HEIDENHAIN

TNC 640

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

NC Software
340590-11
340591-11
340595-11

English (en)
01/2021

Contents

1 Fundamentals....21
2 Fundamentals / Overviews....37
3 Using Touch Probe Cycles....41
4 Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment....55
5 Touch Probe Cycles: Automatic Presetting....107
6 Touch Probe Cycles: Automatic Workpiece Inspection....165
7 Touch Probe Cycles: Special Functions....213
8 Touch Probe Cycles: Automatic Kinematics Measurement....241
9 Touch Probe Cycles: Automatic Tool Measurement....281
10 VSC: camera-based setup control (option 136)....311
11 Cycles: Special Functions....331
12 Tables of Cycles....335

1 Fundamentals 21

1.1 About this manual 22

1.2 Control model, software, and features 24

Software options 25

New or modified cycle functions of software 34059x-11 31

2 Fundamentals / Overviews 37

2.1 Introduction 38

2.2 Available cycle groups 39

Overview of fixed cycles 39
Overview of touch probe cycles 40

3 Using Touch Probe Cycles 41

3.1 General information about touch probe cycles 42

Method of function 42
Consideration of a basic rotation in the Manual operation mode 42
Touch probe cycles in the Manual Operation and Electronic Handwheel operating modes 42
Touch probe cycles for automatic operation 43

3.2 Before You Start Working with Touch Probe Cycles 45

Maximum traverse to touch point: DIST in touch probe table 45
Set-up clearance to touch point: SET_UP in touch probe table 45
Orient the infrared touch probe to the programmed probe direction: TRACK in touch probe table 45
Touch trigger probe, probing feed rate: F in touch probe table 46
Touch trigger probe, rapid traverse for positioning: FMAX 46
Touch trigger probe, rapid traverse for positioning: F_PREPOS in touch probe table 46
Executing touch probe cycles 47

3.3 Program defaults for cycles 49

Overview 49
Entering GLOBAL DEF definitions 50
Using GLOBAL DEF information 50
Global data valid everywhere 51
Global data for probing functions 51

3.4 Touch-probe table 52

General information 52
Editing touch probe tables 52
Touch probe data 53

4 Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment 55

4.1 Overview 56

4.2 Touch probe cycles 14xx: fundamentals 57

Functionalities common to touch probe cycles 14xx for measuring rotations 57
Semi-automatic mode 59
Evaluation of tolerances 64
Transferring the actual position 65

4.3 PROBING IN PLANE (Cycle 1420, DIN/ISO: G1420) 66

Application 66
Please note while programming! 67
Cycle parameters 68

4.4 PROBING ON EDGE (Cycle 1410, DIN/ISO: G1410) 71

Application 71
Please note while programming! 73
Cycle parameters 74

4.5 PROBING TWO CIRCLES (Cycle 1411, DIN/ISO: G1411) 77

Application 77
Please note while programming! 79
Cycle parameters 80

4.6 Touch probe cycles 4xx: fundamentals 83

Characteristics common to all touch probe cycles for measuring workpiece misalignment 83

4.7 BASIC ROTATION (Cycle 400, DIN/ISO: G400) 84

Application 84
Please note while programming: 84
Cycle parameters 85

4.8 ROT. OF 2 HOLES (Cycle 401, DIN/ISO: G401) 87

Application 87
Please note while programming: 87
Cycle parameters 88

4.9 ROT. OF 2 STUDS (Cycle 402, DIN/ISO: G402) 91

Application 91
Please note while programming: 92
Cycle parameters 93

4.10 ROT. IN ROTARY AXIS (Cycle 403, DIN/ISO: G403) 96

Application 96
Please note while programming: 97
Cycle parameters 98

4.11 ROT. IN C AXIS (Cycle 405, DIN/ISO: G405.....101

Application.....101
Please note while programming:.....102
Cycle parameters.....103

4.12 SET BASIC ROTATION (Cycle 404, DIN/ISO: G404).....105

Application.....105
Cycle parameters.....105

4.13 Example: Determining a basic rotation from two holes.....106

5 Touch Probe Cycles: Automatic Presetting 107

5.1 Fundamentals 108

Overview 108
Characteristics common to all touch probe cycles for presetting 110

5.2 PRESET INSIDE RECTANGLE (Cycle 410, DIN/ISO: G410) 111

Application 111
Please note while programming: 112
Cycle parameters 113

5.3 PRESET OUTS. RECTAN (Cycle 411, DIN/ISO: G411) 115

Application 115
Please note while programming: 116
Cycle parameters 117

5.4 PRESET INSIDE CIRCLE (Cycle 412, DIN/ISO: G412) 119

Application 119
Please note while programming: 120
Cycle parameters 121

5.5 PRESET OUTS. CIRCLE (Cycle 413, DIN/ISO: G413) 124

Application 124
Please note while programming: 125
Cycle parameters 126

5.6 PRESET OUTSIDE CORNER (Cycle 414, DIN/ISO: G414) 129

Application 129
Please note while programming: 130
Cycle parameters 131

5.7 PRESET INSIDE CORNER (Cycle 415, DIN/ISO: G415) 134

Application 134
Please note while programming: 135
Cycle parameters 136

5.8 PRESET CIRCLE CENTER (Cycle 416, DIN/ISO: G416) 139

Application 139
Please note while programming: 140
Cycle parameters 141

5.9 PRESET IN TS AXIS (Cycle 417, DIN/ISO: G417) 144

Application 144
Please note while programming: 144
Cycle parameters 145

5.10 PRESET FROM 4 HOLES (Cycle 418, DIN/ISO: G418)..... 146

Application..... 146
Please note while programming:..... 147
Cycle parameters..... 148

