polar angles to define the probing direction in the cycle.
- 3 After the control has saved the position, the touch probe returns to the starting point. The control stores the coordinates of the position of the touch probe at the time of the triggering signal in parameters **Q115** to **Q119**



#### Please note while programming:

**NOTICE**

**Danger of collision!**  
The control moves the touch probe in a 3-D movement at rapid traverse to the pre-position programmed in the cycle. Depending on the previous position of the tool, there is danger of collision!  
► Pre-position in order to avoid a collision when the programmed pre-positioning point is approached

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



- **Probing axis?:** Specify the probing axis with the axis key or the alphabetic keyboard. Confirm with the **ENT** key.  
Input range: **X, Y** or **Z**
- **Probing angle?:** Angle, measured from the probing axis, at which the touch probe is to move.  
Input range: -180.0000 to 180.0000
- **Position value?:** Use the axis keys or the alphabetic keyboard to enter all coordinates for pre-positioning of the touch probe.  
Input range: -99999.9999 to 99999.9999
- To confirm your input, press the **ENT** key.

#### Example

67 TCH PROBE 1.0 POLAR PRESET
68 TCH PROBE 1.1 X ANGLE: +30
69 TCH PROBE 1.2 X+5 Y+0 Z-5



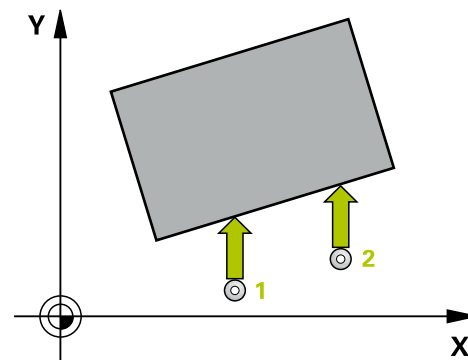
## 6.4 MEASURE ANGLE (Cycle 420, DIN/ISO: G420, option 17)

### Application

Touch probe cycle **420** measures the angle that any straight line on the workpiece forms with the reference axis of the working plane.

### Cycle run

- 1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), at the programmed touch point **1**. The sum of **Q320**, **SET\_UP** and the ball-tip radius is taken into account for probe movements in any probing direction. When the probe movement starts, the center of the ball tip will be offset by this sum in the direction opposite the probing direction.
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 The touch probe then moves to the next touch point **2** and probes again.
- 4 The control returns the touch probe to the clearance height and saves the measured angle in the following Q parameter:



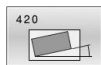
Parameter number	Meaning
<b>Q150</b>	The measured angle is referenced to the reference axis of the machining plane.

### Please note while programming:

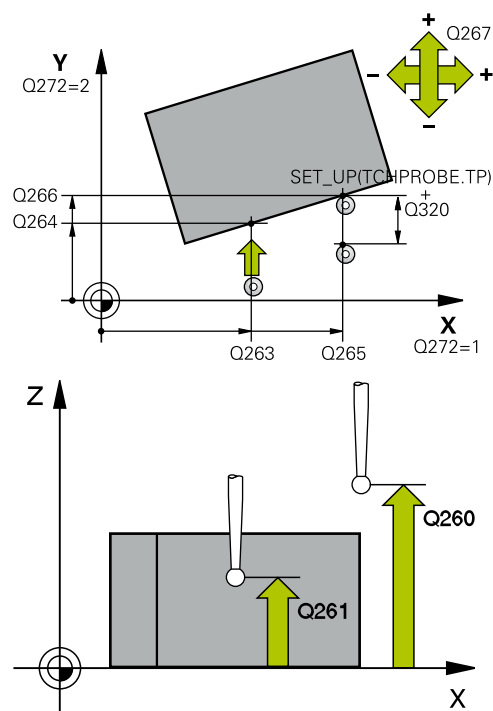
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- If touch probe axis = measuring axis, you can measure the angle in the direction of the A axis or B axis:
  - If you want to measure the angle in the direction of the A axis, set **Q263** equal to **Q265** and **Q264** unequal to **Q266**.
  - If you want to measure the angle in the direction of the B axis, set **Q263** not equal to **Q265** and **Q264** equal to **Q266**.



## Cycle parameters



- ▶ **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q272 Meas. axis (1/2/3, 1=ref. axis)?**: Axis in which the measurement is to be made:
  - 1: Principal axis = measuring axis
  - 2: Secondary axis = measuring axis
  - 3: Touch probe axis = measuring axis
- ▶ **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
  - 1: Negative traverse direction
  - +1: Positive traverse direction
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Additional distance between measuring point and ball tip. The touch probe movement will start with an offset of the sum of **Q320**, **SET\_UP**, and the ball-tip radius, even when probing in the tool axis direction.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999



### Example

5 TCH PROBE 420 MEASURE ANGLE	
Q263=+10	;1ST POINT 1ST AXIS
Q264=+10	;1ST POINT 2ND AXIS
Q265=+15	;2ND PNT IN 1ST AXIS
Q266=+95	;2ND POINT 2ND AXIS
Q272=1	;MEASURING AXIS
Q267=-1	;TRAVERSE DIRECTION
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+10	;CLEARANCE HEIGHT
Q301=1	;MOVE TO CLEARANCE
Q281=1	;MEASURING LOG



- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  - 0**: Move at measuring height between measuring points
  - 1**: Move at clearance height between measuring points
- ▶ **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  - 0**: Do not create a measuring log
  - 1**: Create a measuring log: The control saves the **log file named TCHPR420.TXT** in the same folder as the associated NC program.
  - 2**: Interrupt program run and display the measuring log on the control screen (you can later resume the NC program run with **NC Start**)



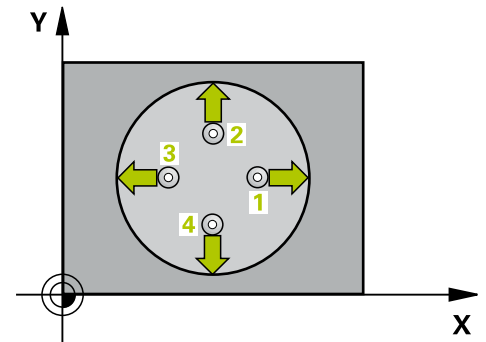
## 6.5 MEASURE HOLE (Circle 421, DIN/ISO: G421, option 17)

### 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.

### Cycle run

- 1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the SET\_UP column of the touch probe table.
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
- 3 Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:



Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation from diameter

### Please note while programming:

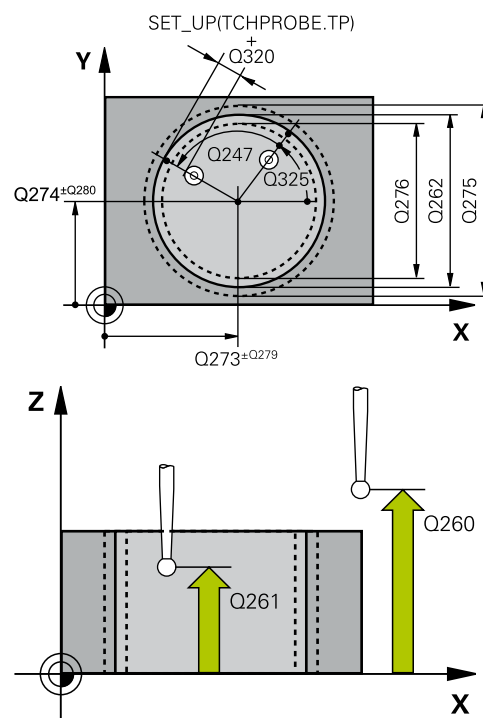
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°.
- Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.



## Cycle parameters



- ▶ **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the hole in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the hole in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q262 Nominal diameter?**: Enter the diameter of the hole.  
Input range: 0 to 99999.9999
- ▶ **Q325 Starting angle?** (absolute): Angle between the principal axis of the working plane and the first touch point.  
Input range: -360.000 to 360.000
- ▶ **Q247 Intermediate stepping angle?** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°. Input range: -120.000 to 120.000
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999



### Example

5 TCH PROBE 421 MEASURE HOLE	
Q273=+50	;CENTER IN 1ST AXIS
Q274=+50	;CENTER IN 2ND AXIS
Q262=75	;NOMINAL DIAMETER
Q325=+0	;STARTING ANGLE
Q247=+60	;STEPPING ANGLE
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=1	;MOVE TO CLEARANCE
Q275=75.12	;MAXIMUM LIMIT



- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:  
**0**: Move at measuring height between measuring points  
**1**: Move at clearance height between measuring points
- ▶ **Q275 Maximum limit of size for hole?**: Maximum permissible diameter for the hole (circular pocket).  
Input range: 0 to 99999.9999
- ▶ **Q276 Minimum limit of size?**: Minimum permissible diameter for the hole (circular pocket).  
Input range: 0 to 99999.9999
- ▶ **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the principal axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the secondary axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q281 Measuring log (0/1/2)?**: Definition of whether the control should create a measuring log:  
**0**: Do not create a measuring log  
**1**: Create a measuring log: The control will save the **log file named TCHPR421.TXT** by default in the directory that also contains the associated NC program.  
**2**: Interrupt the program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.
- ▶ **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:  
**0**: Do not interrupt program run, do not display an error message  
**1**: Interrupt program run and display an error message

Q276=74.95	MINIMUM LIMIT
Q279=0.1	;TOLERANCE 1ST CENTER
Q280=0.1	;TOLERANCE 2ND CENTER
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP TOLERANCE
Q330=0	;TOOL
Q423=4	;NO. OF PROBE POINTS
Q365=1	;TYPE OF TRAVERSE
Q498=0	;REVERSE TOOL
Q531=0	;ANGLE OF INCIDENCE



- ▶ **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 166); alternatively tool name with a maximum of 16 characters  
**0**: Monitoring inactive  
**>0**: Number or name of the tool that the control used for machining. It is possible to take over the tool via soft key directly from the tool table.  
 Input range: 0 to 999999.9
- ▶ **Q423 No. probe points in plane (4/3)?**: Specify whether the control should measure the circle with 4 or 3 touch points:  
**4**: Use 4 measuring points (default setting)  
**3**: Use 3 measuring points
- ▶ **Q365 Type of traverse? Line=0/arc=1**: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" is active (**Q301=1**):  
**0**: Move in a straight line between machining operations  
**1**: Move in a circular arc on the pitch circle diameter between machining operations
- ▶ Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.



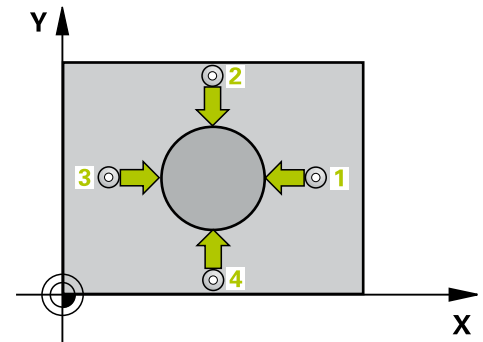
## 6.6 MEASURE CIRCLE OUTSIDE (Cycle 422, DIN/ISO: G422, option 17)

### 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.

### Cycle run

- 1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET\_UP** column of the touch probe table.
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
- 3 Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:



Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation from diameter

### Please note while programming:

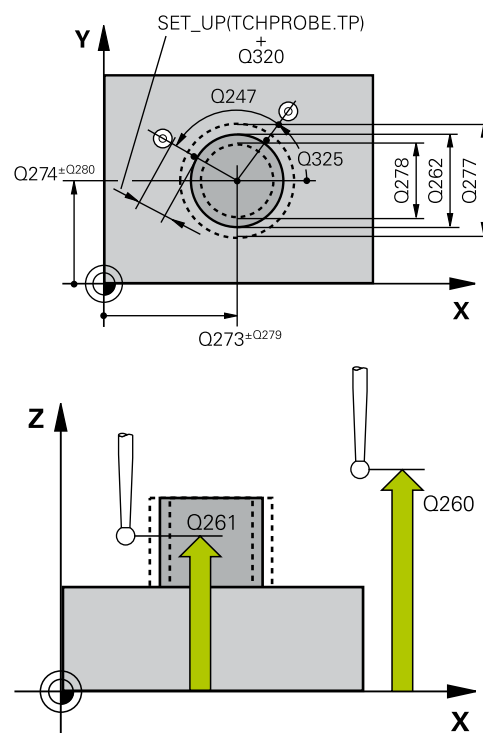
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°.
- Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.



## Cycle parameters



- ▶ **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the stud in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the stud in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q262 Nominal diameter?:** Enter the diameter of the stud.  
Input range: 0 to 99999.9999
- ▶ **Q325 Starting angle?** (absolute): Angle between the principal axis of the working plane and the first touch point.  
Input range: -360.000 to 360.000
- ▶ **Q247 Intermediate stepping angle?** (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the machining direction (negative = clockwise). If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°.  
Input range: -120.0000 to 120.0000
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:  
**0**: Move at measuring height between measuring points  
**1**: Move at clearance height between measuring points



### Example

5 TCH PROBE 422 MEAS. CIRCLE OUTSIDE	
Q273=+50	;CENTER IN 1ST AXIS
Q274=+50	;CENTER IN 2ND AXIS
Q262=75	;NOMINAL DIAMETER
Q325=+90	;STARTING ANGLE
Q247=+30	;STEPPING ANGLE
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+10	;CLEARANCE HEIGHT
Q301=0	;MOVE TO CLEARANCE
Q277=35.15	;MAXIMUM LIMIT
Q278=34.9	;MINIMUM LIMIT
Q279=0.05	;TOLERANCE 1ST CENTER



- ▶ **Q277 Maximum limit of size for stud?:** Maximum permissible diameter for the stud.  
Input range: 0 to 99999.9999
- ▶ **Q278 Minimum limit of size for stud?:** Minimum permissible diameter for the stud.  
Input range: 0 to 99999.9999
- ▶ **Q279 Tolerance for center 1st axis?:** Permissible position deviation in the principal axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q280 Tolerance for center 2nd axis?:** Permissible position deviation in the secondary axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q281 Measuring log (0/1/2)?:** Define whether the control should create a measuring log:  
**0:** Do not create a measuring log  
**1:** Create a measuring log: The control saves the **log file named TCHPR422.TXT** in the same folder as the associated NC program.  
**2:** Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.
- ▶ **Q309 PGM stop if tolerance exceeded?:** Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:  
**0:** Do not interrupt program run, do not display an error message  
**1:** Interrupt program run and display an error message
- ▶ **Q330 Tool for monitoring?:** Define whether the control is to monitor the tool (see "Tool monitoring", Page 166).  
**0:** Monitoring inactive  
**>0:** Tool number in the tool table TOOL.T  
Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters
- ▶ **Q423 No. probe points in plane (4/3)?:** Specify whether the control should measure the circle with 4 or 3 touch points:  
**4:** Use 4 measuring points (default setting)  
**3:** Use 3 measuring points

Q280=0.05	;TOLERANCE 2ND CENTER
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP TOLERANCE
Q330=0	;TOOL
Q423=4	;NO. OF PROBE POINTS
Q365=1	;TYPE OF TRAVERSE
Q498=0	;REVERSE TOOL
Q531=0	;ANGLE OF INCIDENCE



- ▶ **Q365 Type of traverse? Line=0/arc=1:** Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" is active (**Q301=1**):
  - 0:** Move in a straight line between machining operations
  - 1:** Move in a circular arc on the pitch circle diameter between machining operations
- ▶ Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.



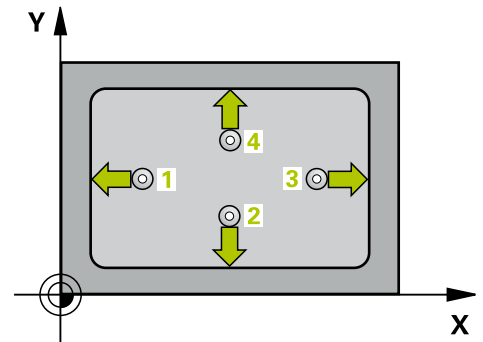
## 6.7 MEASURE RECTANGLE INSIDE (Cycle 423, DIN/ISO: G423, option 17)

### 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 at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET\_UP** column of the touch probe table.
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:



Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q164	Deviation of side length in the reference axis
Q165	Deviation of side length in minor axis

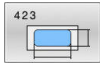


**Please note while programming:**

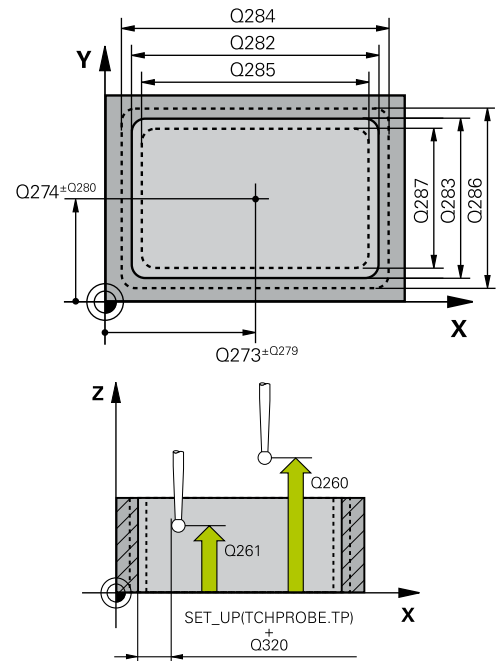
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points.
- Tool monitoring is dependent on the deviation of the first side length.



## Cycle parameters



- ▶ **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the pocket in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the pocket in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q282 1st side length (nominal value)?**: Pocket length, parallel to the principal axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q283 2nd side length (nominal value)?**: Pocket length, parallel to the secondary axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:  
  - 0**: Move at measuring height between measuring points
  - 1**: Move at clearance height between measuring points
- ▶ **Q284 Max. size limit 1st side length?**: Maximum permissible length of the pocket.  
Input range: 0 to 99999.9999
- ▶ **Q285 Min. size limit 1st side length?**: Minimum permissible length of the pocket.  
Input range: 0 to 99999.9999
- ▶ **Q286 Max. size limit 2nd side length?**: Maximum permissible width of the pocket.  
Input range: 0 to 99999.9999



### Example

#### 5 TCH PROBE 423 MEAS. RECTAN. INSIDE

Q273=+50	;CENTER IN 1ST AXIS
Q274=+50	;CENTER IN 2ND AXIS
Q282=80	;FIRST SIDE LENGTH
Q283=60	;2ND SIDE LENGTH
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+10	;CLEARANCE HEIGHT
Q301=1	;MOVE TO CLEARANCE
Q284=0	;MAX. LIMIT 1ST SIDE
Q285=0	;MIN. LIMIT 1ST SIDE
Q286=0	;MAX. LIMIT 2ND SIDE
Q287=0	;MIN. LIMIT 2ND SIDE
Q279=0	;TOLERANCE 1ST CENTER
Q280=0	;TOLERANCE 2ND CENTER
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP TOLERANCE



- ▶ **Q287 Min. size limit 2nd side length?:** Minimum permissible width of the pocket.  
Input range: 0 to 99999.9999
- ▶ **Q279 Tolerance for center 1st axis?:** Permissible position deviation in the principal axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q280 Tolerance for center 2nd axis?:** Permissible position deviation in the secondary axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q281 Measuring log (0/1/2)?:** Define whether the control should create a measuring log:
  - 0:** Do not create a measuring log
  - 1:** Create a measuring log: The control saves the **log file named TCHPR423.TXT** in the same folder as the associated NC program.
  - 2:** Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.
- ▶ **Q309 PGM stop if tolerance exceeded?:** Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  - 0:** Do not interrupt program run, do not display an error message
  - 1:** Interrupt program run and display an error message
- ▶ **Q330 Tool for monitoring?:** Define whether the control is to monitor the tool (see "Tool monitoring", Page 166).
  - 0:** Monitoring inactive
  - >0:** Tool number in the tool table TOOL.T
 Input range: 0 to 32767.9; alternatively tool name with a maximum of 16 characters

Q330=0 ;TOOL



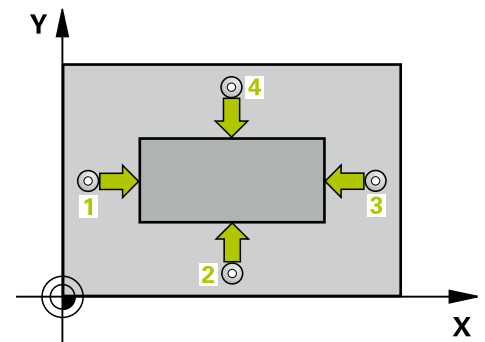
## 6.8 MEASURE RECTANGLE OUTSIDE (Cycle 424, DIN/ISO: G424, option 17)

### 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 at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET\_UP** column of the touch probe table.
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:



Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q164	Deviation of side length in the reference axis
Q165	Deviation of side length in minor axis

### Please note while programming:

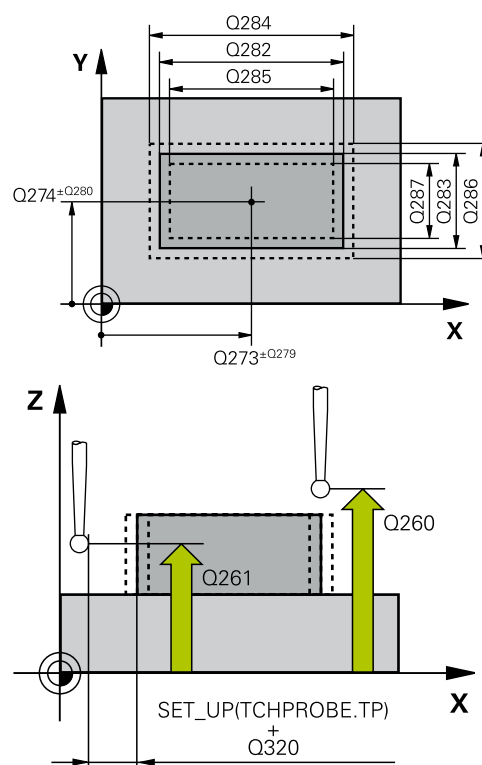
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- Tool monitoring is dependent on the deviation of the first side length.



## Cycle parameters



- ▶ **Q273 Center in 1st axis (nom. value)?** (absolute): Center of the stud in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q274 Center in 2nd axis (nom. value)?** (absolute): Center of the stud in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q282 1st side length (nominal value)?**: Stud length, parallel to the principal axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q283 2nd side length (nominal value)?**: Stud length, parallel to the secondary axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:  
 0: Move at measuring height between measuring points  
 1: Move at clearance height between measuring points
- ▶ **Q284 Max. size limit 1st side length?**: Maximum permissible length of the stud.  
Input range: 0 to 99999.9999
- ▶ **Q285 Min. size limit 1st side length?**: Minimum permissible length of the stud.  
Input range: 0 to 99999.9999



### Example

<b>5 TCH PROBE 424 MEAS. RECTAN. OUTS.</b>
<b>Q273=+50 ;CENTER IN 1ST AXIS</b>
<b>Q274=+50 ;2ND CENTER 2ND AXIS</b>
<b>Q282=75 ;FIRST SIDE LENGTH</b>
<b>Q283=35 ;2ND SIDE LENGTH</b>
<b>Q261=-5 ;MEASURING HEIGHT</b>
<b>Q320=0 ;SET-UP CLEARANCE</b>
<b>Q260=+20 ;CLEARANCE HEIGHT</b>
<b>Q301=0 ;MOVE TO CLEARANCE</b>
<b>Q284=75.1 ;MAX. LIMIT 1ST SIDE</b>
<b>Q285=74.9 ;MIN. LIMIT 1ST SIDE</b>
<b>Q286=35 ;MAX. LIMIT 2ND SIDE</b>
<b>Q287=34.95;MIN. LIMIT 2ND SIDE</b>



- ▶ **Q286 Max. size limit 2nd side length?:** Maximum permissible width of the stud.  
Input range: 0 to 99999.9999
- ▶ **Q287 Min. size limit 2nd side length?:** Minimum permissible width of the stud.  
Input range: 0 to 99999.9999
- ▶ **Q279 Tolerance for center 1st axis?:** Permissible position deviation in the principal axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q280 Tolerance for center 2nd axis?:** Permissible position deviation in the secondary axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q281 Measuring log (0/1/2)?:** Define whether the control should create a measuring log:  
**0:** Do not create a measuring log  
**1:** Create a measuring log: The control will save the **log file named TCHPR424.TXT** by default in the directory that also contains the associated .h file.  
**2:** Interrupt the program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.
- ▶ **Q309 PGM stop if tolerance exceeded?:** Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:  
**0:** Do not interrupt program run, do not display an error message  
**1:** Interrupt program run and display an error message
- ▶ **Q330 Tool for monitoring?:** Define whether the control is to monitor the tool (see "Tool monitoring", Page 166); alternatively tool name with a maximum of 16 characters  
**0:** Monitoring inactive  
**>0:** Number or name of the tool that the control used for machining. It is possible to take over the tool via soft key directly from the tool table.  
Input range: 0 to 999999.9

Q279=0.1	;TOLERANCE 1ST CENTER
Q280=0.1	;TOLERANCE 2ND CENTER
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP TOLERANCE
Q330=0	;TOOL



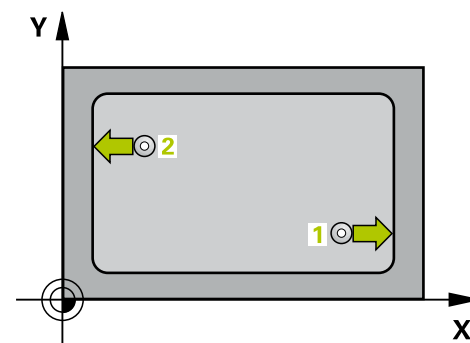
## 6.9 MEASURE INSIDE WIDTH (Cycle 425, DIN/ISO: G425, option 17)

### 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.

### Cycle run

- 1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic "Executing touch probe cycles", at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET\_UP** column of the touch probe table.
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The first probing is always in the positive direction of the programmed axis.
- 3 If you enter an offset for the second measurement, the control then moves the touch probe (if required, at clearance height) to the next touch point **2** and probes that point. If the nominal length is large, the control moves the touch probe to the second touch point at rapid traverse. If you do not enter an offset, the control measures the width in the exact opposite direction.
- 4 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:



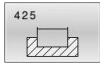
Parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

### Please note while programming:

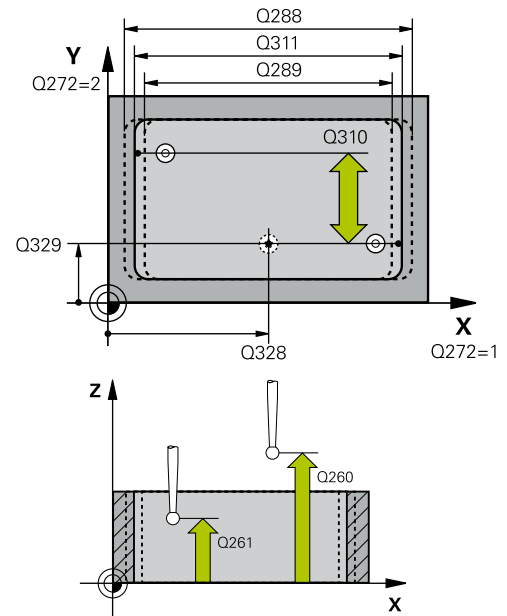
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.



## Cycle parameters



- ▶ **Q328 Starting point in 1st axis?** (absolute):  
Starting point for probing in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q329 Starting point in 2nd axis?** (absolute):  
Starting point for probing in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q310 Offset for 2nd measurement (+/-)?**  
(incremental): Distance by which the touch probe is displaced before the second measurement. If you enter 0, the control does not offset the touch probe.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:  
1: Principal axis = measuring axis  
2: Secondary axis = measuring axis
- ▶ **Q261 Measuring height in probe axis?** (absolute):  
Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q311 Nominal length?** : Nominal value of the length to be measured.  
Input range: 0 to 99999.9999
- ▶ **Q288 Maximum limit of size?**: Maximum permissible length.  
Input range: 0 to 99999.9999
- ▶ **Q289 Minimum limit of size?**: Minimum permissible length.  
Input range: 0 to 99999.9999
- ▶ **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:  
0: Do not create a measuring log  
1: Create a measuring log: By default, the control will save the **log file named TCHPR425.TXT** in the directory that also contains the associated .h file.  
2: Interrupt the program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.



### Example

5 TCH PROBE 425 MEASURE INSIDE WIDTH	
Q328=+75	;STARTNG PNT 1ST AXIS
Q329=-12.5	;STARTNG PNT 2ND AXIS
Q310=+0	;OFFS. 2ND MEASUREMNT
Q272=1	;MEASURING AXIS
Q261=-5	;MEASURING HEIGHT
Q260=+10	;CLEARANCE HEIGHT
Q311=25	;NOMINAL LENGTH
Q288=25.05	;MAXIMUM LIMIT
Q289=25	;MINIMUM LIMIT
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP TOLERANCE
Q330=0	;TOOL
Q320=0	;SET-UP CLEARANCE
Q301=0	;MOVE TO CLEARANCE



- ▶ **Q309 PGM stop if tolerance exceeded?:** Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:  
**0:** Do not interrupt program run, do not display an error message  
**1:** Interrupt program run and display an error message
- ▶ **Q330 Tool for monitoring?:** Define whether the control is to monitor the tool (see "Tool monitoring", Page 166); alternatively tool name with a maximum of 16 characters  
**0:** Monitoring inactive  
**>0:** Number or name of the tool that the control used for machining. It is possible to take over the tool via soft key directly from the tool table.  
Input range: 0 to 999999.9
- ▶ **Q320 Set-up clearance?** (incremental): Additional distance between the touch point and ball tip.  
**Q320** is added to **SET\_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis.  
Input range: 0 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?:** Definition of how the touch probe is to move between the measuring points:  
**0:** Move at measuring height between measuring points  
**1:** Move at clearance height between measuring points



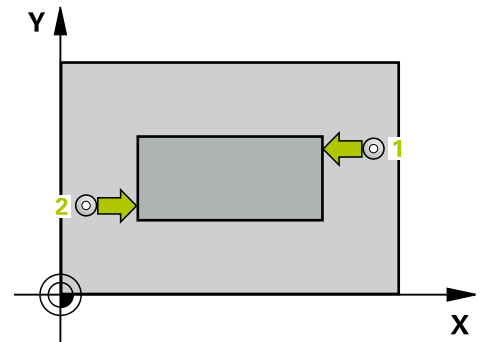
## 6.10 MEASURE RIDGE WIDTH (Cycle 426, DIN/ISO: G426, option 17)

### 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.

### Cycle run

- 1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET\_UP** column of the touch probe table.
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The first probing is always in the negative direction of the programmed axis.
- 3 Then the touch probe moves at clearance height to the next touch point and probes it.
- 4 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:



Parameter number	Meaning
<b>Q156</b>	Actual value of measured length
<b>Q157</b>	Actual value of the centerline
<b>Q166</b>	Deviation of the measured length

### Please note while programming:

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.







- ▶ **Q309 PGM stop if tolerance exceeded?:** Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  - 0:** Do not interrupt program run, do not display an error message
  - 1:** Interrupt program run and display an error message
- ▶ **Q330 Tool for monitoring?:** Define whether the control is to monitor the tool (see "Tool monitoring", Page 166); alternatively tool name with a maximum of 16 characters
  - 0:** Monitoring inactive
  - >0:** Number or name of the tool that the control used for machining. It is possible to take over the tool via soft key directly from the tool table.  
Input range: 0 to 999999.9



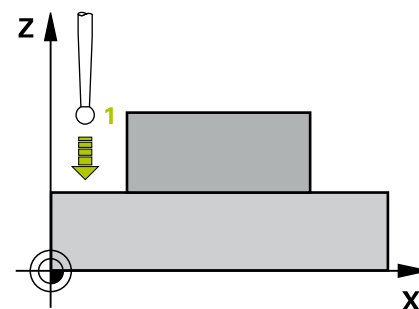
## 6.11 MEASURE COORDINATE (Cycle 427, DIN/ISO: G427, option 17)

### 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.

### Cycle run

- 1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic "Executing touch probe cycles", at touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the defined traverse direction
- 2 Then the control positions the touch probe to the specified touch point **1** in the working plane and measures the actual value in the selected axis.
- 3 Finally, the control returns the touch probe to the clearance height and saves the measured coordinate in the following Q parameter:

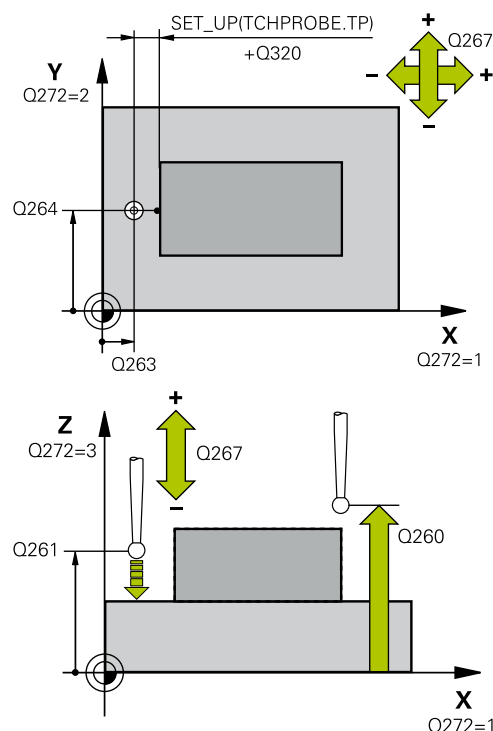
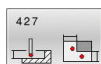


Parameter number	Meaning
<b>Q160</b>	Measured coordinate

### Please note while programming:

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- If an axis of the active working plane is defined as the measuring axis (**Q272** = 1 or 2), the control will perform a tool radius compensation. The control determines the direction of compensation from the defined traversing direction (**Q267**).
- If the touch probe axis is defined as the measuring axis (**Q272** = 3), the control will perform a tool length compensation.
- Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.





5 TCH PROBE 427 MEASURE COORDINATE	
Q263=+35	;1ST POINT 1ST AXIS
Q264=+45	;1ST POINT 2ND AXIS
Q261=+5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q272=3	;MEASURING AXIS
Q267=-1	;TRAVERSE DIRECTION
Q260=+20	;CLEARANCE HEIGHT
Q281=1	;MEASURING LOG
Q288=5.1	;MAXIMUM LIMIT
Q289=4.95	;MINIMUM LIMIT
Q309=0	;PGM STOP TOLERANCE
Q330=0	;TOOL
Q498=0	;REVERSE TOOL
Q531=0	;ANGLE OF INCIDENCE



- ▶ **Q309 PGM stop if tolerance exceeded?:** Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:  
**0:** Do not interrupt program run, do not display an error message  
**1:** Interrupt program run and display an error message
- ▶ **Q330 Tool for monitoring?:** Define whether the control is to monitor the tool (see "Tool monitoring", Page 166); alternatively tool name with a maximum of 16 characters  
**0:** Monitoring inactive  
**>0:** Number or name of the tool that the control used for machining. It is possible to take over the tool via soft key directly from the tool table.  
Input range: 0 to 999999.9
- ▶ 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.



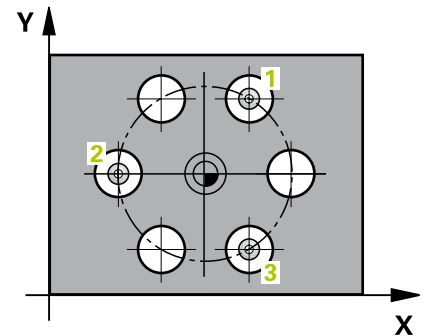
## 6.12 MEAS. BOLT HOLE (Cycle 430, DIN/ISO: G430, option 17)

### 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 rapid traverse speed (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), to the programmed center point of the first hole **1**.
- 2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
- 3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
- 4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
- 5 The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
- 6 The control moves the touch probe to the entered measuring height and probes four points to determine the third hole center point.
- 7 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:



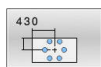
Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of bolt hole circle diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation of bolt circle diameter

### Please note while programming:

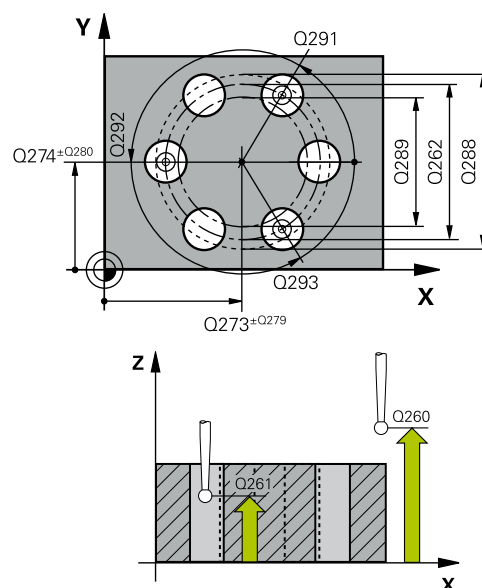
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- Cycle **430** only monitors for tool breakage; there is no automatic tool compensation.