5.11 PRESET IN ONE AXIS (Cycle 419, DIN/ISO: G419)..... 151

Application..... 151
Please note while programming:..... 151
Cycle parameters..... 152

5.12 SLOT CENTER PRESET (Cycle 408, DIN/ISO: G408)..... 154

Application..... 154
Please note while programming:..... 155
Cycle parameters..... 156

5.13 RIDGE CENTER PRESET (Cycle 409, DIN/ISO: G409)..... 158

Application..... 158
Please note while programming:..... 159
Cycle parameters..... 160

5.14 Example: Presetting in center of a circular segment and on top surface of workpiece..... 162

5.15 Example: Presetting on top surface of workpiece and in center of a bolt hole circle..... 163

6 Touch Probe Cycles: Automatic Workpiece Inspection 165

6.1 Fundamentals 166

Overview 166
Recording the results of measurement 167
Measurement results in Q parameters 169
Classification of results 169
Tolerance monitoring 169
Tool monitoring 170
Reference system for measurement results 171

6.2 REFERENCE PLANE (Cycle 0, DIN/ISO: G55) 172

Application 172
Please note while programming: 172
Cycle parameters 172

6.3 POLAR PRESET (Cycle 1) 173

Application 173
Please note while programming: 173
Cycle parameters 173

6.4 MEASURE ANGLE (Cycle 420, DIN/ISO: G420) 174

Application 174
Please note while programming: 174
Cycle parameters 175

6.5 MEASURE HOLE (Circle 421, DIN/ISO: G421) 177

Application 177
Please note while programming: 178
Cycle parameters 179

6.6 MEASURE CIRCLE OUTSIDE (Cycle 422, DIN/ISO: G422) 182

Application 182
Please note while programming: 183
Cycle parameters 184

6.7 MEASURE RECTANGLE INSIDE (Cycle 423, DIN/ISO: G423) 187

Application 187
Please note while programming: 188
Cycle parameters 189

6.8 MEASURE RECTANGLE OUTSIDE (Cycle 424, DIN/ISO: G424) 191

Application 191
Please note while programming: 191
Cycle parameters 192

6.9 MEASURE INSIDE WIDTH (Cycle 425, DIN/ISO: G425)..... 194

Application..... 194
Please note while programming:..... 194
Cycle parameters..... 195

6.10 MEASURE RIDGE WIDTH (Cycle 426, DIN/ISO: G426)..... 197

Application..... 197
Please note while programming:..... 197
Cycle parameters..... 198

6.11 MEASURE COORDINATE (Cycle 427, DIN/ISO: G427)..... 200

Application..... 200
Please note while programming:..... 200
Cycle parameters..... 201

6.12 MEAS. BOLT HOLE (Cycle 430, DIN/ISO: G430)..... 203

Application..... 203
Please note while programming:..... 203
Cycle parameters..... 204

6.13 MEASURE PLANE (Cycle 431, DIN/ISO: G431)..... 206

Application..... 206
Please note while programming:..... 207
Cycle parameters..... 207

6.14 Programming Examples..... 209

Example: Measuring and reworking a rectangular stud..... 209
Example: Measuring a rectangular pocket and recording the results..... 211

7 Touch Probe Cycles: Special Functions 213

7.1 Fundamentals 214
Overview 214

7.2 MEASURING (Cycle 3) 215
Application 215
Please note while programming: 215
Cycle parameters 216

7.3 MEASURING IN 3-D (Cycle 4) 217
Application 217
Please note while programming: 217
Cycle parameters 218

7.4 PROBING IN 3-D (Cycle 444, ISO: G444) 219
Application 219
Please note while programming! 221
Cycle parameters 222

7.5 FAST PROBING (Cycle 441, DIN/ISO: G441) 224
Application 224
Please note while programming: 224
Cycle parameters 225

7.6 Calibrating a touch trigger probe 226

7.7 Displaying calibration values 227

7.8 TS CALIBRATION OF TOOL LENGTH (Cycle 461, DIN/ISO: G461) 228

7.9 CALIBRATION OF A TS IN A RING (Cycle 462, DIN/ISO: G462) 230

7.10 TS CALIBRATION ON STUD (Cycle 463, DIN/ISO: G463) 233

7.11 CALIBRATION OF TS ON A SPHERE (Cycle 460, DIN/ISO: G460) 236

8 Touch Probe Cycles: Automatic Kinematics Measurement 241

8.1 Kinematics measurement with TS touch probes (option 48) 242

Fundamentals 242
Overview 243

8.2 Prerequisites 244

Please note while programming: 245

8.3 SAVE KINEMATICS (Cycle 450, DIN/ISO: G450, option 48) 246

Application 246
Please note while programming: 246
Cycle parameters 247
Log function 247
Notes on data management 248

8.4 MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option 48) 249

Application 249
Positioning direction 251
Machines with Hirth-coupled axes 252
Example calculation of measuring positions for an A axis: 252
Choice of number of measuring points 253
Choice of the calibration sphere position on the machine table 254
Notes on the accuracy 254
Notes on various calibration methods 255
Backlash 256
Please note while programming: 257
Cycle parameters 258
Various modes (Q406) 261
Log function 262