## Cycle parameters



- ▶ **Q273 Center in 1st axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q274 Center in 2nd axis (nom. value)?** (absolute): Bolt hole circle center (nominal value) in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q262 Nominal diameter?**: Enter the diameter of the hole.  
Input range: 0 to 99999.9999
- ▶ **Q291 Polar coord. angle of 1st hole?** (absolute): Polar coordinate angle of the first hole center in the working plane.  
Input range: -360.0000 to 360.0000
- ▶ **Q292 Polar coord. angle of 2nd hole?** (absolute): Polar coordinate angle of the second hole center in the working plane.  
Input range: -360.0000 to 360.0000
- ▶ **Q293 Polar coord. angle of 3rd hole?** (absolute): Polar coordinate angle of the third hole center in the working plane.  
Input range: -360.0000 to 360.0000
- ▶ **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q288 Maximum limit of size?**: Maximum permissible diameter of bolt hole circle.  
Input range: 0 to 99999.9999



### Example

5 TCH PROBE 430 MEAS. BOLT HOLE CIRC
Q273=+50 ;CENTER IN 1ST AXIS
Q274=+50 ;CENTER IN 2ND AXIS
Q262=80 ;NOMINAL DIAMETER
Q291=+0 ;ANGLE OF 1ST HOLE
Q292=+90 ;ANGLE OF 2ND HOLE
Q293=+180 ;ANGLE OF 3RD HOLE
Q261=-5 ;MEASURING HEIGHT
Q260=+10 ;CLEARANCE HEIGHT
Q288=80.1 ;MAXIMUM LIMIT
Q289=79.9 ;MINIMUM LIMIT



- ▶ **Q289 Minimum limit of size?:** Minimum permissible diameter of bolt hole circle.  
Input range: 0 to 99999.9999
- ▶ **Q279 Tolerance for center 1st axis?:** Permissible position deviation in the principal axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q280 Tolerance for center 2nd axis?:** Permissible position deviation in the secondary axis of the working plane.  
Input range: 0 to 99999.9999
- ▶ **Q281 Measuring log (0/1/2)?:** Define whether the control should create a measuring log:  
  - 0:** Do not create a measuring log
  - 1:** Create a measuring log: The control will save the **log file named TCHPR430.TXT** in the directory that also contains the associated NC program
  - 2:** Interrupt the program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.
- ▶ **Q309 PGM stop if tolerance exceeded?:** Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:  
  - 0:** Do not interrupt program run, do not display an error message
  - 1:** Interrupt program run and display an error message
- ▶ **Q330 Tool for monitoring?:** Define whether the control is to monitor the tool (see "Tool monitoring", Page 166); alternatively tool name with a maximum of 16 characters  
  - 0:** Monitoring inactive
  - >0:** Number or name of the tool that the control used for machining. It is possible to take over the tool via soft key directly from the tool table.  
Input range: 0 to 999999.9

Q279=0.15 ;TOLERANCE 1ST CENTER

Q280=0.15 ;TOLERANCE 2ND CENTER

Q281=1 ;MEASURING LOG

Q309=0 ;PGM STOP TOLERANCE

Q330=0 ;TOOL



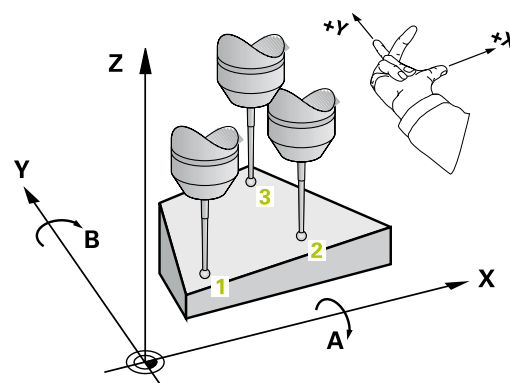
## 6.13 MEASURE PLANE (Cycle 431, DIN/ISO: G431, option 17)

### Application

Touch probe cycle **431** finds the angles of a plane by measuring three points. It saves the measured values in the Q parameters.

### Cycle run

- 1 The control positions the touch probe at rapid traverse (value from the **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 43), at the programmed touch point **1** and measures the first point of the plane. The control offsets the touch probe by the safety clearance in the direction opposite to the direction of probing.
- 2 The touch probe returns to the clearance height and then moves in the working plane to touch point **2** and measures the actual value of the second touch point in the plane.
- 3 The touch probe returns to the clearance height and then moves in the working plane to touch point **3** and measures the actual value of the third touch point in the plane.
- 4 Finally the control returns the touch probe to the clearance height and saves the measured angle values in the following Q parameters:



Parameter number	Meaning
Q158	Projection angle of the A axis
Q159	Projection angle of the B axis
Q170	Spatial angle A
Q171	Spatial angle B
Q172	Spatial angle C
Q173 to Q175	Measured values in the touch probe axis (first to third measurement)



Please note while programming:

**NOTICE**

**Danger of collision!**

If you save the values in the preset table and then tilt the tool by programming **PLANE SPATIAL** with **SPA=0; SPB=0; SPC=0**, there are multiple solutions in which the tilting axes are at 0.

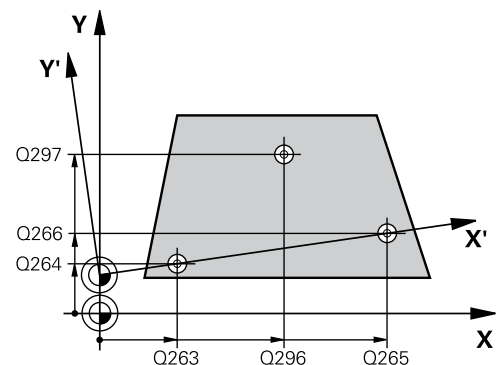
- Make sure to program **SYM (SEQ) +** or **SYM (SEQ) -**

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The control can only calculate the angular values if the three measuring points are not positioned on a straight line.
- The spatial angles that are needed for the **Tilt working plane** function are saved in parameters **Q170** to **Q172**. With the first two measuring points you also specify the direction of the reference axis when tilting the working plane.
- The third measuring point determines the direction of the tool axis. Define the third measuring point in the direction of the positive Y axis to ensure that the position of the tool axis in a clockwise coordinate system is correct.

**Cycle parameters**

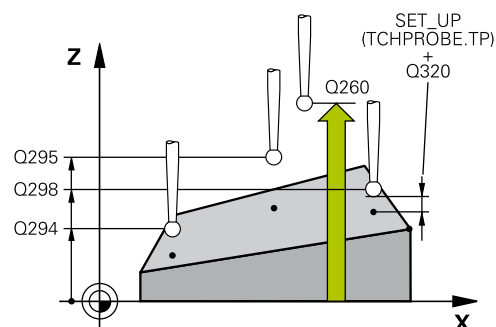


- **Q263 1st measuring point in 1st axis?** (absolute):  
Coordinate of the first touch point in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute):  
Coordinate of the first touch point in the secondary axis of the working plane.  
Input range -99999.9999 to 99999.9999
- **Q294 1st measuring point in 3rd axis?** (absolute):  
Coordinate of the first touch point in the touch probe axis.  
Input range: -99999.9999 to 99999.9999
- **Q265 2nd measuring point in 1st axis?** (absolute):  
Coordinate of the second touch point in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- **Q266 2nd measuring point in 2nd axis?** (absolute):  
Coordinate of the second touch point in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999





- ▶ **Q295 2nd measuring point in 3rd axis?**  
(absolute): Coordinate of the second touch point in the touch probe axis.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q296 3rd measuring point in 1st axis?** (absolute):  
Coordinate of the third touch point in the principal axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q297 3rd measuring point in 2nd axis?**  
(absolute): Coordinate of the third touch point in the secondary axis of the working plane.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q298 3rd measuring point in 3rd axis?**  
(absolute): Coordinate of the third touch point in the touch probe axis.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  - 0**: Do not create a measuring log
  - 1**: Create a measuring log: The control will save the **log file named TCHPR431.TXT** in the directory that also contains the associated NC program
  - 2**: Interrupt the program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.

**Example**

5 TCH PROBE 431 MEASURE PLANE	
Q263=+20	;1ST POINT 1ST AXIS
Q264=+20	;1ST POINT 2ND AXIS
Q294=-10	;1ST POINT 3RD AXIS
Q265=+50	;2ND PNT IN 1ST AXIS
Q266=+80	;2ND PNT IN 2ND AXIS
Q295=+0	;2ND PNT IN 3RD AXIS
Q296=+90	;3RD PNT IN 1ST AXIS
Q297=+35	;THIRD POINT 2ND AXIS
Q298=+12	;3RD PNT IN 3RD AXIS
Q320=0	;SET-UP CLEARANCE
Q260=+5	;CLEARANCE HEIGHT
Q281=1	;MEASURING LOG

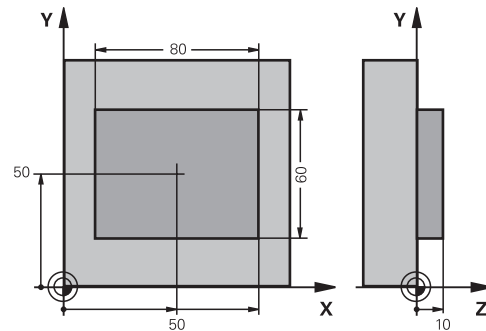


## 6.14 Programming Examples

### Example: Measuring and reworking a rectangular stud

#### Program run

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



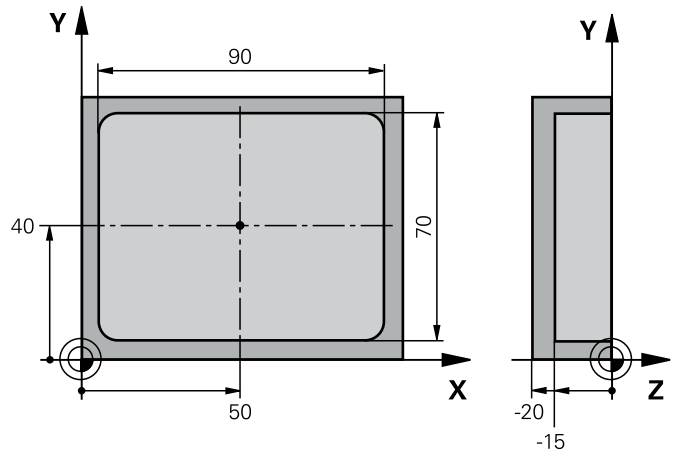
0 BEGIN PGM BEAMS MM	
1 TOOL CALL 69 Z	Tool call: roughing
2 L Z+100 R0 FMAX	Retract the tool
3 FN 0: Q1 = +81	Rectangle length in X (roughing dimension)
4 FN 0: Q2 = +61	Rectangle length in Y (roughing dimension)
5 CALL LBL 1	Call subprogram for machining
6 L Z+100 R0 FMAX	Retract the tool
7 TOOL CALL 99 Z	Call the touch probe
8 TCH PROBE 424 MEAS. RECTAN. OUTS.	Measure the rough-milled rectangle
Q273=+50 ;CENTER IN 1ST AXIS	
Q274=+50 ;CENTER IN 2ND AXIS	
Q282=80 ;FIRST SIDE LENGTH	Nominal length in X (final dimension)
Q283=60 ;2ND SIDE LENGTH	Nominal length in Y (final dimension)
Q261=-5 ;MEASURING HEIGHT	
Q320=0 ;SET-UP CLEARANCE	
Q260=+30 ;CLEARANCE HEIGHT	
Q301=0 ;MOVE TO CLEARANCE	
Q284=0 ;MAX. LIMIT 1ST SIDE	Input values for tolerance checking not required
Q285=0 ;MIN. LIMIT 1ST SIDE	
Q286=0 ;MAX. LIMIT 2ND SIDE	
Q287=0 ;MIN. LIMIT 2ND SIDE	
Q279=0 ;TOLERANCE 1ST CENTER	
Q280=0 ;TOLERANCE 2ND CENTER	
Q281=0 ;MEASURING LOG	No measuring log transmission
Q309=0 ;PGM STOP TOLERANCE	Do not output an error message
Q330=0 ;TOOL	No tool monitoring
9 FN 2: Q1 = +Q1 - +Q164	Calculate length in X including the measured deviation
10 FN 2: Q2 = +Q2 - +Q165	Calculate length in Y including the measured deviation
11 L Z+100 R0 FMAX	Retract the touch probe



12 TOOL CALL 1 Z S5000	Tool call: finishing
13 CALL LBL 1	Call subprogram for machining
14 L Z+100 R0 FMAX M2	Retract the tool, end program
15 LBL 1	Subprogram with rectangular stud machining cycle
16 CYCL DEF 256 RECTANGULAR STUD	
Q218=+Q1 ;FIRST SIDE LENGTH	
Q424=+81 ;WORKPC. BLANK SIDE 1	
Q219=+Q2 ;2ND SIDE LENGTH	
Q425=+61 ;WORKPC. BLANK SIDE 2	
Q220=+0 ;RADIUS / CHAMFER	
Q368=+0.1 ;ALLOWANCE FOR SIDE	
Q224=+0 ;ANGLE OF ROTATION	
Q367=+0 ;STUD POSITION	
Q207=AUTO ;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=+2 ;MACHINING OPERATION	Length in X variable for roughing and finishing
Q369=+0 ;ALLOWANCE FOR FLOOR	Length in Y variable for roughing and finishing
Q338=+20 ;INFEED FOR FINISHING	
Q385=AUTO ;FINISHING FEED RATE	
17 L X+50 Y+50 R0 FMAX M99	Cycle call
18 LBL 0	End of subprogram
19 END PGM BEAMS MM	



### Example: Measuring a rectangular pocket and recording the results



0 BEGIN PGM BSMEAS MM	
1 TOOL CALL 1 Z	Tool call: touch probe
2 L Z+100 R0 FMAX	Retract the touch probe
3 TCH PROBE 423 MEAS. RECTAN. INSIDE	
Q273=+50 ;CENTER IN 1ST AXIS	
Q274=+40 ;CENTER IN 2ND AXIS	
Q282=90 ;FIRST SIDE LENGTH	Nominal length in X
Q283=70 ;2ND SIDE LENGTH	Nominal length in Y
Q261=-5 ;MEASURING HEIGHT	
Q320=0 ;SET-UP CLEARANCE	
Q260=+20 ;CLEARANCE HEIGHT	
Q301=0 ;MOVE TO CLEARANCE	
Q284=90.15 ;MAX. LIMIT 1ST SIDE	Maximum limit in X
Q285=89.95 ;MIN. LIMIT 1ST SIDE	Minimum limit in X
Q286=70.1 ;MAX. LIMIT 2ND SIDE	Maximum limit in Y
Q287=69.9 ;MIN. LIMIT 2ND SIDE	Minimum limit in Y
Q279=0.15 ;TOLERANCE 1ST CENTER	Permissible position deviation in X
Q280=0.1 ;TOLERANCE 2ND CENTER	Permissible position deviation in Y
Q281=1 ;MEASURING LOG	Save measuring log to a file
Q309=0 ;PGM STOP TOLERANCE	Do not display an error message in case of a tolerance violation
Q330=0 ;TOOL	No tool monitoring
4 L Z+100 R0 FMAX M2	Retract the tool, end program
5 END PGM BSMEAS MM	







# 7

**Touch Probe  
Cycles: Special  
Functions**



## 7.1 Fundamentals

### Overview



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

HEIDENHAIN only assumes liability for functionality of the probing cycles if HEIDENHAIN touch probes are used.




### NOTICE

#### Danger of collision!

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- ▶ 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
	MEASURING (Cycle 3, option 17) ■ Touch probe cycle for defining OEM cycles	209
	MEASURING IN 3-D (Cycle 4, option 17) ■ Measuring any position	211
	FAST PROBING (Cycle 441, DIN/ISO: G441, option 17) ■ Touch probe cycle for defining various touch probe parameters	213



## 7.2 MEASURING (Cycle 3, option 17)

### 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 run

- 1 The touch probe moves from the current position at the specified feed rate in the defined probing direction. Use polar angles to define the probing direction in the cycle.
- 2 After the control has saved the position, the touch probe stops. The control saves the X, Y, Z coordinates of the probe-tip center in three successive Q parameters. The control does not conduct any length or radius compensations. You define the number of the first result parameter in the cycle.
- 3 Finally, the control retracts the touch probe by the value that you defined in parameter **MB** in the direction opposite to the probing direction.

### Please note while programming:



The exact behavior of touch probe cycle **3** is defined by your machine tool builder or a software manufacturer who uses it within specific touch probe cycles.

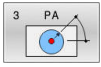
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The **DIST** (maximum traverse to touch point) and **F** (probing feed rate) touch-probe data, which are effective in other touch probe cycles, do not apply in touch probe cycle **3**.
- Remember 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 error message. In this case the control assigns the value -1 to the fourth result parameter so that you can deal with the error yourself.
- The control retracts the touch probe by at most the retraction distance **MB**, but not beyond the starting point of the measurement. This rules out any collision during retraction.



With function **FN17: SYSWRITE ID 990 NR 6** you can set whether the cycle runs through the probe input X12 or X13.



## Cycle parameters



- ▶ **Parameter number for result?:** Enter the number of the Q parameter to which you want the control to assign the first measured coordinate (X). The Y and Z values will be written to the immediately following Q parameters.  
Input range: 0 to 1999
- ▶ **Probing axis?:** Enter the axis in whose direction the probe is to move and confirm with the **ENT** key.  
Input range: X, Y, or Z
- ▶ **Probing angle?:** Angle, measured from the defined **probing axis** in which the touch probe is to move. Confirm with **ENT**.  
Input range: -180.0000 to 180.0000
- ▶ **Maximum measuring range?:** Enter the maximum traverse distance from the starting point by which the touch probe is to move. Confirm with **ENT**.  
Input range: -99999.9999 to 99999.9999
- ▶ **Feed rate measurement:** Enter the feed rate for measurement in mm/min.  
Input range: 0 to 3000.000
- ▶ **Maximum retraction distance?:** Traverse distance in the direction opposite the probing direction, after the stylus was deflected. The control returns the touch probe to a point no farther than the starting point, so that there can be no collision.  
Input range: 0 to 99999.9999
- ▶ **Reference system? (0=ACT/1=REF):** Define whether the probing direction and measuring result should reference the current coordinate system (**ACT**, can be shifted or rotated) or the machine coordinate system (**REF**):  
  - 0:** Probe in the current system and save the measuring result to the **ACT** system
  - 1:** Probe in the machine-based REF system. Save the measuring result in the REF system.
- ▶ **Error mode? (0=OFF/1=ON):** Specify whether the control is to issue an error message if the stylus is deflected at cycle start. If mode **1** is selected, the control saves the value **-1** in the fourth result parameter and continues the cycle:  
  - 0:** Display error message
  - 1:** Do not display error message

## Example

4 TCH PROBE 3.0 MEASURING
5 TCH PROBE 3.1 Q1
6 TCH PROBE 3.2 X ANGLE: +15
7 TCH PROBE 3.3 ABST +10 F100 MB1 REFERENCE SYSTEM: 0
8 TCH PROBE 3.4 ERRORMODE1



## 7.3 MEASURING IN 3-D (Cycle 4, option 17)

### 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 determining the measured value.

### Cycle run

- 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.



#### Operating notes:

- 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.
- Ensure during pre-positioning that the control moves the probe-tip center without compensation to the defined position.

### Please note while programming:

#### NOTICE

#### Danger of collision!

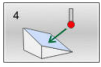
If the control was not able to determine a valid touch point, the 4th result parameter will have the value -1. The control does **not** interrupt the program run!

- ▶ Make sure that all touch points can be reached.

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The control retracts the touch probe by at most the retraction distance **MB**, but not beyond the starting point of the measurement. This rules out any collision during retraction.
- Remember that the control always writes to four successive Q parameters.



## Cycle parameters



- ▶ **Parameter number for result?:** Enter the number of the Q parameter to which you want the control to assign the first measured coordinate (X). The Y and Z values will be written to the immediately following Q parameters.  
Input range: 0 to 1999
- ▶ **Relative measuring path in X?:** X component of the direction vector defining the direction in which the touch probe is to move.  
Input range: -99999.9999 to 99999.9999
- ▶ **Relative measuring path in Y?:** Y component of the direction vector defining the direction in which the touch probe is to move.  
Input range: -99999.9999 to 99999.9999
- ▶ **Relative measuring path in Z?:** Z component of the direction vector defining the direction in which the touch probe is to move.  
Input range: -99999.9999 to 99999.9999
- ▶ **Maximum measuring range?:** Enter the maximum traverse distance from the starting point by which the touch probe may move along the direction vector.  
Input range: -99999.9999 to 99999.9999
- ▶ **Feed rate measurement:** Enter the feed rate for measurement in mm/min.  
Input range: 0 to 3000.000
- ▶ **Maximum retraction distance?:** Traverse distance in the direction opposite the probing direction, after the stylus was deflected.  
Input range: 0 to 99999.9999
- ▶ **Reference system? (0=ACT/1=REF):** Specify whether the result of probing is to be saved in the input coordinate system (**ACT**), or with respect to the machine coordinate system (**REF**):  
**0:** Save measuring result to the **ACT** system  
**1:** Save measuring result to the **REF** system

## Example

4 TCH PROBE 4.0 MEASURING IN 3-D
5 TCH PROBE 4.1 Q1
6 TCH PROBE 4.2 IX-0.5 IY-1 IZ-1
7 TCH PROBE 4.3 ABST+45 F100 MB50 REFERENCE SYSTEM:0



## 7.4 FAST PROBING (Cycle 441, DIN/ISO: G441, option 17)

### 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.



The purpose of Cycle **441** is to set parameters for probing cycles. In this cycle, no machine movements will be performed.

### Please note while programming:



Your machine tool builder may have limited the feed rate additionally. You can set the maximum absolute feed rate in the **maxTouchFeed** machine parameter (no. 122602).

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- **END PGM, M2, M30** reset the global settings of Cycle **441**.
- Cycle parameter **Q399** depends on your machine configuration. Your machine tool builder 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**.



## Cycle parameters



- ▶ **Q396 Positioning feed rate?:** Define the feed rate at which the touch probe will be moved to the specified positions.  
Input range: 0 to 99999.9999; alternatively **FMAX**, **FAUTO**
- ▶ **Q397 Pre-pos. at machine's rapid?:** Define whether the control, when pre-positioning the touch probe, traverses at **FMAX** feed rate (machine's rapid traverse):  
**0:** Pre-position with feed rate from **Q396**  
**1:** Pre-position with machine's rapid traverse **FMAX**. Even if your machine has separate potentiometers for rapid traverse and feed rate, you can control the feed rate with the feed rate potentiometer only, even with **Q397=1**. Your machine tool builder may have limited the feed rate additionally. You can set the maximum absolute feed rate in the **maxTouchFeed** machine parameter (no. 122602).
- ▶ **Q399 Angle tracking (0/1)?:** Define whether the control orients the touch probe before each probing operation:  
**0:** No orientation  
**1:** Orient spindle before each probing operation (improves the accuracy)
- ▶ **Q400 Automatic interruption?** Define whether the control interrupts the program run and outputs the measuring results on the screen following a touch probe cycle for automatic workpiece measurement:  
**0:** Do not interrupt program run even if, in the specific probing cycle, the output of measuring results on the screen is selected  
**1:** Interrupt program run and output measuring results on the screen. You can then resume the NC program run with **NC Start**.

### Example

5 TCH PROBE 441 FAST PROBING	
Q 396=3000;	POSITIONING FEED RATE
Q 397=0	;SELECT FEED RATE
Q 399=1	;ANGLE TRACKING
Q 400=1	;INTERRUPTION



## 7.5 Calibrating a touch trigger probe

In order to precisely specify the actual trigger point of a 3-D touch probe, you must calibrate the touch probe; otherwise the control cannot provide precise measuring results.



Always calibrate a touch probe in the following cases:

- Commissioning
- Broken stylus
- Stylus replacement
- Change in the probe feed rate
- Irregularities caused, for example, due to heating up of the machine
- Change of active tool axis

The control assumes the calibration values for the active probe system directly after the calibration process. The updated tool data are immediately effective. It is not necessary to repeat the tool call.

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

The control provides calibration cycles for calibrating the length and the radius:

Proceed as follows:







- ▶ Press the **TOUCH PROBE** key.



- ▶ Press the **CALIBRATE TS** soft key
- ▶ Select the calibration cycle

Calibration cycles provided by the control

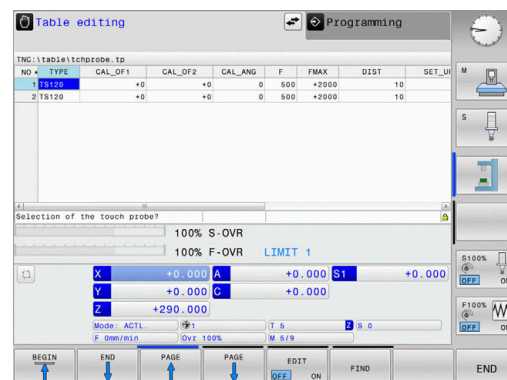
Soft key	Function	Page
	TS CALIBRATION OF TOOL LENGTH (Cycle 461, DIN/ISO: G461, option 17) ■ Calibrating the length	217
	CALIBRATION OF A TS IN A RING (Cycle 462, DIN/ISO: G462, option 17) ■ Measuring the radius using a ring gauge ■ Measuring the center offset using a ring gauge	219
	TS CALIBRATION ON STUD (Cycle 463, DIN/ISO: G463, option 17) ■ Measuring the radius using a stud or a calibration pin ■ Measuring the center offset using a stud or a calibration pin	222
	CALIBRATION OF TS ON A SPHERE (Cycle 460, DIN/ISO: G460, option 17) ■ Measuring the radius using a calibration sphere ■ Measuring the center offset using a calibration sphere	225



## 7.6 Displaying calibration values

The control saves the effective length and effective radius of the touch probe in the tool table. The control saves the touch probe center offset to the touch probe table in the columns **CAL\_OF1** (principal axis) and **CAL\_OF2** (secondary axis). You can display the values on the screen by pressing the TOUCH PROBE TABLE soft key.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs. When running a touch probe cycle in the Manual Operation mode, the control saves the measuring log under the name TCHPRMAN.html. This file is stored in the folder TNC:\\*.



Ensure that the tool number in the tool table and the touch-probe number of the touch-probe table match. This is regardless of whether you want to use a touch-probe cycle in automatic mode or in **Manual operation** mode.



For more information, see Chapter Touch-probe table



## 7.7 TS CALIBRATION OF TOOL LENGTH (Cycle 461, DIN/ISO: G461, option 17)

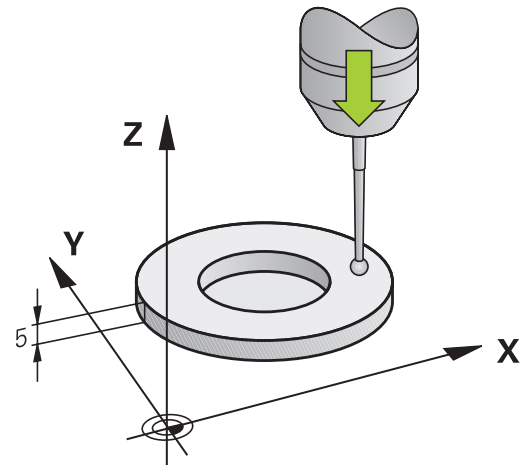
### Application



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 run

- 1 The control orients the touch probe to the angle **CAL\_ANG** specified in the touch probe table (only if your touch probe can be oriented).
- 2 The control probes from the current position in the negative spindle axis direction at the probing feed rate (column **F** from the touch probe table).
- 3 The control then retracts the touch probe at rapid traverse (column **FMAX** from the touch probe table) to the starting position.



**Please note while programming:**

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

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

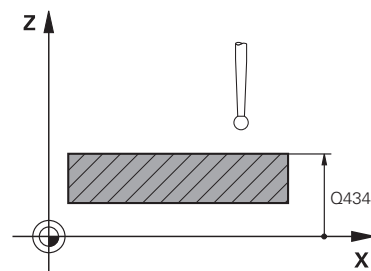
When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- ▶ 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 (and face of the spindle). The machine manufacturer may also place the tool reference point at a different point.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

**Cycle parameters**

- ▶ **Q434 Preset for length?** (absolute): Preset for the length (e.g. height of the calibration ring).  
Input range: -99999.9999 to 99999.9999

**Example**

**5 TCH PROBE 461 TS CALIBRATION OF TOOL LENGTH**

**Q434=+5 ;PRESET**



## 7.8 CALIBRATION OF A TS IN A RING (Cycle 462, DIN/ISO: G462, option 17)

### Application



Refer to your machine manual!

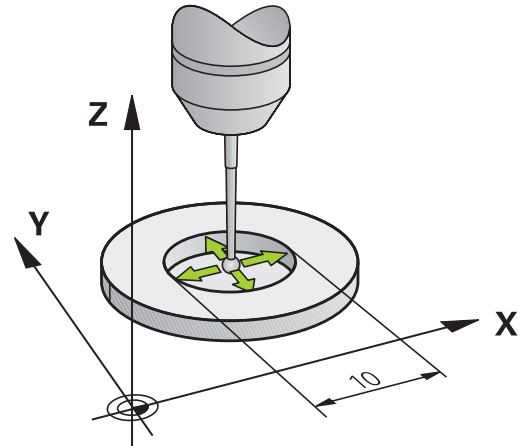
Before starting the calibration cycle, you need to pre-position the touch probe in the center of the calibration ring and at the required measuring height.

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").





**Please note while programming:**

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 tool builder.

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

### NOTICE

**Danger of collision!**

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- ▶ 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.

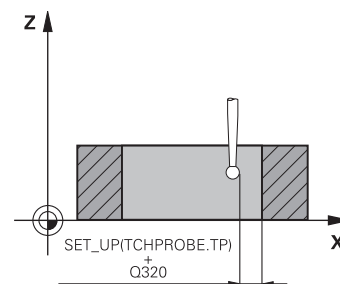
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The center offset can be determined only with a suitable touch probe.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.



### Cycle parameters



- ▶ **Q407 Radius of ring gauge?** Enter the radius of the ring gauge.  
Input range: 0 to 9.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q423 Number of probes?** (absolute): Number of measuring points on the diameter.  
Input range: 3 to 8
- ▶ **Q380 Ref. angle in ref. axis?** (absolute): Angle between the principal axis of the working plane and the first touch point.  
Input range: 0 to 360.0000



### Example

#### 5 TCH PROBE 462 CALIBRATION OF A TS IN A RING

Q407=+5 ;RING RADIUS

Q320=+0 ;SET-UP CLEARANCE

Q423=+8 ;NO. OF PROBE POINTS

Q380=+0 ;REFERENCE ANGLE



## 7.9 TS CALIBRATION ON STUD (Cycle 463, DIN/ISO: G463, option 17)

### 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").



**Please note while programming:**

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 tool builder.

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

### NOTICE

**Danger of collision!**

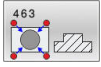
When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- ▶ 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.

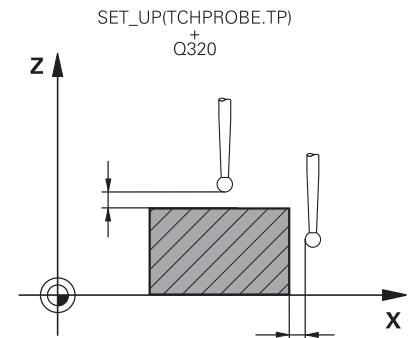
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The center offset can be determined only with a suitable touch probe.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.



### Cycle parameters



- ▶ **Q407 Radius of calibr. stud?:** Diameter of the calibration ring or pin.  
Input range: 0 to 99.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?:** Definition of how the touch probe is to move between the measuring points:  
  - 0:** Move at measuring height between measuring points
  - 1:** Move at clearance height between measuring points
- ▶ **Q423 Number of probes?** (absolute): Number of measuring points on the diameter.  
Input range: 3 to 8
- ▶ **Q380 Ref. angle in ref. axis?** (absolute): Angle between the principal axis of the working plane and the first touch point.  
Input range: 0 to 360.0000



### Example

5 TCH PROBE 463 TS CALIBRATION ON STUD	
Q407=+5	;STUD RADIUS
Q320=+0	;SET-UP CLEARANCE
Q301=+1	;MOVE TO CLEARANCE
Q423=+8	;NO. OF PROBE POINTS
Q380=+0	;REFERENCE ANGLE



## 7.10 CALIBRATION OF TS ON A SPHERE (Cycle 460, DIN/ISO: G460, option 17)

### Application



Refer to your machine manual!

Before starting the calibration cycle, you need to preposition the touch probe above the center of the calibration sphere. Position the touch probe in the touch probe axis by approximately the set-up clearance (value from touch probe table + value from cycle) above the calibration sphere.

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

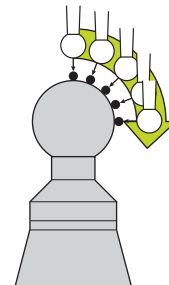
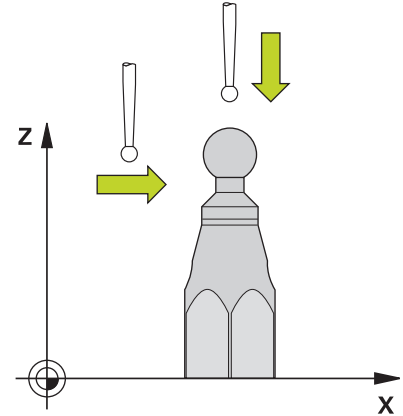
It is also possible to capture 3#D calibration data. Option 92, 3D-ToolComp, is required for this purpose. 3-D calibration data describe the deflection behavior of the touch probe in any probing direction. The 3-D calibration data are stored under TNC: \system\3D-ToolComp\\*. The DR2TABLE column of the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

### Cycle run

The setting in parameter **Q433** specifies whether you can perform radius and length calibration, or just radius calibration.

#### Radius calibration Q433=0

- 1 Clamp the calibration sphere. Ensure the prevention of collisions
- 2 In the touch probe axis, position the touch probe over the calibration sphere, and in the working plane, approximately over the sphere center.
- 3 The first movement is in the plane, depending on the reference angle (**Q380**).
- 4 The control then positions the touch probe in touch-probe axis.
- 5 The probing process starts, and the control begins by searching for the equator of the calibration sphere.
- 6 Once the equator has been determined, the radius calibration begins
- 7 Finally, the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned.





**Radius and length calibration Q433=1**

- 1 Clamp the calibration sphere. Ensure the prevention of collisions
- 2 In the touch probe axis, position the touch probe over the calibration sphere, and in the working plane, approximately over the sphere center.
- 3 The first movement is in the plane, depending on the reference angle (**Q380**).
- 4 The control then positions the touch probe in touch-probe axis.
- 5 The probing process starts, and the control begins by searching for the equator of the calibration sphere.
- 6 Once the equator has been determined, the radius calibration begins
- 7 The control then retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned.
- 8 The control determines the length of the touch probe at the north pole of the calibration sphere.
- 9 At the end of the cycle the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned.

The setting in parameter **Q455** specifies whether you can perform an additional 3-D calibration.

**3-D calibration Q455= 1...30**

- 1 Clamp the calibration sphere. Ensure the prevention of collisions
- 2 After calibration of the radius and length, the control retracts the touch probe in touch-probe axis. Then the control positions the touch probe above the north pole.
- 3 The probing process goes from the north pole to the equator in several steps. Deviations from the nominal value, and therefore the specific deflection behavior, are thus determined
- 4 You can specify the number of probing points between the north pole and the equator. This number depends on input parameter **Q455**. A value between 1 and 30 can be programmed. If you program **Q455=0**, no 3-D calibration will be performed.
- 5 The deviations determined during the calibration are stored in a 3DTC table.
- 6 At the end of the cycle the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned.



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.



**Please note while programming:**



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

## NOTICE

### Danger of collision!

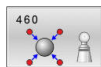
When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- ▶ 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.

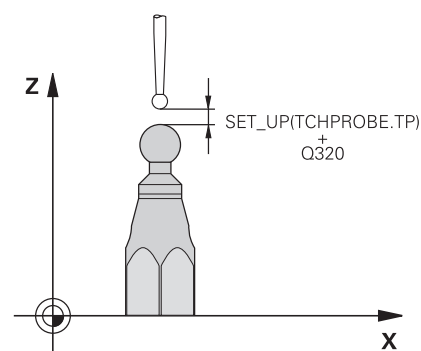
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed 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 (and face of the spindle). The machine manufacturer may also place the tool reference point at a different point.
- Before a cycle definition you must program a tool call to define the touch-probe axis.
- Pre-position the touch probe so that it is located approximately above the center of the calibration sphere.
- Depending on the accuracy of the pre-positioning, finding the equator of the calibration sphere will require a different number of touch points.
- If you program **Q455=0**, the control will not perform a 3-D calibration.
- If you program **Q455=1** to 30, the control will perform a 3-D calibration of the touch probe. Deviations of the deflection behavior will thus be determined under various angles.
- If you program **Q455=1** to 30, a table will be stored under TNC: \system\3D-ToolComp\\*.
- If there is already a reference to a calibration table (entry in DR2TABLE), this table will be overwritten.
- If there is no reference to a calibration table (entry in DR2TABLE), then, in dependence of the tool number, a reference and the associated table will be created.



## Cycle parameters



- ▶ **Q407 Radius of calib. sphere?** Enter the exact radius of the calibration sphere to be used.  
Input range: 0.0001 to 99.9999
- ▶ **Q320 Set-up clearance?** (incremental): Additional distance between the touch point and ball tip.  
**Q320** is added to **SET\_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis.  
Input range: 0 to 99999.9999
- ▶ **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:  
  - 0**: Move at measuring height between measuring points
  - 1**: Move at clearance height between measuring points
- ▶ **Q423 Number of probes?** (absolute): Number of measuring points on the diameter.  
Input range: 3 to 8
- ▶ **Q380 Ref. angle in ref. axis?** (absolute): Enter the reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis.  
Input range: 0 to 360.0000
- ▶ **Q433 Calibrate length (0/1)?**: Define whether the control is to calibrate the touch probe length after radius calibration, as well:  
  - 0**: Do not calibrate touch probe length
  - 1**: Calibrate touch probe length
- ▶ **Q434 Preset for length?** (absolute): Coordinate of the calibration sphere center. This value must only be defined if length calibration is to be carried out.  
Input range: -99999.9999 to 99999.9999
- ▶ **Q455 No. of points for 3-D calibrtn.?** Enter the number of touch points for 3-D calibration. A value of about 15 touch points is useful. If you enter 0, the control will not perform a 3-D calibration. During 3-D calibration, the deflecting behavior of the touch probe is determined under various angles, and the values are stored in a table. 3D-ToolComp is required for 3-D calibration.  
Input range: 1 to 30



## Example

5 TCH PROBE 460 CALIBRATION OF TS ON A SPHERE	
Q407=12.5	;SPHERE RADIUS
Q320=0	;SET-UP CLEARANCE
Q301=1	;MOVE TO CLEARANCE
Q423=4	;NO. OF PROBE POINTS
Q380=+0	;REFERENCE ANGLE
Q433=0	;CALIBRATE LENGTH
Q434=-2.5	;PRESET
Q455=15	;NO. POINTS 3-D CAL.