8.5 PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option 48) 263

Application 263
Please note while programming: 265
Cycle parameters 266
Adjustment of interchangeable heads 269
Drift compensation 271
Log function 273

8.6 KINEMATICS GRID (Cycle 453, DIN/ISO: G453, option 48) 274

Application 274
Various modes (Q406) 275
Choice of the calibration sphere position on the machine table 275
Please note while programming: 276
Cycle parameters 277
Log function 279

9 Touch Probe Cycles: Automatic Tool Measurement 281

9.1 Fundamentals 282

Overview 282
Differences between Cycles 30 to 33 and Cycles 480 to 483 283
Setting machine parameters 284
Entries in the tool table for milling and turning tools 286

9.2 CALIBRATE TT (Cycle 30 or 480, ISO: G480) 287

Application 287
Please note while programming: 288
Cycle parameters 289

9.3 Measuring tool length (Cycle 31 or 481, DIN/ISO: G481) 290

Application 290
Please note while programming: 291
Cycle parameters 292

9.4 Measuring a tool radius (Cycle 32 or 482, ISO: G482) 294

Application 294
Please note while programming: 295
Cycle parameters 296

9.5 Measuring tool length and radius (Cycle 33 or 483, ISO: G483) 298

Application 298
Please note while programming: 299
Cycle parameters 300

9.6 CALIBRATE IR TT (Cycle 484, ISO: G484) 302

Application 302
Cycle run 302
Please note while programming: 304
Cycle parameters 304

9.7 MEASURE LATHE TOOL (Cycle 485, ISO: G485, option 50) 305

Application 305
Please note while programming! 308
Cycle parameters 309

10 VSC: camera-based setup control (option 136)..... 311

10.1 Camera-based Visual Setup Control (VSC) (option 136)..... 312

Fundamentals..... 312
Managing monitoring data..... 314
Overview..... 315
Configuration..... 316
Defining the monitoring area..... 317
Result of the image evaluation..... 318

10.2 GLOBAL WORKING SPACE (Cycle 600, ISO: G600, option 136)..... 319

Application..... 319
Creating reference images..... 320
Monitoring phase..... 321
Please note while programming!..... 322
Cycle parameters..... 323

10.3 LOCAL WORKING SPACE (Cycle 601, ISO: G601, option 136)..... 324

Application..... 324
Creating reference images..... 324
Monitoring phase..... 326
Please note while programming!..... 327
Cycle parameters..... 328

10.4 Possible queries..... 329

11 Cycles: Special Functions 331

11.1 Fundamentals 332

Overview 332

11.2 SPINDLE ORIENTATION (Cycle 13, ISO: G36) 334

Application 334
Please note while programming: 334
Cycle parameters 334

12 Tables of Cycles..... 335

12.1 Table of cycles..... 336

Touch probe cycles.....336

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 640	340590-11
TNC 640 E	340591-11
TNC 640 Programming station	340595-11

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: 1303406-xx

User's Manual:

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

Conversational Programming User's Manual ID: 892903-xx

ISO Programming User's Manual ID: 892909-xx

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

Software options

The TNC 640 features various software options, each of which can be enabled separately by your machine tool builder. The respective options provide the functions listed below:

Additional Axis (options 0 to 7)	
Additional axis	Additional control loops 1 to 8
Advanced Function Set 1 (option 8)	
Advanced functions (set 1)	**Machining with rotary tables** ■ Cylindrical contours as if in two axes ■ Feed rate in distance per minute **Coordinate conversions:** Tilting the working plane
Advanced Function Set 2 (option 9)	
Advanced functions (set 2) Export license required	**3-D machining:** ■ 3-D tool compensation through surface-normal vectors ■ Using the electronic handwheel to change the angle of the swivel head during program run; the position of the tool point remains unchanged (TCPM = Tool Center Point Management) ■ Keeping the tool normal to the contour ■ Tool radius compensation normal to the tool direction ■ Manual traverse in the active tool-axis system **Interpolation:** Linear in > 4 axes (export license required)
HEIDENHAIN DNC (option 18)	
	Communication with external PC applications over COM component
Dynamic Collision Monitoring – DCM (option 40)	
Dynamic Collision Monitoring	■ The machine manufacturer defines objects to be monitored ■ Warning in Manual operation ■ Collision monitoring in the Test Run mode ■ Program interrupt in Automatic operation ■ Includes monitoring of 5-axis movements
CAD Import (option 42)	
CAD import	■ Support for DXF, STEP and IGES ■ Adoption of contours and point patterns ■ Simple and convenient specification of presets ■ Selecting graphical features of contour sections from conversational programs
Global PGM Settings – GPS (option 44)	
Global program settings	■ Superimposition of coordinate transformations during program run ■ Handwheel superimpositioning