# 8

**Touch Probe  
Cycles: Automatic  
Kinematics  
Measurement**



## 8.1 Kinematics measurement with TS touch probes (option 48)

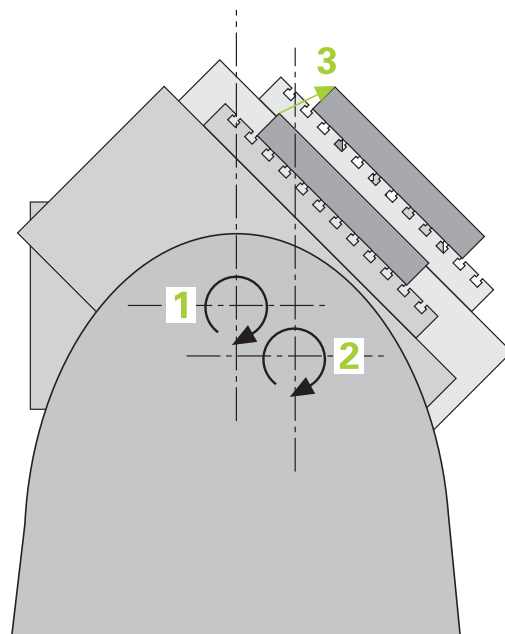
### Fundamentals

Accuracy requirements are becoming increasingly stringent, particularly in the area of 5-axis machining. Complex parts must be manufactured with both precision and reproducible accuracy, including over extended periods of time.

Some of the reasons for inaccuracy in multi-axis machining are deviations between the kinematic model saved in the control (see **1** in the figure at right) and the kinematic conditions actually existing on the machine (see **2** in the figure at right). When the rotary axes are positioned, these deviations cause inaccuracy of the workpiece (see **3** in the figure at right). It is therefore necessary for the model to approach reality as closely as possible.

The **KinematicsOpt** function of the control is an important component that helps you to really meet these complex requirements: a 3-D touch probe cycle measures the rotary axes on your machine fully automatically, regardless of whether they are realized as tables or spindle heads. For this purpose, a calibration sphere is attached at any position on the machine table, and measured with a resolution that you define. During cycle definition, you simply define for each rotary axis the area that you want to measure.




From the measured values, the control calculates the static tilting accuracy. The software minimizes the positioning error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematics table.





## Overview

The control offers the following cycles that enable you to automatically save, restore, check, and optimize the machine kinematics:

Soft key	Cycle	Page
	<b>SAVE KINEMATICS</b> (Cycle 450, DIN/ISO: G450, option 48) <ul style="list-style-type: none"> <li>■ Storing the active machine kinematic configuration</li> <li>■ Restoring previously saved kinematic configuration</li> </ul>	234
	<b>MEASURE KINEMATICS</b> (Cycle 451, DIN/ISO: G451, option 48) <ul style="list-style-type: none"> <li>■ Automatic checking of the machine kinematic configuration</li> <li>■ Optimizing the machine kinematic configuration</li> </ul>	237
	<b>PRESET COMPENSATION</b> (Cycle 452, DIN/ISO: G452, option 48) <ul style="list-style-type: none"> <li>■ Automatic checking of the machine kinematic configuration</li> <li>■ Optimizing the kinematic transformation chain of the machine</li> </ul>	251



## 8.2 Prerequisites



Refer to your machine manual!

Advanced Function Set 1 (option 8) must have been enabled.

Option 17 must have been enabled.

Option 48 must have been enabled.

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

The following are prerequisites for using the KinematicsOpt option:

- The 3-D touch probe used for the measurement must be calibrated.
- The cycles can only be carried out with the tool axis Z.
- A calibration sphere with an exactly known radius and sufficient rigidity must be attached to any position on the machine table
- The kinematics description of the machine must be complete and correct, and the transformation dimensions must have been entered with an accuracy of approx. 1 mm.
- The complete machine geometry must have been measured (by the machine tool builder during commissioning).
- The machine tool builder must have defined the machine parameters for **CfgKinematicsOpt** (no. 204800) in the configuration data.
  - **maxModification** (no. 204801) specifies the tolerance limit starting from which the control is to display a message if the changes made to the kinematic data exceed this limit value.
  - **maxDevCalBall** (no. 204802) defines how much the measured radius of the calibration sphere may deviate from the entered cycle parameter
  - **mStrobeRotAxPos** (no. 204803) defines an M function that is specifically configured by the machine tool builder and is used to position the rotary axes.



HEIDENHAIN recommends using the calibration spheres **KKH 250** (ID number 655475-01) or **KKH 100 (ID number 655475-02)**, which are particularly rigid and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.



**Please note while programming:**

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

If an M function has been defined in the optional **mStrokeRotAxPos** machine parameter (no. 204803), you have to position the rotary axes to 0° (ACTUAL system) before starting one of the KinematicsOpt cycles (except for **450**).

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

When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- ▶ 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.



If machine parameters were changed through the KinematicsOpt cycles, the control must be restarted. Otherwise the changes could be lost in certain circumstances.



## 8.3 SAVE KINEMATICS (Cycle 450, DIN/ISO: G450, option 48)

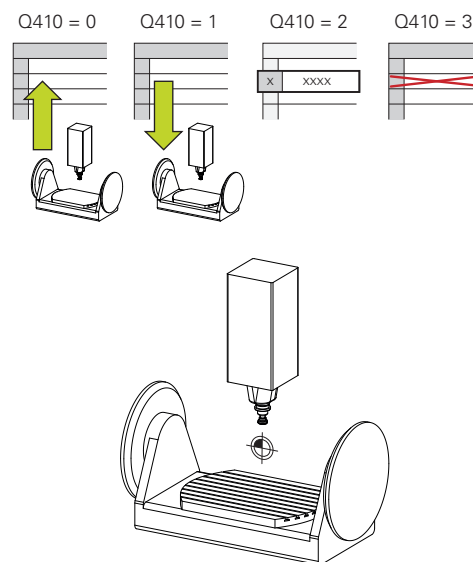
### Application



Refer to your machine manual!

This function must be enabled and adapted by the machine tool builder.

With touch probe cycle **450** you can save the active machine kinematic configuration or restore a previously saved one. The saved data can be displayed and deleted. 16 memory spaces in total are available.



### Please note while programming:



Only save and restore data with Cycle **450** while no tool carrier kinematics configuration that includes transformations is active.

- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- Always save the active kinematics configuration before running a kinematics optimization. Advantage:
  - You can restore the old data if you are not satisfied with the results or if errors occur during optimization (e.g. power failure).
- With the **Restore** mode, note the following:
  - The control can restore saved data only to a matching kinematic configuration
  - A change in the kinematics always changes the preset as well. So redefine the preset, if required.
- The cycle does not restore identical values. It only restores values that differ from the present values. Compensations can only be restored if they had been saved before.



## Cycle parameters



- ▶ **Q410 Mode (0/1/2/3)?**: Define whether you wish to backup or restore the kinematics configuration:
  - 0**: Backup active kinematics configuration
  - 1**: Restore saved kinematics configuration
  - 2**: Display current memory status
  - 3**: Delete a data record.
- ▶ **Q409/QS409 Name of data record?**: Number or name of the record designator. **Q409** has no function if mode 2 has been selected. Wildcards can be used for searches in modes 1 and 3 (Restore and Delete). If the control finds several possible data records because of the wildcards, the control restores the mean values of the data (mode 1) or deletes all data blocks after confirmation (mode 3). You can use the following wildcards for searching:
  - ?**: Any single character
  - \$**: Any single alphabetic character (letter)
  - #**: Any single number
  - \***: Any character string of any length

When specifying a number, you can enter values from 0 to 99999, and when entering letters the string length may not exceed 16 characters. 16 memory locations in total are available.

### Saving the current kinematics

5 TCH PROBE 450 SAVE KINEMATICS
Q410=0 ;MODE
Q409=947 ;MEMORY DESIGNATION

### Restoring data blocks

5 TCH PROBE 450 SAVE KINEMATICS
Q410=1 ;MODE
Q409=948 ;MEMORY DESIGNATION

### Displaying all saved data blocks

5 TCH PROBE 450 SAVE KINEMATICS
Q410=2 ;MODE
Q409=949 ;MEMORY DESIGNATION

### Deleting data blocks

5 TCH PROBE 450 SAVE KINEMATICS
Q410=3 ;MODE
Q409=950 ;MEMORY DESIGNATION

## Log function

After running Cycle **450**, the control creates a log (**tchpr450.txt**) containing the following information:

- Creation date and time of the log
- Name of the NC program from which the cycle was run
- Designator of the current kinematics
- Active tool

The other data in the log vary depending on the selected mode:

- Mode 0: Logging of all axis entries and transformation entries of the kinematics chain that the control has saved.
- Mode 1: Logging of all transformation entries before and after restoring the kinematics configuration.
- Mode 2: List of the saved data records
- Mode 3: List of the deleted data records



## Notes on data management

The control stores the saved data in the file **TNC:\table\DATA450.KD**. This file can be backed up to an external PC with **TNCremo**, for example. If you delete the file, the stored data are removed, too. If the data in the file are changed manually, the data records may become corrupted so that they are unusable.



Operating notes:

- If the file **TNC:\table\DATA450.KD** does not exist, it is generated automatically when Cycle **450** is run.
- Make sure that you delete any empty files with the name **TNC:\table\DATA450.KD** before starting Cycle **450**. If there is an empty memory table (**TNC:\table\DATA450.KD**) without any rows in it, an error message will be issued when running Cycle **450**. In this case, delete the empty memory table and call the cycle again.
- Do not change stored data manually.
- Make a backup of the **TNC:\table\DATA450.KD** file so that you can restore the file, if necessary (e.g. if the data medium is damaged).



## 8.4 MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option 48)

### Application

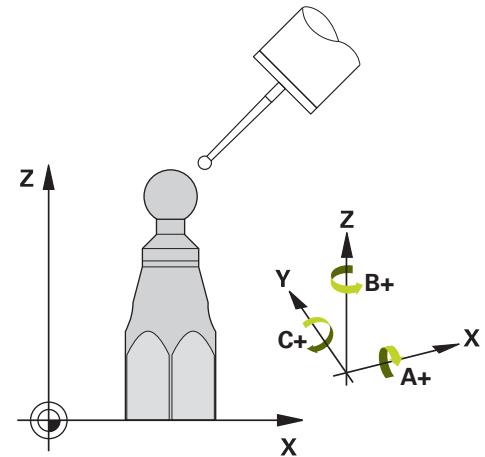


Refer to your machine manual!

This function must be enabled and adapted by the machine tool builder.

Touch probe cycle **451** enables you to check and, if required, optimize the kinematics of your machine. Use the 3-D TS touch probe to measure a HEIDENHAIN calibration sphere that you have attached to the machine table.

The control will determine the static tilting accuracy. The software minimizes the spatial error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematics description.



### Cycle run

- 1 Clamp the calibration sphere and check for potential collisions.
- 2 In Manual Operation mode, set the preset to the center of the sphere or, if you defined **Q431=1** or **Q431=3**: Manually position the touch probe above the calibration sphere in the touch probe axis and at the center of the sphere in the working plane.
- 3 Select the Program Run operating mode and start the calibration program.
- 4 The control automatically measures all rotary axes successively in the resolution you defined.



Programming and operating notes:

- If the kinematics data determined in Optimize mode exceed the permissible limit (**maxModification** no. 204801), the control displays a warning. Then you have to confirm acceptance of the determined values by pressing **NC start**.
- During presetting, the programmed radius of the calibration sphere will only be monitored for the second measurement. The reason is that if pre-positioning with respect to the calibration sphere is inaccurate and you then start presetting, the calibration sphere will be probed twice.



The control saves the measured values in the following Q parameters:

Parameter number	Meaning
Q141	Standard deviation measured in the A axis (-1 if axis was not measured)
Q142	Standard deviation measured in the B axis (-1 if axis was not measured)
Q143	Standard deviation measured in the C axis (-1 if axis was not measured)
Q144	Optimized standard deviation in the A axis (-1 if axis was not optimized)
Q145	Optimized standard deviation in the B axis (-1 if axis was not optimized)
Q146	Optimized standard deviation in the C axis (-1 if axis was not optimized)
Q147	Offset error in X direction, for manual transfer to the corresponding machine parameter
Q148	Offset error in Y direction, for manual transfer to the corresponding machine parameter
Q149	Offset error in Z direction, for manual transfer to the corresponding machine parameter



## Positioning direction

The positioning direction of the rotary axis to be measured is determined from the start angle and the end angle that you define in the cycle. A reference measurement is automatically performed at 0°.

Specify the start and end angles in such a way that the same position is not measured twice. A duplicated point measurement (e.g. measuring positions +90° and -270°) is not advisable, but it will not generate an error message.

- Example: Start angle = +90°, end angle = -90°
  - Start angle = +90°
  - End angle = -90°
  - No. of measuring points = 4
  - Stepping angle resulting from the calculation =  $(-90^\circ - +90^\circ) / (4 - 1) = -60^\circ$
  - Measuring point 1 = +90°
  - Measuring point 2 = +30°
  - Measuring point 3 = -30°
  - Measuring point 4 = -90°
- Example: start angle = +90°, end angle = +270°
  - Start angle = +90°
  - End angle = +270°
  - No. of measuring points = 4
  - Stepping angle resulting from the calculation =  $(270^\circ - 90^\circ) / (4 - 1) = +60^\circ$
  - Measuring point 1 = +90°
  - Measuring point 2 = +150°
  - Measuring point 3 = +210°
  - Measuring point 4 = +270°



**Machines with Hirth-coupled axes****NOTICE****Danger of collision!**

In order to be positioned, the axis must move out of the Hirth grid. If necessary, the control rounds the calculated measuring positions so that they fit into the Hirth grid (depending on the start angle, end angle and number of measuring points).

- ▶ So remember to leave a large enough safety clearance to prevent any risk of collision between the touch probe and calibration sphere
- ▶ Also ensure that there is enough space to reach the safety clearance (software limit switch)

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

Depending on the machine configuration, the control cannot position the rotary axes automatically. If this is the case, you need a special M function from the machine tool builder, enabling the control to move the rotary axes. The machine tool builder must have entered the number of the M function in machine parameter **mStrobeRotAxPos** (no. 244803) for this purpose.

- ▶ Note the documentation of the machine tool builder



Programming and operating notes:

- Define a retraction height greater than 0 if option 2 is not available.
- The measured positions are calculated from the start angle, end angle, and number of measurements for the respective axis and from the Hirth grid.

**Example calculation of measuring positions for an A axis:**

Start angle **Q411** = -30

End angle **Q412** = +90

Number of measuring points **Q414** = 4

Hirth grid = 3°

Calculated stepping angle =  $(Q412 - Q411) / (Q414 - 1)$

Calculated stepping angle =  $(90^\circ - (-30^\circ)) / (4 - 1) = 120 / 3 = 40^\circ$

Measuring position 1 = **Q411** + 0 \* stepping angle = -30° → -30°

Measuring position 2 = **Q411** + 1 \* stepping angle = +10° → 9°

Measuring position 3 = **Q411** + 2 \* stepping angle = +50° → 51°

Measuring position 4 = **Q411** + 3 \* stepping angle = +90° → 90°



### Choice of number of measuring points

To save time, you can make a rough optimization with a small number of measuring points (1 or 2), e.g. when commissioning the machine.

You then make a fine optimization with a medium number of measuring points (recommended value = approx. 4). Higher numbers of measuring points do not usually improve the results. Ideally, you should distribute the measuring points evenly over the tilting range of the axis.

This is why you should measure an axis with a tilting range of  $0^\circ$  to  $360^\circ$  at three measuring points, namely at  $90^\circ$ ,  $180^\circ$  and  $270^\circ$ . Thus, define a starting angle of  $90^\circ$  and an end angle of  $270^\circ$ .

If you want to check the accuracy accordingly, you can also enter a higher number of measuring points in the **Check** mode.



If a measuring point has been defined at  $0^\circ$ , it will be ignored because the reference measurement is always done at  $0^\circ$ .



## Choice of the calibration sphere position on the machine table

In principle, you can fix the calibration sphere to any accessible position on the machine table and also on fixtures or workpieces. The following factors should positively influence the result of measurement:

- On machine with rotary tables/tilting tables: Clamp the calibrating ball as far as possible away from the center of rotation.
- On machines with very large traverse paths: Clamp the calibration sphere as closely as possible to the position intended for subsequent machining.



Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

## Notes on the accuracy



If required, deactivate the lock on the rotary axes for the duration of the calibration. Otherwise it may falsify the results of measurement. The machine manual provides further information.

The geometrical and positioning errors of the machine influence the measured values and therefore also the optimization of a rotary axis. For this reason there will always be a certain amount of error.

If there were no geometrical and positioning errors, any values measured by the cycle at any point on the machine at a certain time would be exactly reproducible. The greater the geometrical and positioning errors are, the greater is the dispersion of measured results when you perform measurements at different positions.

The dispersion of results recorded by the control in the measuring log is a measure of the machine's static tilting accuracy. However, the measuring circle radius and the number and position of measuring points have to be included in the evaluation of accuracy. One measuring point alone is not enough to calculate dispersion. For only one point, the result of the calculation is the spatial error of that measuring point.

If several rotary axes are moved simultaneously, their error values are combined. In the worst case they are added together.



If your machine is equipped with a controlled spindle, you should activate the angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3-D touch probe.



## Notes on various calibration methods

- **Rough optimization during commissioning after entering approximate dimensions.**
  - Number of measuring points between 1 and 2
  - Angular step of the rotary axes: Approx. 90°
- **Fine optimization over the entire range of traverse**
  - Number of measuring points between 3 and 6
  - The start and end angles should cover the largest possible traverse range of the rotary axes.
  - Position the calibration sphere in such a way on the machine table that with rotary table axes, there is a large measuring circle or that on swivel head axes, measurement can be made at a representative position (e.g. in the center of the traverse range).
- **Optimization of a specific rotary axis position**
  - Number of measuring points between 2 and 3
  - The measurements are made with the aid of the inclination angle of an axis (**Q413/Q417/Q421**) around the rotary axis angle at which the workpiece is to be machined later.
  - Position the calibration sphere on the machine table for calibration at the position subsequently intended for machining.
- **Inspecting the machine accuracy**
  - Number of measuring points between 4 and 8
  - The start and end angles should cover the largest possible traverse range of the rotary axes.
- **Determination of the rotary axis backlash**
  - Number of measuring points between 8 and 12
  - The start and end angles should cover the largest possible traverse range of the rotary axes.



## Backlash

Backlash is a small amount of play between the rotary or angle encoder and the table that occurs when the traverse direction is reversed. If the rotary axes have backlash outside of the control loop, for example because the angle measurement is performed with the motor encoder, this can result in significant error during tilting.

With input parameter **Q432**, you can activate backlash measurement. Enter an angle that the control uses as the traversing angle. The cycle will then carry out two measurements per rotary axis. If you take over the angle value 0, the control will not measure any backlash.



Backlash measurement is not possible if an M function for positioning the rotary axes is set in the optional **mStrobeRotAxPos** machine parameter (no. 204803) or if the axis is a Hirth axis.



Programming and operating notes:

- The control does not perform an automatic backlash compensation.
- If the measuring circle radius is  $< 1$  mm, the control does not calculate the backlash. The larger the measuring circle radius, the more accurately the control can determine the rotary axis backlash (see "Log function", Page 250).



### Please note while programming:



Angle compensation is only possible with option 52 KinematicsComp.

If the optional **mStrobeRotAxPos** machine parameter (no. 204803) is not equal to -1 (M function positions the rotary axes), only start a measurement if all rotary axes are at 0°.

In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the optional **maxDevCalBall** machine parameter (no. 204802), the control displays an error message and ends the measurement.