Adaptive Feed Control – AFC (option 45)	
Adaptive Feed Control	**Milling:** ■ Recording the actual spindle power by means of a teach-in cut ■ Defining the limits of automatic feed rate control ■ Fully automatic feed control during program run **Turning (option 50):** ■ Cutting force monitoring during machining
KinematicsOpt (option 48)	
Optimizing the machine kinematics	■ Backup/restore active kinematics ■ Test active kinematics ■ Optimize active kinematics
Mill-Turning (option 50)	
Milling and turning modes	**Functions:** ■ Switching between Milling/Turning mode of operation ■ Constant surface speed ■ Tool-tip radius compensation ■ Turning cycles ■ Cycle **GEAR HOBBING** (options 50 and 131)
KinematicsComp (option 52)	
Three-dimensional compensation	Compensation of position and component errors
OPC UA NC Server (1 to 6) (options 56 to 61)	
Standardized interface	The OPC UA NC Server provides a standardized interface (OPC UA) for external access to the control's data and functions These software options allow you to create up to six parallel client connections
3D-ToolComp (option 92)	
3-D tool radius compensation depending on the tool's contact angle Export license required	■ Compensate the deviation of the tool radius depending on the tool's contact angle ■ Compensation values in a separate compensation value table ■ Prerequisite: Working with surface normal vectors (**LN** blocks)
Extended Tool Management (option 93)	
Extended tool management	Python-based
Advanced Spindle Interpolation (option 96)	
Interpolating spindle	**Interpolation turning:** ■ Cycle **COUPLG.TURNG.INTERP.** ■ Cycle **CONTOUR.TURNG.INTRP.**
Spindle Synchronism (option 131)	
Spindle synchronization	■ Synchronization of milling spindle and turning spindle ■ Cycle **GEAR HOBBING** (options 50 and 131)

Remote Desktop Manager (option 133)	
Remote operation of external computer units	■ Windows on a separate computer unit ■ Incorporated in the control's interface
Synchronizing Functions (option 135)	
Synchronization functions	**Real Time Coupling (RTC):** Coupling of axes
Visual Setup Control – VSC (option 136)	
Camera-based monitoring of the setup situation	■ Record the setup situation with a HEIDENHAIN camera system ■ Visual comparison of planned and actual status in the workspace
State Reporting Interface – SRI (option 137)	
HTTP accesses to the control status	■ Reading out the times of status changes ■ Reading out the active NC programs
Cross Talk Compensation – CTC (option 141)	
Compensation of axis couplings	■ Determination of dynamically caused position deviation through axis acceleration ■ Compensation of the TCP (**T**ool **C**enter **P**oint)
Position Adaptive Control – PAC (option 142)	
Adaptive position control	■ Adaptation of the control parameters depending on the position of the axes in the working space ■ Adaptation of the control parameters depending on the speed or acceleration of an axis
Load Adaptive Control – LAC (option 143)	
Adaptive load control	■ Automatic determination of workpiece weight and frictional forces ■ Adaptation of the control parameters depending on the current mass of the workpiece
Active Chatter Control – ACC (option 145)	
Active chatter control	Fully automatic function for chatter control during machining
Machine Vibration Control – MVC (option 146)	
Vibration damping for machines	Damping of machine oscillations for improving the workpiece surface quality through the following functions: ■ Active Vibration Damping (**AVD**) ■ Frequency Shaping Control (**FSC**)
Batch Process Manager (option 154)	
Batch process manager	Planning of production orders
Component Monitoring (option 155)	
Component monitoring without external sensors	Monitoring configured machine components for overload

Grinding (option 156)	
Jig grinding	■ Reciprocating stroke cycles ■ Cycles for dressing ■ Support of the "dressing tool" and "grinding tool" tool types
Gear Cutting (option 157)	
Machining gear systems	■ Cycle **DEFINE GEAR** ■ Cycle **GEAR HOBBING** ■ Cycle **GEAR SKIVING**
Advanced Function Set Turning (option 158)	
Advanced turning functions	Cycle **TURNING SIMULTANEOUS FINISHING**
Opt. contour milling (option 167)	
Optimized contour cycles	Cycles for machining any pockets and islands using trochoidal milling

Further options available

HEIDENHAIN offers further hardware enhancements and software options that can be configured and implemented only by your machine 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 informationGeneral 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 34059x-11" 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 640. Even if new optional parameters ("Optional parameters") have been added to existing cycles, you will generally be able to run your NC programs as usual. This is achieved because the stored default value will be used. The other way round, if you want to run an NC program created with a new software version on an older control, you can delete the respective optional Q parameters from the cycle definition with the NO ENT key. In this way you can ensure that the NC program is be downward compatible. If NC blocks contain invalid elements, the control will mark them as ERROR blocks when the file is opened.

New or modified cycle functions of software 34059x-11

Overview of new and modified software functions

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

ID: 1322095-xx

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.
- Cycle **1016 DRESSING OF CUP WHEEL** (ISO: **G1016**, option 156)

  You use this cycle to dress the front face of a cup wheel. You define the optional angle for the relief cut in the tool table. You can use this cycle only in dressing mode (**FUNCTION MODE DRESS**).
- Cycle **1025 GRINDING CONTOUR** (ISO: **G1025**, option 156)

  The control uses this cycle to grind open or closed contours. You define the contour in a subprogram and select it with Cycle **14 CONTOUR** (ISO: **G37**).
- Cycle **882 SIMULTANEOUS ROUGHING FOR TURNING** (ISO: **G882**, option 50, option 158)

  This cycle enables the roughing of turning contours using different inclination angles. This allows you to machine undercut contours, for example, with just one tool. In addition, you can increase the service life of the tool by utilizing a large area of the indexable insert.

  You define the contour in a subprogram and select it with Cycle **14 CONTOUR** (ISO: **G37**) or the **SEL CONTOUR** function.
- 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**) 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**), 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**)
  - Cycle 23 **FLOOR FINISHING** (ISO: **G123**)
  - Cycle 24 **SIDE FINISHING** (ISO: **G124**)
  - Cycle 233 **FACE MILLING** (ISO: **G233**)
  - Cycle 272 **OCM ROUGHING** (ISO: **G272**, option 167)
  - Cycle 273 **OCM FINISHING FLOOR** (ISO: **G273**, option 167)
  - Cycle 274 **OCM FINISHING SIDE** (ISO: **G274**, option 167)
- Cycles **251 RECTANGULAR POCKET** (ISO: **G251**), **252 CIRCULAR POCKET** (ISO: **G252**), and **272 OCM ROUGHING** (ISO: **G272**, option 167) take into consideration a cutting width defined in the **RCUTS** column when calculating the plunging path.
- Cycles **208 BORE MILLING** (ISO: **G208**), **253 SLOT MILLING** (ISO: **G208**), and **254 CIRCULAR SLOT** (ISO: **G254**) 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 INFEED 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
- Cycle **1010 DRESSING DIAMETER** (ISO: **G1010**, option 156) uses the value from parameter **Q1018 DRESSING FEED RATE** for the infeed movement.
- In parameter **QS1000 PROFILE PROGRAM** of Cycle **1015 PROFILE DRESSING** (ISO: **G1015**, option 156), you can use the **SELECT FILE** soft key to select the NC program for the profile of the grinding tool.
  Further information: User's Manual for Programming of Machining Cycles

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

New functions

- Cycle **485 MEASURE LATHE TOOL** (ISO: **G485**, option 50)

  This cycle enables you to measure lathe tools with a tool touch probe. This cycle can only be executed in the **FUNCTION MODE MILL** machining mode. Furthermore, you need a tool touch probe with a cuboid probe contact.

  Further information: "MEASURE LATHE TOOL (Cycle 485, ISO: G485, option 50)", Page 305

Changed Functions

- Cycles **480 CALIBRATE TT** (ISO: **G480**) and **484 CALIBRATE IR TT** (ISO: **G484**) can be used to calibrate a tool touch probe with cuboid probe contacts.

  Further information: "CALIBRATE TT (Cycle 30 or 480, ISO: G480)", Page 287

  Further information: "CALIBRATE IR TT (Cycle 484, ISO: G484)", Page 302
- For rotating tools, Cycle **483 MEASURE TOOL** (ISO: **G483**) first measures the tool length and then the tool radius.

  Further information: "Measuring tool length and radius (Cycle 33 or 483, ISO: G483)", Page 298
- By default, Cycles **1410 PROBING ON EDGE** (ISO: **G1410**) and **1411 PROBING TWO CIRCLES** (ISO: **G1411**) calculate the basic rotation in the input coordinate system (I-CS). If the axis angles and tilt angles don't coincide, the cycles calculate the basic rotation in the workpiece coordinate system (W-CS).

  Further information: "PROBING ON EDGE (Cycle 1410, DIN/ISO: G1410)", Page 71

  Further information: "PROBING TWO CIRCLES (Cycle 1411, DIN/ISO: G1411)", Page 77

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

CYCL DEF

- Press the **CYCL DEF** key

Soft key	Cycle group	Page
DRILLING/ THREAD	Cycles for pecking, reaming, boring and counterboring	**Further information:** User's Manual for Programming of Machining Cycles
DRILLING/ THREAD	Cycles for tapping, thread cutting and thread milling	**Further information:** User's Manual for Programming of Machining Cycles
POCKETS/ STUDS/ SLOTS	Cycles for milling pockets, studs, slots, and face milling	**Further information:** User's Manual for Programming of Machining Cycles
COORD. TRANSF.	Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	**Further information:** User's Manual for Programming of Machining Cycles
SL CYCLES	SL (Subcontour List) cycles for machining contours that consist of multiple overlapping subcontours as well as cycles for cylinder surface machining and trochoidal milling	**Further information:** User's Manual for Programming of Machining Cycles
PATTERN	Cycles for producing point patterns, such as circular or linear hole patterns, DataMatrix code	**Further information:** User's Manual for Programming of Machining Cycles
TURNING	Cycles for turning and gear hobbing	**Further information:** User's Manual for Programming of Machining Cycles
SPECIAL CYCLES	Special cycles: dwell time, program call, oriented spindle stop, engraving, tolerance, interpolation turning, determining the load, gear cycles	**Further information:** User's Manual for Programming of Machining Cycles
GRINDING	Cycles for grinding operations and sharpening of grinding tools	**Further information:** User's Manual for Programming of Machining Cycles

▷

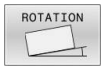
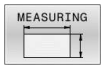
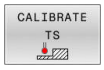
- If required, switch to machine-specific machining cycles

The machine tool builder can integrate these types of machining cycles.

Overview of touch probe cycles

TOUCH PROBE

- Press the **TOUCH PROBE** key.

Soft key	Cycle group	Page
	Cycles for automatic measurement and compensation of workpiece misalignment	56
	Cycles for automatic workpiece presetting	108
	Cycles for automatic workpiece inspection	166
	Special cycles	214
	Touch probe calibration	226
	Cycles for automatic kinematics measurement	243
	Cycles for automatic tool measurement (enabled by the machine manufacturer)	282
	Cycles for VSC (camera-based setup control, option 136)	315

▷

- 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.

Touch probe functions deactivate the **Global Program Settings** temporarily.