For angle optimization, the machine manufacturer must adapt the configuration correspondingly.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.
- As with Cycles **451** and **452**, Cycle **453** ends with active 3D#ROT in automatic mode, matching the position of the rotary axes.
- Before defining the cycle, you must set the preset to the center of the calibration sphere and activate it, or set input parameter **Q431** to 1 or 3, respectively.
- For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while probe monitoring is inactive.
- The control ignores cycle definition data that applies to inactive axes.
- A correction in the machine datum (**Q406=3**) is only possible if superimposed rotary axes on the spindle head side or table side are measured.
- If you have activated presetting before the calibration (**Q431** = 1/3), then move the touch probe to the set-up clearance (**Q320** + SET\_UP) to a position approximately above the center of the calibration sphere before the start of the cycle.
- Programming in inches: The control always records the log data and results of measurement in millimeters.



- Please note that a change in the kinematics always changes the preset as well. After an optimization, reset the preset.



## Cycle parameters



- ▶ **Q406 Mode (0/1/2/3)?**: Specify whether the control should check or optimize the active kinematics:
  - 0**: Check active machine kinematics. The control measures the kinematics in the rotary axes you have defined, but it does not make any changes to it. The control displays the results of measurement in a measurement log.
  - 1**: Optimize active machine kinematics: the control measures the kinematics in the rotary axes you have defined. It then optimizes **the position of the rotary axes** of the active kinematics.
  - 2**: Optimize active machine kinematics: the control measures the kinematics in the rotary axes you have defined. It then optimizes **angle and positioning errors**. Software option 52, KinematicsComp, is required for the compensation of angle errors.
  - 3**: Optimize active machine kinematics: the control measures the kinematics in the rotary axes you have defined. It then automatically compensates the machine datum. It then optimizes **angle and positioning errors**. Software option 52, KinematicsComp, is required.
- ▶ **Q407 Radius of calib. sphere?** Enter the exact radius of the calibration sphere to be used.  
Input range: 0.0001 to 99.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999 Alternatively **PREDEF**
- ▶ **Q408 Retraction height?** (absolute)
  - 0**: Do not move to any retraction height. The control moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C
  - >0**: Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter **Q253**  
Input range: 0.0001 to 99999.9999
- ▶ **Q253 Feed rate for pre-positioning?** Specify the traversing speed of the tool during pre-positioning in mm/min.  
Input range: 0.0001 to 99999.9999; alternatively **FMAX, FAUTO, PREDEF**

## Saving and checking the kinematics

<b>4 TOOL CALL "TCH PROBE" Z</b>
<b>5 TCH PROBE 450 SAVE KINEMATICS</b>
Q410=0 ;MODE
Q409=5 ;MEMORY DESIGNATION
<b>6 TCH PROBE 451 MEASURE KINEMATICS</b>
Q406=0 ;MODE
Q407=12.5 ;SPHERE RADIUS
Q320=0 ;SET-UP CLEARANCE
Q408=0 ;RETR. HEIGHT
Q253=750 ;F PRE-POSITIONING
Q380=0 ;REFERENCE ANGLE
Q411=-90 ;START ANGLE A AXIS
Q412=+90 ;END ANGLE A AXIS
Q413=0 ;INCID. ANGLE A AXIS
Q414=0 ;MEAS. POINTS A AXIS
Q415=-90 ;START ANGLE B AXIS
Q416=+90 ;END ANGLE B AXIS
Q417=0 ;INCID. ANGLE B AXIS
Q418=2 ;MEAS. POINTS B AXIS
Q419=-90 ;START ANGLE C AXIS
Q420=+90 ;END ANGLE C AXIS
Q421=0 ;INCID. ANGLE C AXIS
Q422=2 ;MEAS. POINTS C AXIS
Q423=4 ;NO. OF PROBE POINTS
Q431=0 ;PRESET
Q432=0 ;BACKLASH, ANG. RANGE



- ▶ **Q380 Ref. angle in ref. axis?** (absolute): Enter the reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis.  
Input range: 0 to 360.0000
- ▶ **Q411 Starting angle in A axis?** (absolute): Starting angle in the A axis at which the first measurement is to be made.  
Input range: -359.999 to 359.999
- ▶ **Q412 End angle in A axis?** (absolute): End angle in the A axis at which the last measurement is to be made.  
Input range: -359.999 to 359.999
- ▶ **Q413 Angle of incidence in A axis?:** Angle of incidence in the A axis at which the other rotary axes are to be measured.  
Input range: -359.999 to 359.999
- ▶ **Q414 No. of meas. points in A (0...12)?:** Number of probe measurements to be used for measuring the A axis. If the input value = 0, the control does not measure the respective axis.  
Input range: 0 to 12
- ▶ **Q415 Starting angle in B axis?** (absolute): Starting angle in the B axis at which the first measurement is to be made.  
Input range: -359.999 to 359.999
- ▶ **Q416 End angle in B axis?** (absolute): End angle in the B axis at which the last measurement is to be made.  
Input range: -359.999 to 359.999
- ▶ **Q417 Angle of incidence in B axis?:** Inclination angle in the B axis at which the other rotary axes are to be measured.  
Input range: -359.999 to 359.999
- ▶ **Q418 No. of meas. points in B (0...12)?:** Number of probe measurements to be used for measuring the B axis. If the input value = 0, the control does not measure the respective axis.  
Input range: 0 to 12
- ▶ **Q419 Starting angle in C axis?** (absolute): Starting angle in the C axis at which the first measurement is to be made.  
Input range: -359.999 to 359.999
- ▶ **Q420 End angle in C axis?** (absolute): End angle in the C axis at which the last measurement is to be made.  
Input range: -359.999 to 359.999
- ▶ **Q421 Angle of incidence in C axis?:** Inclination angle in the C axis at which the other rotary axes are to be measured.  
Input range: -359.999 to 359.999



- ▶ **Q422 No. of meas. points in C (0...12)?**: Number of probe measurements to be used for measuring the C axis. If the input value = 0, the control does not measure the respective axis  
Input range: 0 to 12
- ▶ **Q423 Number of probes?** Define the number of probe measurements with which the control is to measure the calibration sphere in the plane. Fewer measuring points increase speed and more measuring points increase measurement precision.  
Input range: 3 to 8
- ▶ **Q431 Preset (0/1/2/3)?** Define whether the control automatically sets the active preset to the center of the sphere:
  - 0**: Do not define the preset automatically at the center of the sphere: preset manually before cycle start
  - 1**: Define the preset automatically at the center of the sphere (this overwrites the active preset): manually pre-position the touch probe before cycle start using the calibration sphere
  - 2**: Automatically define the preset at the center of the sphere after measuring): preset manually before cycle start
  - 3**: Define the preset before and after measurement at the center of the sphere (the active preset will be overwritten): pre-position the touch probe manually before cycle start using the calibration sphere
- ▶ **Q432 Angular range of backlash comp.?**: Here, you define the angle value to be used as traversing angle when measuring the rotary axis backlash. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the control does not measure the backlash.  
Input range: -3.0000 to +3.0000



## Various modes (Q406)

### Test mode Q406 = 0

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- The control records the results of a possible position optimization but does not make any adjustments.

### "Optimize position of rotary axes" mode Q406 = 1

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- During this, the control tries to change the position of the rotary axis in the kinematics model in order to achieve higher accuracy.
- The machine data are adjusted automatically.

### Position and Angle Optimization mode Q406 = 2

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- First the control tries to optimize the angular orientation of the rotary axis by means of compensation (option 52, KinematicsComp)
- After angle optimization, the control will perform a position optimization. No additional measurements are necessary for this; the control calculates the optimization of the position automatically.



Depending on the machine kinematics for correctly determining the angles, HEIDENHAIN recommends performing the measurement once with an inclination angle of 0°.

### "Optimize machine datum, position, and angle" mode (Q406 = 3)

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- The control automatically tries to optimize the datum (option 52, KinematicsComp). In order to use a machine datum to compensate for the angular position of a rotary axis, the rotary axis to be compensated must be nearer to the machine base than the measured rotary axis.
- The control then tries to optimize the angular orientation of the rotary axis by means of compensation (option 52, KinematicsComp)
- After angle optimization, the control will perform a position optimization. No additional measurements are necessary for this; the control calculates the optimization of the position automatically.



For correct determination of the angles, HEIDENHAIN recommends performing the measurement once with an inclination angle of 0°.

### Position optimization of the rotary axes with preceding, automatic presetting and measurement of the rotary axis backlash

1	TOOL CALL "TCH PROBE" Z
2	TCH PROBE 451 MEASURE KINEMATICS
Q406=1	;MODE
Q407=12.5	;SPHERE RADIUS
Q320=0	;SET-UP CLEARANCE
Q408=0	;RETR. HEIGHT
Q253=750	;F PRE-POSITIONING
Q380=0	;REFERENCE ANGLE
Q411=-90	;START ANGLE A AXIS
Q412=+90	;END ANGLE A AXIS
Q413=0	;INCID. ANGLE A AXIS
Q414=0	;MEAS. POINTS A AXIS
Q415=-90	;START ANGLE B AXIS
Q416=+90	;END ANGLE B AXIS
Q417=0	;INCID. ANGLE B AXIS
Q418=0	;MEAS. POINTS B AXIS
Q419=+90	;START ANGLE C AXIS
Q420=+270	;END ANGLE C AXIS
Q421=0	;INCID. ANGLE C AXIS
Q422=3	;MEAS. POINTS C AXIS
Q423=3	;NO. OF PROBE POINTS
Q431=1	;PRESET
Q432=0.5	;BACKLASH, ANG. RANGE



## Log function

After running Cycle 451, the control will create a log (**TCHPR451.html**) and saves it in the folder that also contains the associated NC program. This log contains the following data:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Mode used (0=Check/1=Optimize position/2=Optimize pose)
- Active kinematic number
- Entered calibration sphere radius
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Angle of incidence
  - Number of measuring points
  - Dispersion (standard deviation)
  - Maximum error
  - Angular error
  - Averaged backlash
  - Averaged positioning error
  - Measuring circle radius
  - Compensation values in all axes (preset shift)
  - Position before optimization of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
  - Position after optimization of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)



## 8.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option 48)

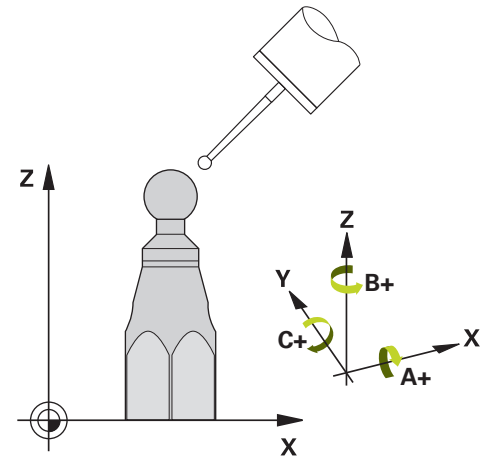
### Application



Refer to your machine manual!

This function must be enabled and adapted by the machine tool builder.

Touch probe cycle **452** optimizes the kinematic transformation chain of your machine (see "MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option 48)", Page 237). Then the control corrects the workpiece coordinate system in the kinematics model in such a way that the current preset is in the center of the calibration sphere after optimization.



### Cycle run



Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

This cycle enables you, for example, to adjust different interchangeable heads so that the workpiece preset applies for all heads.

- 1 Clamp the calibration sphere
- 2 Measure the complete reference head with Cycle **451**, and then use Cycle **451** to set the preset in the center of the sphere.
- 3 Insert the second head
- 4 Use Cycle **452** to measure the interchangeable head up to the point where the head is changed.
- 5 Use Cycle **452** to adjust other interchangeable heads to the reference head

If it is possible to leave the calibration sphere clamped to the machine table during machining, you can compensate for machine drift, for example. This procedure is also possible on a machine without rotary axes.

- 1 Clamp the calibration sphere and check for potential collisions.
- 2 Set the preset in the calibration sphere.
- 3 Set the preset on the workpiece, and start machining the workpiece.
- 4 Use Cycle **452** for preset compensation at regular intervals. The control measures the drift of the axes involved and compensates it in the kinematics description.



Parameter number	Meaning
Q141	Standard deviation measured in the A axis (-1 if axis was not measured)
Q142	Standard deviation measured in the B axis (-1 if axis was not measured)
Q143	Standard deviation measured in the C axis (-1 if axis was not measured)
Q144	Optimized standard deviation in the A axis (-1 if axis was not measured)
Q145	Optimized standard deviation in the B axis (-1 if axis was not measured)
Q146	Optimized standard deviation in the C axis (-1 if axis was not measured)
Q147	Offset error in X direction, for manual transfer to the corresponding machine parameter
Q148	Offset error in Y direction, for manual transfer to the corresponding machine parameter
Q149	Offset error in Z direction, for manual transfer to the corresponding machine parameter



### Please note while programming:



If the kinematics data determined exceed the permissible limit (**maxModification** no. 204801), the control displays a warning. Then you have to confirm acceptance of the determined values by pressing **NC start**.

In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the optional **maxDevCalBall** machine parameter (no. 204802), the control displays an error message and ends the measurement.

In order to be able to perform a preset compensation, the kinematics must be specially prepared. The machine manual provides further information.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.
- As with Cycles **451** and **452**, Cycle **453** ends with active 3D#ROT in automatic mode, matching the position of the rotary axes.
- Note that all functions for tilting in the working plane are reset.
- Before defining the cycle, you must set the preset in the center of the calibration sphere and activate it.
- For rotary axes without separate position encoders, select the measuring points in such a way that you have to traverse an angle of 1° to the limit switch. The control needs this traverse for internal backlash compensation.
- For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while probe monitoring is inactive.
- Programming in inches: The control always records the log data and results of measurement in millimeters.



- If you interrupt the cycle during the measurement, the kinematic data might no longer be in the original condition. Save the active kinematic configuration before an optimization with Cycle **450**, so that in case of a failure the most recently active kinematic configuration can be restored.
- Please note that a change in the kinematics always changes the preset as well. After an optimization, reset the preset.



## Cycle parameters



- ▶ **Q407 Radius of calib. sphere?** Enter the exact radius of the calibration sphere to be used.  
Input range: 0.0001 to 99.9999
- ▶ **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET\_UP** in the touch probe table.  
Input range: 0 to 99999.9999
- ▶ **Q408 Retraction height?** (absolute)  
**0:** Do not move to any retraction height. The control moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C  
**>0:** Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter **Q253**  
 Input range: 0.0001 to 99999.9999
- ▶ **Q253 Feed rate for pre-positioning?** Specify the traversing speed of the tool during pre-positioning in mm/min.  
Input range: 0.0001 to 99999.9999; alternatively **FMAX, FAUTO, PREDEF**
- ▶ **Q380 Ref. angle in ref. axis?** (absolute): Enter the reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis.  
Input range: 0 to 360.0000
- ▶ **Q411 Starting angle in A axis?** (absolute): Starting angle in the A axis at which the first measurement is to be made.  
Input range: -359.999 to 359.999
- ▶ **Q412 End angle in A axis?** (absolute): End angle in the A axis at which the last measurement is to be made.  
Input range: -359.999 to 359.999
- ▶ **Q413 Angle of incidence in A axis?** Angle of incidence in the A axis at which the other rotary axes are to be measured.  
Input range: -359.999 to 359.999
- ▶ **Q414 No. of meas. points in A (0...12)?**: Number of probe measurements to be used for measuring the A axis. If the input value = 0, the control does not measure the respective axis.  
Input range: 0 to 12

## Calibration program

4	TOOL CALL "TCH PROBE" Z
5	TCH PROBE 450 SAVE KINEMATICS
Q410=0	;MODE
Q409=5	;MEMORY DESIGNATION
6	TCH PROBE 452 PRESET COMPENSATION
Q407=12.5	;SPHERE RADIUS
Q320=0	;SET-UP CLEARANCE
Q408=0	;RETR. HEIGHT
Q253=750	;F PRE-POSITIONING
Q380=0	;REFERENCE ANGLE
Q411=-90	;START ANGLE A AXIS
Q412=+90	;END ANGLE A AXIS
Q413=0	;INCID. ANGLE A AXIS
Q414=0	;MEAS. POINTS A AXIS
Q415=-90	;START ANGLE B AXIS
Q416=+90	;END ANGLE B AXIS
Q417=0	;INCID. ANGLE B AXIS
Q418=2	;MEAS. POINTS B AXIS
Q419=-90	;START ANGLE C AXIS
Q420=+90	;END ANGLE C AXIS
Q421=0	;INCID. ANGLE C AXIS
Q422=2	;MEAS. POINTS C AXIS
Q423=4	;NO. OF PROBE POINTS
Q432=0	;BACKLASH, ANG. RANGE



- ▶ **Q415 Starting angle in B axis?** (absolute): Starting angle in the B axis at which the first measurement is to be made.  
Input range: -359.999 to 359.999
- ▶ **Q416 End angle in B axis?** (absolute): End angle in the B axis at which the last measurement is to be made.  
Input range: -359.999 to 359.999
- ▶ **Q417 Angle of incidence in B axis?:** Inclination angle in the B axis at which the other rotary axes are to be measured.  
Input range: -359.999 to 359.999
- ▶ **Q418 No. of meas. points in B (0...12)?:** Number of probe measurements to be used for measuring the B axis. If the input value = 0, the control does not measure the respective axis.  
Input range: 0 to 12
- ▶ **Q419 Starting angle in C axis?** (absolute): Starting angle in the C axis at which the first measurement is to be made.  
Input range: -359.999 to 359.999
- ▶ **Q420 End angle in C axis?** (absolute): End angle in the C axis at which the last measurement is to be made.  
Input range: -359.999 to 359.999
- ▶ **Q421 Angle of incidence in C axis?:** Inclination angle in the C axis at which the other rotary axes are to be measured.  
Input range: -359.999 to 359.999



- ▶ **Q422 No. of meas. points in C (0...12)?:** Number of probe measurements to be used for measuring the C axis. If the input value = 0, the control does not measure the respective axis  
Input range: 0 to 12
- ▶ **Q423 Number of probes?** Define the number of probe measurements with which the control is to measure the calibration sphere in the plane. Fewer measuring points increase speed and more measuring points increase measurement precision.  
Input range: 3 to 8
- ▶ **Q432 Angular range of backlash comp.?:** Here, you define the angle value to be used as traversing angle when measuring the rotary axis backlash. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the control does not measure the backlash.  
Input range: -3.0000 to +3.0000



## Adjustment of interchangeable heads



The head change function can vary depending on the individual machine tool. Refer to your machine manual.