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

Method of function

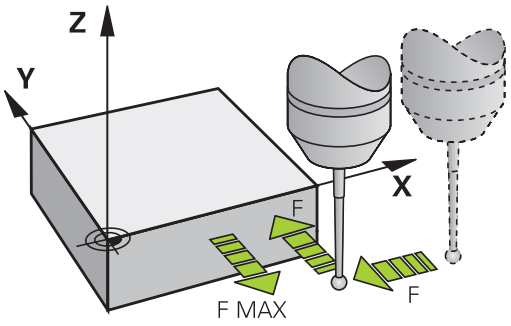
Whenever the control runs a touch probe cycle, the 3-D touch probe approaches the workpiece in one linear axis. This is also true during an active basic rotation or with a tilted working plane. The machine tool builder will determine the probing feed rate in a machine parameter.

Further information: "Before You Start Working with Touch Probe Cycles", Page 45

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

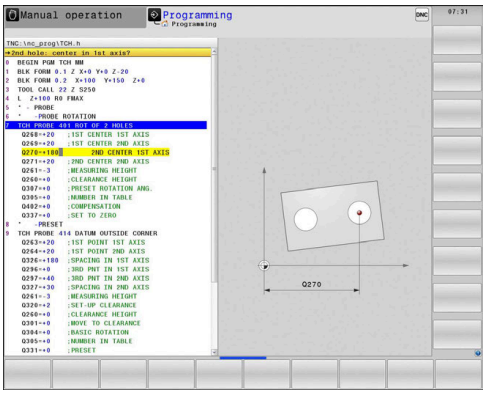
Touch probe cycles for automatic operation

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).



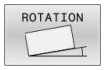
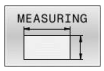
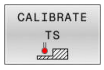
Defining the touch probe cycle in the Programming mode of operation

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	56
	Cycles for automatic workpiece presetting	108
	Cycles for automatic workpiece inspection	166
	Special cycles	214
	Calibrate TS	226
	Kinematics	243
	Cycles for automatic tool measurement (enabled by the machine manufacturer)	282
	Camera-based monitoring (option 136 VSC)	315