- ▶ Load the second interchangeable head.
- ▶ Insert the touch probe
- ▶ Measure the interchangeable head with Cycle **452**
- ▶ Measure only the axes that have actually been changed (in this example: only the A axis; the C axis is hidden with **Q422**)
- ▶ The preset and the position of the calibration sphere must not be changed during the entire process.
- ▶ All other interchangeable heads can be adjusted in the same way

### Adjusting an interchangeable head

**3 TOOL CALL "TCH PROBE" Z**

**4 TCH PROBE 452 PRESET  
COMPENSATION**

**Q407=12.5 ;SPHERE RADIUS**

**Q320=0 ;SET-UP CLEARANCE**

**Q408=0 ;RETR. HEIGHT**

**Q253=2000 ;F PRE-POSITIONING**

**Q380=+45 ;REFERENCE ANGLE**

**Q411=-90 ;START ANGLE A AXIS**

**Q412=+90 ;END ANGLE A AXIS**

**Q413=45 ;INCID. ANGLE A AXIS**

**Q414=4 ;MEAS. POINTS A AXIS**

**Q415=-90 ;START ANGLE B AXIS**

**Q416=+90 ;END ANGLE B AXIS**

**Q417=0 ;INCID. ANGLE B AXIS**

**Q418=2 ;MEAS. POINTS B AXIS**

**Q419=+90 ;START ANGLE C AXIS**

**Q420=+270 ;END ANGLE C AXIS**

**Q421=0 ;INCID. ANGLE C AXIS**

**Q422=0 ;MEAS. POINTS C AXIS**

**Q423=4 ;NO. OF PROBE POINTS**

**Q432=0 ;BACKLASH, ANG. RANGE**



The goal of this procedure is to achieve that the workpiece preset remains unchanged after changing rotary axes (head change).

In the following example, the adjustment of a fork head with A and C axes is described. The A axis is changed, whereas the C axis continues being a part of the basic configuration.

- ▶ Insert the interchangeable head that will be used as a reference head.
- ▶ Clamp the calibration sphere
- ▶ Insert the touch probe
- ▶ Use Cycle **451** to measure the complete kinematics, including the reference head
- ▶ Define the preset (using **Q431** = 2 or 3 in Cycle **451**) after measuring the reference head

#### Measuring a reference head

<b>1 TOOL CALL "TCH PROBE" Z</b>
<b>2 TCH PROBE 451 MEASURE KINEMATICS</b>
Q406=1 ;MODE
Q407=12.5 ;SPHERE RADIUS
Q320=0 ;SET-UP CLEARANCE
Q408=0 ;RETR. HEIGHT
Q253=2000 ;F PRE-POSITIONING
Q380=+45 ;REFERENCE ANGLE
Q411=-90 ;START ANGLE A AXIS
Q412=+90 ;END ANGLE A AXIS
Q413=45 ;INCID. ANGLE A AXIS
Q414=4 ;MEAS. POINTS A AXIS
Q415=-90 ;START ANGLE B AXIS
Q416=+90 ;END ANGLE B AXIS
Q417=0 ;INCID. ANGLE B AXIS
Q418=2 ;MEAS. POINTS B AXIS
Q419=+90 ;START ANGLE C AXIS
Q420=+270 ;END ANGLE C AXIS
Q421=0 ;INCID. ANGLE C AXIS
Q422=3 ;MEAS. POINTS C AXIS
Q423=4 ;NO. OF PROBE POINTS
Q431=3 ;PRESET
Q432=0 ;BACKLASH, ANG. RANGE



## Drift compensation



This procedure can also be performed on machines without rotary axes.

During machining, various machine components are subject to drift due to varying ambient conditions. If the drift remains sufficiently constant over the range of traverse, and if the calibration sphere can be left on the machine table during machining, the drift can be measured and compensated with Cycle **452**.

- ▶ Clamp the calibration sphere
- ▶ Insert the touch probe
- ▶ Measure the complete kinematics with Cycle **451** before starting the machining process
- ▶ Define the preset (using **Q432** = 2 or 3 in Cycle **451**) after measuring the kinematics
- ▶ Then set the presets on your workpiece and start the machining process.

## Reference measurement for drift compensation

<b>1</b>	<b>TOOL CALL "TCH PROBE" Z</b>
<b>2</b>	<b>CYCL DEF 247 PRESETTING</b>
	<b>Q339=1 ;PRESET NUMBER</b>
<b>3</b>	<b>TCH PROBE 451 MEASURE KINEMATICS</b>
	<b>Q406=1 ;MODE</b>
	<b>Q407=12.5 ;SPHERE RADIUS</b>
	<b>Q320=0 ;SET-UP CLEARANCE</b>
	<b>Q408=0 ;RETR. HEIGHT</b>
	<b>Q253=750 ;F PRE-POSITIONING</b>
	<b>Q380=+45 ;REFERENCE ANGLE</b>
	<b>Q411=+90 ;START ANGLE A AXIS</b>
	<b>Q412=+270 ;END ANGLE A AXIS</b>
	<b>Q413=45 ;INCID. ANGLE A AXIS</b>
	<b>Q414=4 ;MEAS. POINTS A AXIS</b>
	<b>Q415=-90 ;START ANGLE B AXIS</b>
	<b>Q416=+90 ;END ANGLE B AXIS</b>
	<b>Q417=0 ;INCID. ANGLE B AXIS</b>
	<b>Q418=2 ;MEAS. POINTS B AXIS</b>
	<b>Q419=+90 ;START ANGLE C AXIS</b>
	<b>Q420=+270 ;END ANGLE C AXIS</b>
	<b>Q421=0 ;INCID. ANGLE C AXIS</b>
	<b>Q422=3 ;MEAS. POINTS C AXIS</b>
	<b>Q423=4 ;NO. OF PROBE POINTS</b>
	<b>Q431=3 ;PRESET</b>
	<b>Q432=0 ;BACKLASH, ANG. RANGE</b>



- ▶ Measure the drift of the axes at regular intervals.
- ▶ Insert the touch probe
- ▶ Activate the preset in the calibration sphere.
- ▶ Use Cycle **452** to measure the kinematics.
- ▶ The preset and the position of the calibration sphere must not be changed during the entire process.

#### Drift compensation

<b>4 TOOL CALL "TCH PROBE" Z</b>	
<b>5 TCH PROBE 452 PRESET COMPENSATION</b>	
Q407=12.5	;SPHERE RADIUS
Q320=0	;SET-UP CLEARANCE
Q408=0	;RETR. HEIGHT
Q253=99999F	PRE-POSITIONING
Q380=+45	;REFERENCE ANGLE
Q411=-90	;START ANGLE A AXIS
Q412=+90	;END ANGLE A AXIS
Q413=45	;INCID. ANGLE A AXIS
Q414=4	;MEAS. POINTS A AXIS
Q415=-90	;START ANGLE B AXIS
Q416=+90	;END ANGLE B AXIS
Q417=0	;INCID. ANGLE B AXIS
Q418=2	;MEAS. POINTS B AXIS
Q419=+90	;START ANGLE C AXIS
Q420=+270	;END ANGLE C AXIS
Q421=0	;INCID. ANGLE C AXIS
Q422=3	;MEAS. POINTS C AXIS
Q423=3	;NO. OF PROBE POINTS
Q432=0	;BACKLASH, ANG. RANGE



## Log function

After running Cycle **452**, the control creates a log  
(**TCHPR452.html**) containing the following information:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Active kinematic number
- Entered calibration sphere radius
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Angle of incidence
  - Number of measuring points
  - Dispersion (standard deviation)
  - Maximum error
  - Angular error
  - Averaged backlash
  - Averaged positioning error
  - Measuring circle radius
  - Compensation values in all axes (preset shift)
  - Measurement uncertainty of rotary axes
  - Position before preset compensation of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
  - Position after preset compensation of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)

## Notes on log data

(see "Log function", Page 250)







# 9

**Touch Probe  
Cycles: Automatic  
Tool Measurement**



## 9.1 Fundamentals

### Overview



Refer to your machine manual!

Some cycles and functions may not be provided on your machine.

Option 17 is required.

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



Operating notes










- When running touch probe cycles, Cycle **8 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** mode of operation using the **TOUCH PROBE** key. The following cycles are available:

New format	Old format	Cycle	Page
		CALIBRATE TT (Cycle 30 or 480, ISO: G480, option 17) ■ Calibrating the tool touch probe	269
		Measuring tool length (Cycle 31 or 481, DIN/ISO: G481, option 17) ■ Measuring the tool length	272
		Measuring a tool radius (Cycle 32 or 482, ISO: G482, option 17) ■ Measuring the tool radius	276
		Measuring tool length and radius (Cycle 33 or 483, ISO: G483, option 17) ■ Measuring the tool length and radius	280
		CALIBRATE IR TT (Cycle 484, ISO: G484, option 17) ■ Calibrating the tool touch probe (e.g., infrared tool touch probe)	284



#### 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 30 to 33 and Cycles 480 to 483

The features and the operating sequences are absolutely identical. There are only the following differences between Cycles **30 to 33** and Cycles **480 to 483**:

- Cycles **480 to 483** are also available as **G480 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**.



## 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).

When measuring a rotating tool, the control automatically calculates the spindle speed and feed rate for probing.

The spindle speed is calculated as follows:

$n = \text{maxPeriphSpeedMeas} / (r \cdot 0.0063)$  where

- n:** Spindle speed [rpm]  
**maxPeriphSpeedMeas:** Maximum permissible cutting speed in m/min  
**r:** Active tool radius [mm]

The probing feed rate is calculated as follows:

$v = \text{measuring tolerance} \cdot n$  with

- v:** Probing feed rate [mm/min]  
**Measuring tolerance** Measuring tolerance [mm], depending on **maxPeriphSpeedMeas**  
**n:** Shaft speed [rpm]



**probingFeedCalc** (no. 122710) determines the calculation of the probing feed rate:

**probingFeedCalc** (no. 122710) = **ConstantTolerance**:

The measuring tolerance remains constant—regardless of the tool radius. With very large tools, however, the feed rate for probing is reduced to zero. The lower you set the maximum permissible rotational speed (**maxPeriphSpeedMeas** (no. 122712) and the permissible tolerance (**measureTolerance1** (no. 122715), the sooner you will encounter this effect.

**probingFeedCalc** (no. 122710) = **VariableTolerance**:

The measuring tolerance is adjusted relative to the size of the tool radius. This ensures a sufficient feed rate for probing even with large tool radii. The control adjusts the measuring tolerance according to the following table:

Tool radius	Measuring tolerance
Up to 30 mm	<b>measureTolerance1</b>
30 to 60 mm	$2 \bullet \text{measureTolerance1}$
60 to 90 mm	$3 \bullet \text{measureTolerance1}$
90 to 120 mm	$4 \bullet \text{measureTolerance1}$

**probingFeedCalc** (No. 122710) = **ConstantFeed**:

The measuring feed rate remains constant; the measuring error, however, rises linearly with the increase in tool radius:

Measuring tolerance =  $(r \bullet \text{measureTolerance1}) / 5 \text{ mm}$  where

**r:** Active tool radius [mm]  
**measureTolerance1:** Maximum permissible error of measurement



## Entries in the tool table for milling tools

Abbr.	Inputs	Dialog
<b>CUT</b>	Number of teeth (20 teeth maximum)	<b>Number of teeth?</b>
<b>LTOL</b>	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the control locks the tool (status <b>L</b> ). Input range: 0 to 0.9999 mm	<b>Wear tolerance: length?</b>
<b>RTOL</b>	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the control locks the tool (status <b>L</b> ). Input range: 0 to 0.9999 mm	<b>Wear tolerance: radius?</b>
<b>DIRECT.</b>	Cutting direction of the tool for measuring a rotating tool	<b>Cutting direction (M3 = -)?</b>
<b>R-OFFS</b>	Tool length measurement: Tool offset between stylus center and tool center. Default setting: No value entered (offset = tool radius)	<b>Tool offset: radius?</b>
<b>L-OFFS</b>	Radius measurement: Tool offset between upper edge of stylus and lower edge of tool in addition to <b>offsetToolAxis</b> . Default: 0	<b>Tool offset: length?</b>
<b>LBREAK</b>	Permissible deviation from tool length L for breakage detection. If the entered value is exceeded, the control locks the tool (status <b>L</b> ). Input range: 0 to 0.9999 mm	<b>Breakage tolerance: length?</b>
<b>RBREAK</b>	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the control locks the tool (status <b>L</b> ). Input range: 0 to 0.9999 mm	<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.



## 9.2 CALIBRATE TT (Cycle 30 or 480, ISO: G480, option 17)

### Application



Refer to your machine manual!

You calibrate the TT with touch probe cycle **30** or **480**(see "Differences between Cycles 30 to 33 and Cycles 480 to 483", Page 265). The calibration process runs automatically. The control also measures the center misalignment of the calibration tool automatically by rotating the spindle by 180° after the first half of the calibration cycle.

### 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

**Please note while programming:**

The functionality of this cycle depends on the optional **probingCapability** machine parameter (no. 122723). (This parameter includes the possibility of tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.)

The operation of the calibration cycle depends on machine parameters **CfgTTRoundStylus** (no. 114200) or **CfgTTRectStylus** (no. 114300). Refer to your machine manual.

- 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.
- The position of the TT within the machine working space must be defined by setting machine parameters **centerPos** (no. 114201) > **[0]** to **[2]**.
- If you change the position of the **TT** on the table and the setting of any of the machine parameters **centerPos** (no. 114201) > **[0]** to **[2]**, you must recalibrate.



## Cycle parameters



- **Q260 Clearance height?:** Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height references the active workpiece preset. If you enter such a small clearance height value that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistToolAx** (no. 114203)).  
Input range: -99999.9999 to 99999.9999

### Example of old format

6 TOOL CALL 1 Z

7 TCH PROBE 30.0 CALIBRATE TT

8 TCH PROBE 30.1 HEIGHT: +90

### Example of new format

6 TOOL CALL 1 Z

7 TCH PROBE 480 CALIBRATE TT

Q260=+100 ;CLEARANCE HEIGHT



### 9.3 Measuring tool length (Cycle 31 or 481, DIN/ISO: G481, option 17)

#### Application



Refer to your machine manual!

If you want to measure the tool length, program the touch probe cycle **31** or **481** (see "Differences between Cycles 30 to 33 and Cycles 480 to 483"). Via input parameters you can measure the length of a tool by three methods:

- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the tool while it is rotating.
- If the tool diameter is smaller than the diameter of the measuring surface of the TT, or if you are measuring the length of a drill or spherical cutter, you measure the tool while it is stationary.
- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the individual teeth of the tool while it is stationary.

#### Cycle for measuring a tool during rotation

The control determines the longest tooth of a rotating tool by positioning the tool to be measured at an offset to the center of the touch probe and then moving it toward the measuring surface of the TT until it contacts the surface. The offset is programmed in the tool table under Tool offset: Radius (**R-OFFS**).

#### Cycle for measuring a stationary tool (e.g. for drills)

The control positions the tool to be measured above the center of the measuring surface. It then moves the non-rotating tool toward the measuring surface of the TT until contact is made. For this measurement, enter 0 in the tool table under Tool offset: radius (**R-OFFS**).

#### Cycle for measuring individual teeth

The control pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the tool to the upper edge of the touch probe head is defined in **offsetToolAxis** (no. 122707). You can enter an additional offset in Tool offset: Length ( **L-OFFS** ) in the tool table. The control probes the tool radially while it is rotating to determine the starting angle for measuring the individual teeth. It then measures the length of each tooth by changing the corresponding angle of spindle orientation. To activate this function in Cycle **31**, set the parameter PROBING THE TEETH to 1.



**Please note while programming:**

**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 or turning, grinding, or dressing tools.



## Cycle parameters



- ▶ **Q340 Tool measurement mode (0-2)?:** Specify whether and how the determined data will be entered in the tool table.  
**0:** The measured tool length is written to column L of tool table TOOL.T, and the tool compensation is set to DL=0. If there is already a value in TOOL.T, it will be overwritten.  
**1:** The measured tool length is compared to the tool length L from TOOL.T. The control calculates the deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation can also be used for parameter **Q115**. If the delta value is greater than the permissible tool length tolerance for wear or breakage, the control will lock the tool (status L in TOOL.T)  
**2:** The measured tool length is compared to the tool length L from TOOL.T. The control calculates the deviation from the stored value and writes it to Q parameter **Q115**. Nothing is entered under L or DL in the tool table.
- ▶ **Q260 Clearance height?:** Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height references the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**).  
Input range: -99999.9999 to 99999.9999
- ▶ **Q341 Probe the teeth? 0=no/1=yes:** Choose whether the control is to measure the individual teeth (maximum of 20 teeth).
- ▶ **For further information,** Page 275

### Example of new format

6 TOOL CALL 12 Z	
7 TCH PROBE 481 CAL. TOOL LENGTH	
Q340=1	;CHECK
Q260=+100	;CLEARANCE HEIGHT
Q341=1	;PROBING THE TEETH



Cycle **31** includes an additional parameter:



- **Parameter number for result?:** Parameter number in which the control saves the status of measurement results:
  - 0.0:** Tool is within tolerance
  - 1.0:** Tool is worn (**LTOL** exceeded)
  - 2.0:** Tool is broken (**LBREAK** exceeded). If you do not wish to use the result of measurement in your NC program, answer the dialog prompt with **NO ENT**.

#### Measuring a rotating tool for the first time; old format

6 TOOL CALL 12 Z
7 TCH PROBE 31.0 CAL. TOOL LENGTH
8 TCH PROBE 31.1 CHECK: 0
9 TCH PROBE 31.2 HEIGHT: +120
10 TCH PROBE 31.3 PROBING THE TEETH: 0

#### Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

6 TOOL CALL 12 Z
7 TCH PROBE 31.0 CAL. TOOL LENGTH
8 TCH PROBE 31.1 CHECK: 1 Q5
9 TCH PROBE 31.2 HEIGHT: +120
10 TCH PROBE 31.3 PROBING THE TEETH: 1



## 9.4 Measuring a tool radius (Cycle 32 or 482, ISO: G482, option 17)

### 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 30 to 33 and Cycles 480 to 483", Page 265). Select via input parameters by which of two methods the tool radius is to be measured:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth

The control pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the face of the milling tool to the upper edge of the touch probe head is defined in **offsetToolAxis** (no. 122707). The control probes the tool radially while it is rotating. If you have programmed a subsequent measurement of individual teeth, the control will measure the radius of each tooth with the aid of oriented spindle stops.



**Please note while programming:**



The functionality of this cycle depends on the optional **probingCapability** machine parameter (no. 122723). (This parameter includes the possibility of tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.)

Cylindrical tools with diamond surfaces can be measured while the spindle is stationary. To do so, define the number of teeth **CUT** as 0 in the tool table and adjust machine parameter **CfgTT** (no. 122700). Refer to your machine manual.

**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 or turning, grinding, or dressing tools.



## Cycle parameters



- ▶ **Q340 Tool measurement mode (0-2)?:** Specify whether and how the determined data will be entered in the tool table.  
**0:** The measured tool radius is written to column R of the TOOL.T tool table, and the tool compensation is set to DR=0. If there is already a value in TOOL.T, it will be overwritten.  
**1:** The measured tool radius is compared to the tool radius R from TOOL.T. The control calculates the deviation from the stored value and enters it into TOOL.T as the delta value DR. The deviation can also be used for parameter **Q116**. If the delta value is greater than the permissible tool radius tolerance for wear or breakage, the control will lock the tool (status L in TOOL.T)  
**2:** The measured tool radius is compared to the tool radius R from TOOL.T. The control calculates the deviation from the stored value and writes it to Q parameter **Q116**. Nothing is entered under R or DR in the tool table.
- ▶ **Q260 Clearance height?:** Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height references the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**).  
Input range: -99999.9999 to 99999.9999
- ▶ **Q341 Probe the teeth? 0=no/1=yes:** Choose whether the control is to measure the individual teeth (maximum of 20 teeth).
- ▶ **For further information,** Page 279

## Example of new format

6 TOOL CALL 12 Z	
7 TCH PROBE 482 CAL. TOOL RADIUS	
Q340=1	;CHECK
Q260=+100	;CLEARANCE HEIGHT
Q341=1	;PROBING THE TEETH



Cycle **32** includes an additional parameter:



- **Parameter number for result?:** Parameter number in which the control saves the status of the measurement results:
  - 0.0:** Tool is within tolerance
  - 1.0:** Tool is worn (**RTOL** exceeded)
  - 2.0:** Tool is broken (**RBREAK** exceeded). If you do not wish to use the result of measurement in your NC program, answer the dialog prompt with **NO ENT**.