NC blocks

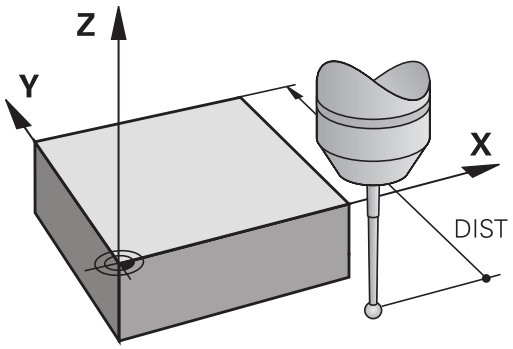
```
5 TCH PROBE 410  PRESET INSIDE RECTAN.
    Q321=+50   ;CENTER IN 1ST AXIS
    Q322=+50   ;CENTER IN 2ND AXIS
    Q323=60    ;FIRST SIDE LENGTH
    Q324=20    ;2ND SIDE LENGTH
    Q261=-5    ;MEASURING HEIGHT
    Q320=0     ;SET-UP CLEARANCE
    Q260=+20   ;CLEARANCE HEIGHT
    Q301=0     ;MOVE TO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

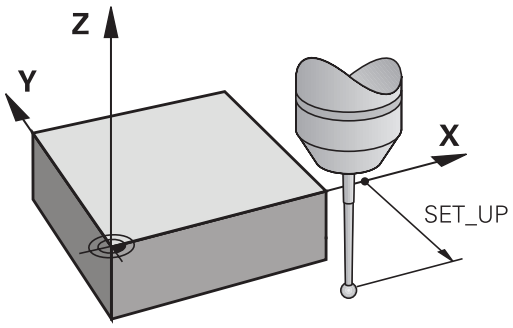
Maximum traverse to touch point: DIST in touch probe table

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:

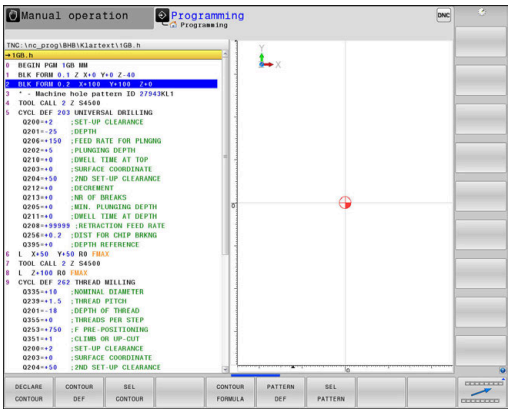
- If the current coordinate of the south pole of the stylus is less than the coordinate of the clearance height (as defined in the cycle), the control first retracts the touch probe in the touch probe axis to clearance height and then positions it in the working plane to the first touch point.
- If the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

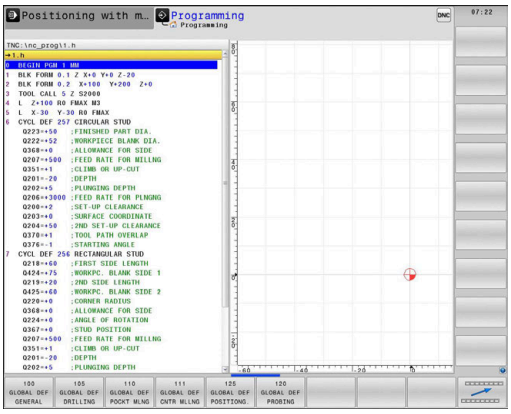
The following GLOBAL DEF functions are available:



Soft key	Machining patterns	Page
100 GLOBAL DEF GENERAL	GLOBAL DEF COMMON Definition of generally valid cycle parameters	51
105 GLOBAL DEF DRILLING	GLOBAL DEF DRILLING Definition of specific drilling cycle parameters	**Further information:** User's Manual for Programming of Machining Cycles
110 GLOBAL DEF POCKT MLNG	GLOBAL DEF POCKET MILLING Definition of specific pocket-milling cycle parameters	**Further information:** User's Manual for Programming of Machining Cycles
111 GLOBAL DEF CNTR MLLNG	GLOBAL DEF CONTOUR MILLING Definition of specific contour milling cycle parameters	**Further information:** User's Manual for Programming of Machining Cycles
125 GLOBAL DEF POSITIONG.	GLOBAL DEF POSITIONING Definition of the positioning behavior for **CYCL CALL PAT**	**Further information:** User's Manual for Programming of Machining Cycles
120 GLOBAL DEF PROBING	GLOBAL DEF PROBING Definition of specific touch probe cycle parameters	51

Entering GLOBAL DEF definitions

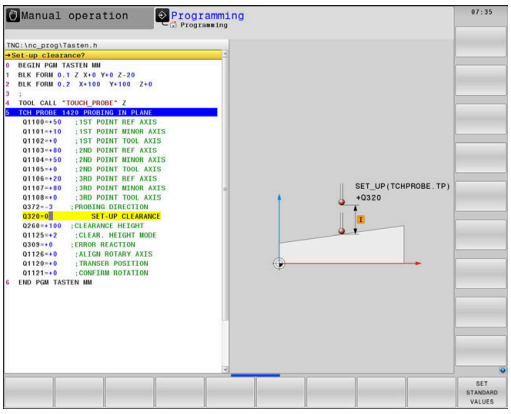
Proceed as follows:



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

Using GLOBAL DEF information

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



Proceed as follows:

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

NOTICE

Danger of collision!

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

- 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** as well as for Cycles **880, 1025** and touch probe cycles **451**, **452**, **453**

- **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**, **286**, **287**, **880**, **1025**, **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

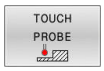
Various data are stored in the touch probe table that defines the probe behavior during the probing process. If you use several touch probes on your machine tool, you can save separate data for each touch probe.

> 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

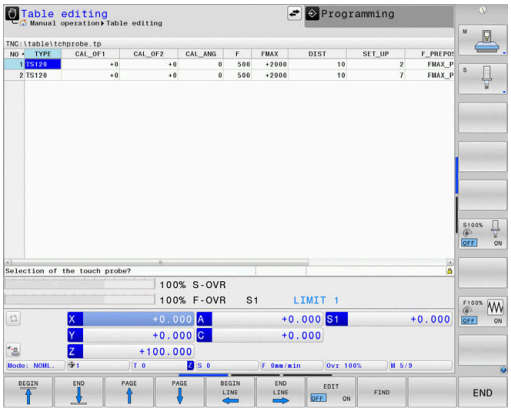


- Press the **TOUCH PROBE** soft key

> 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
NO	Number of the touch probe: Enter this number in the tool table (column: **TP_NO**) under the appropriate tool entries	–
TYPE	Selection of the touch probe used	**Selection of the touch probe?**
CAL_OF1	Offset of the touch probe axis to the spindle axis in the principal axis	**TS center misalignmt. ref. axis? [mm]**
CAL_OF2	Offset of the touch probe axis to the spindle axis in the secondary axis	**TS center misalignmt. aux. axis? [mm]**
CAL_ANG	Prior to calibrating or probing the control aligns the touch probe with the spindle angle (if spindle orientation is possible)	**Spindle angle for calibration?**
F	Feed rate at which the control will probe the workpiece **F** can never be greater than set in the optional **maxTouch-Feed** machine parameter (no. 122602).	**Probing feed rate? [mm/min]**
FMAX	Feed rate at which the touch probe is pre-positioned and is positioned between the measuring points	**Rapid traverse in probing cycle? [mm/min]**
DIST	If the stylus is not deflected within this defined value, the control will issue an error message.	**Maximum measuring range? [mm]**