#### Measuring a rotating tool for the first time; old format

6 TOOL CALL 12 Z
7 TCH PROBE 32.0 CAL. TOOL RADIUS
8 TCH PROBE 32.1 CHECK: 0
9 TCH PROBE 32.2 HEIGHT: +120
10 TCH PROBE 32.3 PROBING THE TEETH: 0

#### Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

6 TOOL CALL 12 Z
7 TCH PROBE 32.0 CAL. TOOL RADIUS
8 TCH PROBE 32.1 CHECK: 1 Q5
9 TCH PROBE 32.2 HEIGHT: +120
10 TCH PROBE 32.3 PROBING THE TEETH: 1



## 9.5 Measuring tool length and radius (Cycle 33 or 483, ISO: G483, option 17)

### Application



Refer to your machine manual!

To measure both the length and radius of a tool, program the touch probe cycle **33** or **483** (see "Differences between Cycles 30 to 33 and Cycles 480 to 483", Page 265). This cycle is particularly suitable for the first measurement of tools, as it saves time when compared with individual measurement of length and radius. Via input parameters you can select the desired type of measurement:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth

#### 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**.



### Please note while programming:



The functionality of this cycle depends on the optional **probingCapability** machine parameter (no. 122723). (This parameter includes the possibility of tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.)

Cylindrical tools with diamond surfaces can be measured while the spindle is stationary. To do so, define the number of teeth **CUT** as 0 in the tool table and adjust machine parameter **CfgTT** (no. 122700). Refer to your machine manual.

### NOTICE

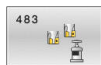
#### Danger of collision!

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 or turning, grinding, or dressing tools.



## Cycle parameters



- ▶ **Q340 Tool measurement mode (0-2)?**: Specify whether and how the determined data will be entered in the tool table.  
**0**: The measured tool length and the measured tool radius are written to columns L and R of the TOOL.T tool table, and the tool compensation is set to DL=0 and DR=0. If there is already a value in TOOL.T, it will be overwritten.  
**1**: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R in TOOL.T. The control calculates the deviation from the stored value and enters them into TOOL.T as the delta values DL and DR. The deviation is also available in Q parameters **Q115** and **Q116**. If the delta value is greater than the permissible tool length or radius tolerance for wear or breakage, the control will lock the tool (status L in TOOL.T)  
**2**: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R from TOOL.T. The control calculates the deviation from the stored values and writes it to Q parameter **Q115** or **Q116**. Nothing is entered under L, R, or DL, DR in the tool table.
- ▶ **Q260 Clearance height?**: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height references the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from **safetyDistStylus**).  
Input range: -99999.9999 to 99999.9999
- ▶ **Q341 Probe the teeth? 0=no/1=yes**: Choose whether the control is to measure the individual teeth (maximum of 20 teeth).
- ▶ **For further information**, Page 283

### Example of new format

6 TOOL CALL 12 Z	
7 TCH PROBE 483 MEASURE TOOL	
Q340=1	;CHECK
Q260=+100	;CLEARANCE HEIGHT
Q341=1	;PROBING THE TEETH



Cycle **33** includes an additional parameter:



- **Parameter number for result?:** Parameter number in which the control saves the status of the measurement results:
  - 0.0:** Tool is within tolerance
  - 1.0:** Tool is worn (**LTOL** and/or **RTOL** exceeded)
  - 2.0:** Tool is broken (**LBREAK** and/or **RBREAK** exceeded). If you do not wish to use the result of measurement in your NC program, answer the dialog prompt with **NO ENT**.

#### Measuring a rotating tool for the first time; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 33.0 MEASURE TOOL
8 TCH PROBE 33.1 CHECK: 0
9 TCH PROBE 33.2 HEIGHT: +120
10 TCH PROBE 33.3 PROBING THE
    TEETH: 0
```

#### Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 33.0 MEASURE TOOL
8 TCH PROBE 33.1 CHECK: 1 Q5
9 TCH PROBE 33.2 HEIGHT: +120
10 TCH PROBE 33.3 PROBING THE
    TEETH: 1
```



## 9.6 CALIBRATE IR TT (Cycle 484, ISO: G484, option 17)

### Application

With Cycle **484**, you can calibrate your tool touch probe, e.g. the wireless infrared TT 460 tool touch probe. The calibration process is either fully automatic or semi-automatic, depending on the parameter setting.

- **Semi-automatic**—stop before running: A dialog asks you to manually move the tool over the TT
- **Fully automatic**—no stop before running: Before using Cycle **484** you must move the tool over the TT

### Cycle run



Refer to your machine manual!

To calibrate the tool touch probe, program the touch probe cycle **484**. In input parameter **Q536**, you can specify whether you want to run the cycle semi-automatically or fully automatically.

### 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.



**Semi-automatic—stop before running**

- ▶ Insert the calibrating tool
- ▶ Define and start the calibration cycle
- > The control interrupts the calibration cycle and displays a dialog in a new window.
- ▶ You are prompted to manually position the calibrating tool above the center of the touch probe.
- > Ensure that the calibrating tool is located above the measuring surface of the probe contact.

**Fully automatic—no stop before running**

- ▶ Insert the calibrating tool
- ▶ Position the calibrating tool above the center of the touch probe.
- > Ensure that the calibrating tool is located above the measuring surface of the probe contact.
- ▶ Define and start the calibration cycle
- > The calibration cycle is executed without stopping.
- > The calibration process starts from the current position of the tool.



**Please note while programming:**

The functionality of this cycle depends on the optional **probingCapability** machine parameter (no. 122723). (This parameter includes the possibility of tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.)

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

To avoid collisions the tool must be pre-positioned before calling the cycle with **Q536=1**! The control also measures the center misalignment of the calibrating tool by rotating the spindle by 180° after the first half of the calibration cycle.

- Specify whether to stop before cycle start or run the cycle automatically without stopping.
- 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.

**Cycle parameters**

- **Q536 Stop before running (0=Stop)?**: Specify whether to stop before cycle start or run the cycle automatically without stopping:
  - 0**: Stop before running the cycle. You are prompted in a dialog to manually position the tool above the tool touch probe. After moving the tool to the approximate position above the tool touch probe, press **NC Start** to continue the calibration process or press the **CANCEL** soft key to cancel the calibration process
  - 1**: No stop before running the cycle. The control starts the calibration process from the current position. Before running Cycle **484**, you must position the tool above the tool touch probe.

**Example**

```
6 TOOL CALL 1 Z
```

```
7 TCH PROBE 484 CALIBRATE TT
```

```
Q536=+0 ;STOP BEFORE RUNNING
```



# 10

**Cycles: Special  
Functions**



## 10.1 Fundamentals

### Overview

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



- Press the **CYCL DEF** key



- Press the **SPECIAL CYCLES** soft key

Soft key	Cycle	Page
	<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 Cycle Programming
	<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 Cycle Programming
	<b>13 ORIENTATION</b> <ul style="list-style-type: none"> <li>■ Rotate spindle to a specific angle</li> </ul>	289
	<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 Cycle Programming
	<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 Cycle Programming
	<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 Cycle Programming
	<b>238 MEASURE MACHINE STATUS</b> <ul style="list-style-type: none"> <li>■ Determine the current machine status or test the measuring sequence</li> </ul>	<b>Further information:</b> User's Manual Cycle Programming
	<b>239 ASCERTAIN THE LOAD</b> <ul style="list-style-type: none"> <li>■ Selection for a weighing run</li> <li>■ Reset the load-dependent feedforward and controller parameters</li> </ul>	<b>Further information:</b> User's Manual Cycle Programming
	<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 Cycle Programming



## 10.2 SPINDLE ORIENTATION (Cycle 13, ISO: G36)

### Application



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

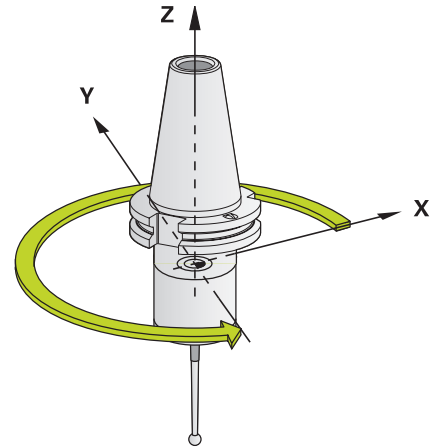
The control can control the main machine tool spindle and rotate it to a given angular position.

Oriented spindle stops are required for purposes such as:

- Tool changing systems with a defined tool change position
- Orientation of the transmitter/receiver window of HEIDENHAIN 3-D touch probes with infrared transmission

With **M19** or **M20**, the control positions the spindle at the angle of orientation defined in the cycle (depending on the machine).

If you program **M19** or **M20** without having defined Cycle **13** beforehand, the control positions the main spindle at an angle that has been set by the machine tool builder.



### Example

93 CYCL DEF 13.0 ORIENTATION

94 CYCL DEF 13.1 ANGLE 180

### Please note while programming:

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

### Cycle parameters



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







# 11

## Tables of Cycles



## 11.1 Table of cycles



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

ID of User's Manual for Programming of Machining Cycles: 1303427-xx

### Touch probe cycles

Cycle number	Cycle name	DEF active	CALL active	Page
0	REF. PLANE	■		168
1	POLAR PRESET	■		169
3	MEASURING	■		209
4	MEASURING IN 3-D	■		211
30	CALIBRATE TT	■		269
31	CAL. TOOL LENGTH	■		272
32	CAL. TOOL RADIUS	■		276
33	MEASURE TOOL	■		280
400	BASIC ROTATION	■		80
401	ROT OF 2 HOLES	■		83
402	ROT OF 2 STUDS	■		87
403	ROT IN ROTARY AXIS	■		92
404	SET BASIC ROTATION	■		101
405	ROT IN C AXIS	■		97
408	SLOT CENTER PRESET	■		150
409	RIDGE CENTER PRESET	■		154
410	PRESET INSIDE RECTAN	■		107
411	PRESET OUTS. RECTAN	■		111
412	PRESET INSIDE CIRCLE	■		115
413	PRESET OUTS. CIRCLE	■		120
414	PRESET OUTS. CORNER	■		125
415	PRESET INSIDE CORNER	■		130
416	PRESET CIRCLE CENTER	■		135
417	PRESET IN TS AXIS	■		140
418	PRESET FROM 4 HOLES	■		142
419	PRESET IN ONE AXIS	■		147
420	MEASURE ANGLE	■		170
421	MEASURE HOLE	■		173
422	MEAS. CIRCLE OUTSIDE	■		177



Cycle number	Cycle name	DEF active	CALL active	Page
423	MEAS. RECTAN. INSIDE	■		181
424	MEAS. RECTAN. OUTS.	■		185
425	MEASURE INSIDE WIDTH	■		188
426	MEASURE RIDGE WIDTH	■		191
427	MEASURE COORDINATE	■		194
430	MEAS. BOLT HOLE CIRC	■		197
431	MEASURE PLANE	■		200
441	FAST PROBING	■		213
450	SAVE KINEMATICS	■		234
451	MEASURE KINEMATICS	■		237
452	PRESET COMPENSATION	■		251
460	CALIBRATION OF TS ON A SPHERE	■		225
461	TS CALIBRATION OF TOOL LENGTH	■		217
462	CALIBRATION OF A TS IN A RING	■		219
463	TS CALIBRATION ON STUD	■		222
480	CALIBRATE TT	■		269
481	CAL. TOOL LENGTH	■		272
482	CAL. TOOL RADIUS	■		276
483	MEASURE TOOL	■		280
484	CALIBRATE IR TT	■		284
1410	PROBING ON EDGE	■		67
1411	PROBING TWO CIRCLES	■		73
1420	PROBING IN PLANE	■		62

#### Machining cycles

Cycle number	Cycle name	DEF active	CALL active	Page
13	ORIENTATION	■		289



## Index

### 3

3D Touch Probes..... 38

### A

About this manual..... 20  
Automatic presetting  
  Bolt circle..... 135  
  Center of 4 holes..... 142  
  Center of slot..... 150  
  Circular pocket (hole)..... 115  
  Circular stud..... 120  
  Fundamentals..... 104  
  Preset inside corner..... 130  
  Preset outside corner..... 125  
  rectangular pocket..... 107  
  rectangular stud..... 111  
  Ridge center..... 154  
  Single axis..... 147  
  Touch probe axis..... 140

### B

Basic rotation..... 80  
  Rot in rotary axis..... 92  
  Rot of 2 holes..... 83  
  Rot of 2 studs..... 87  
  Set directly..... 101

### C

Calibration cycles..... 215  
  Calibration of TS..... 225  
  Calibration of TS in a ring..... 219  
  Calibration on stud..... 222  
  TS Calibration of tool length..... 217  
Classification of results..... 165

### D

Determine inclined workpiece position  
  Probing in plane..... 62  
  Probing on edge..... 67  
  Probing two circles..... 73  
  Set basic rotation..... 101  
  Touch probe cycles 14xx  
    fundamentals..... 53  
  Touch probe cycles 4xx  
    fundamentals..... 79  
Determining inclined workpiece position  
  Basic rotation..... 80  
  Basic rotation over two holes..... 83  
  Basic rotation over two studs..... 87  
  Compensate basic rotation over a rotary axis..... 92  
  Rotation of C axis..... 97

### F

Fast probing..... 213  
Feature content level..... 26

### G

GLOBAL DEF..... 45

### I

Inclined workpiece position checking  
  Fundamentals..... 162  
  Measure angle..... 170  
  Measure bolt hole circle..... 197  
  Measure circle..... 177  
  Measure coordinate..... 194  
  Measure hole..... 173  
  Measure plane..... 200  
  Measure rectangular pocket..... 181  
  Measure rectangular stud..... 185  
  Measure ridge from outside..... 191  
  Measure slot width..... 188  
  Polar preset..... 169  
  Reference plane..... 168

### K

Kinematic measurement  
  Accuracy..... 242  
  Backlash..... 244  
  Prerequisites..... 232  
Kinematics measurement  
  Fundamentals..... 230  
  Hirth coupling..... 240  
  Measure kinematics..... 237  
  Preset compensation..... 251  
  Save kinematics..... 234  
KinematicsOpt..... 230

### M

Measure circle from inside..... 173  
Measure circle from outside..... 177  
Measure inside width..... 188  
Measure ridge width..... 191  
Measure slot width..... 188  
Measuring  
  Angle..... 170  
  Bolt hole circle..... 197  
  Circle from outside..... 177  
  Coordinate..... 194  
  Holes..... 173  
  Inside width..... 188  
  Plane..... 200  
  Rectangle from inside..... 181  
  Rectangle from outside..... 185  
  Ridge width..... 191  
  using Cycle 3..... 209  
Measuring in 3-D..... 211

### O

Option..... 23

### P

Positioning logic..... 44  
Probing feed rate..... 42

### R

Recording measurement results..... 163  
Rectangular pocket  
  Measuring..... 181  
Rectangular stud  
  Measuring..... 185

### S

Software option..... 23  
Spindle orientation..... 289

### T

Table of cycles..... 292  
  Touch probe cycles..... 292  
Tolerance monitoring..... 165  
Tool compensation..... 166  
Tool measurement  
  Fundamentals..... 264  
  IR TT calibration..... 284  
  Machine parameters..... 266  
  Measure tool length and radius..... 280  
  Tool length..... 272  
  Tool radius..... 276  
  TT calibration..... 269  
Tool monitoring..... 166  
Tool table..... 268  
Touch probe cycles 14xx  
  Evaluation of tolerances..... 60  
  Fundamentals..... 53  
  Probing in plane..... 62  
  Probing on edge..... 67  
  Probing two circles..... 73  
  Semi-automatic mode..... 55  
  Transfer an actual position..... 61  
Touch probe data..... 49  
Touch-probe table..... 48



# HEIDENHAIN

## DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ +49 8669 31-0

FAX +49 8669 32-5061

E-mail: [info@heidenhain.de](mailto:info@heidenhain.de)

**Technical support** FAX +49 8669 32-1000

**Measuring systems** ☎ +49 8669 31-3104

E-mail: [service.ms-support@heidenhain.de](mailto:service.ms-support@heidenhain.de)

**NC support** ☎ +49 8669 31-3101

E-mail: [service.nc-support@heidenhain.de](mailto:service.nc-support@heidenhain.de)

**NC programming** ☎ +49 8669 31-3103

E-mail: [service.nc-pgm@heidenhain.de](mailto:service.nc-pgm@heidenhain.de)

**PLC programming** ☎ +49 8669 31-3102

E-mail: [service.plc@heidenhain.de](mailto:service.plc@heidenhain.de)

**APP programming** ☎ +49 8669 31-3106

E-mail: [service.app@heidenhain.de](mailto:service.app@heidenhain.de)

[www.heidenhain.de](http://www.heidenhain.de)

[www.klartext-portal.com](http://www.klartext-portal.com)

The Information Site for  
HEIDENHAIN Controls

### Klartext App

The Klartext on Your  
Mobile Device

Google  
Play Store

Apple  
App Store



## Touch probes from HEIDENHAIN

help you reduce non-productive time and improve the  
dimensional accuracy of the finished workpieces.

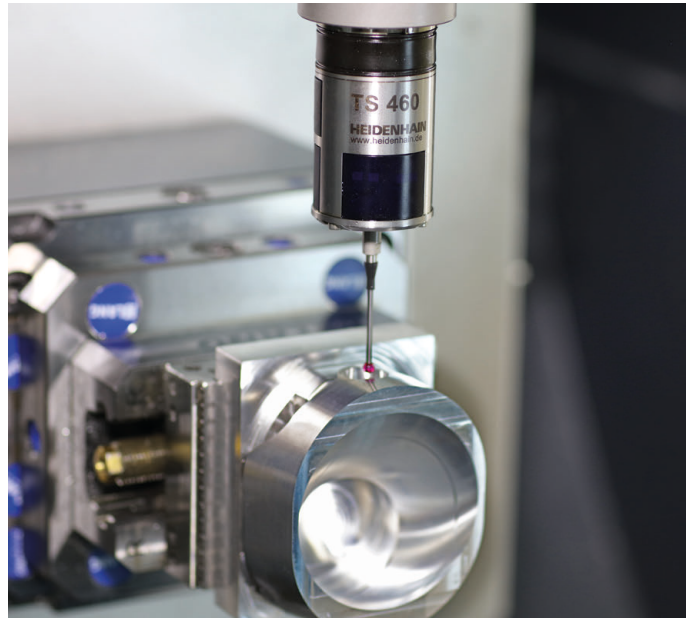
### Workpiece touch probes

**TS 248, TS 260** Signal transmission by cable

**TS 460** Radio or infrared transmission

**TS 640, TS 740** Infrared transmission

- Workpiece alignment
- Preset setting
- Workpiece measurement



### Tool touch probes

**TT 160** Signal transmission by cable

**TT 460** Infrared transmission

- Tool measurement
- Wear monitoring
- Tool breakage detection