SET_UP	In **SET_UP** you define how far from the defined or calculated touch point the control is to pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles, you can also define a set-up clearance that is added to **SET_UP**	**Set-up clearance? [mm]**
F_PREPOS	Define the speed for pre-positioning: ■ Pre-positioning with speed from **FMAX**: **FMAX_PROBE** ■ Pre-positioning with machine rapid traverse: **FMAX_MACHINE**	**Pre-position at rapid? ENT/ NOENT**
TRACK	To increase measuring accuracy, you can use **TRACK = ON** to have an infrared touch probe oriented in the programmed probe direction before every probing process. In this way, the stylus is always deflected in the same direction: ■ **ON**: Perform spindle tracking ■ **OFF**: Do not perform spindle tracking	**Probe oriented? Yes=ENT/ No=NOENT**

Abbr.	Inputs	Dialog
SERIAL	You need not make an entry in this column. The TNC automatically enters the serial number of the touch probe if the touch probe has an EnDat interface.	**Serial number?**
REACTION	As soon as touch probes with a collision protection adapter detect a collision, they react by resetting the ready signal. The entry defines how the control is to react to the resetting of the ready signal ■ **NCSTOP**: The NC program will be interrupted ■ **EMERGSTOP**: Emergency stop, quick braking of the axes	**Reaction? EMERGS-TOP=ENT/NCSTOP=NOENT**

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.

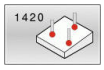
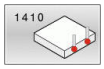
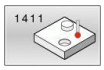
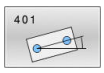
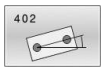
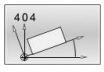
4

Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment

4.1 Overview

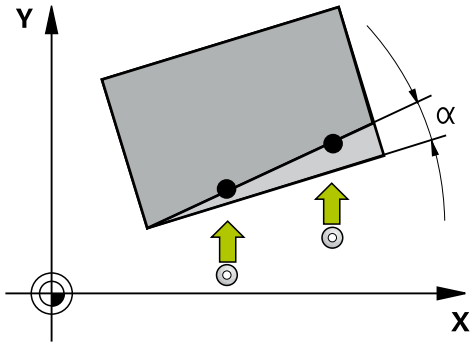
The control must be specifically prepared by the machine tool builder for the use of a 3-D touch probe.

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) ■ Automatic measurement using three points ■ Compensation via basic rotation or rotary table rotation	66
	PROBING ON EDGE (Cycle 1410, DIN/ISO: G1410) ■ Automatic measurement using two points ■ Compensation via basic rotation or rotary table rotation	71
	PROBING TWO CIRCLES (Cycle 1411, DIN/ISO: G1411) ■ Automatic measurement using two holes or studs ■ Compensation via basic rotation or rotary table rotation	77
	BASIC ROTATION (Cycle 400, DIN/ISO: G400) ■ Automatic measurement using two points ■ Compensation via basic rotation	84
	ROT. OF 2 HOLES (Cycle 401, DIN/ISO: G401) ■ Automatic measurement using two holes ■ Compensation via basic rotation	87
	ROT. OF 2 STUDS (Cycle 402, DIN/ISO: G402) ■ Automatic measurement using two studs ■ Compensation via basic rotation	91
	ROT. IN ROTARY AXIS (Cycle 403, DIN/ISO: G403) ■ Automatic measurement using two points ■ Compensation via rotary table rotation	96
	ROT. IN C AXIS (Cycle 405, DIN/ISO: G405 ■ Automatic alignment of an angular offset between a hole center and the positive Y axis ■ Compensation via rotary table rotation	101
	SET BASIC ROTATION (Cycle 404, DIN/ISO: G404) ■ Setting any basic rotation	105

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: **Input Coordinate System**
W-CS	W-CS: **Workpiece Coordinate System**
Object	Object to be probed: circle, stud, plane, edge

Evaluation – preset:

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

Operating notes:

- When probing, existing 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.

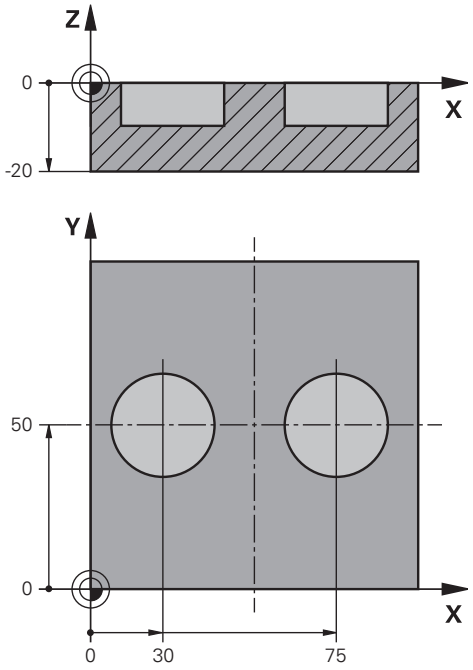
Programming and operating notes:

- 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

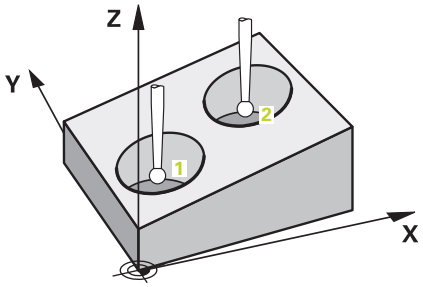
Important: Specify the **nominal positions** from the drawing!

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



Hole

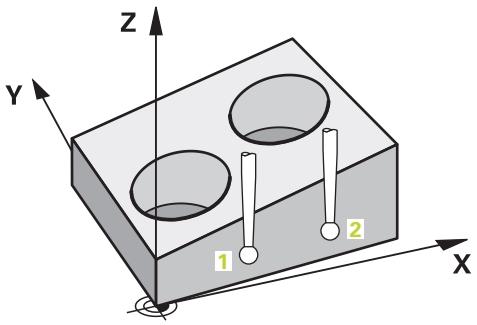
In this example, you will align two holes. Probing is done in the X axis (principal axis) and in the Y axis (secondary axis). This means that it is mandatory to define a nominal position for these axes! A nominal position for the Z axis (tool axis) is not necessary as you will not measure in this direction.



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

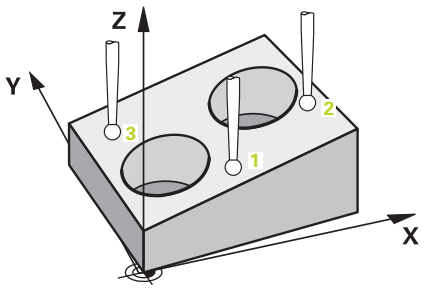
In this example, you will align an edge. Probing is on done in the Y axis (secondary axis). This means that it is mandatory to define a nominal position for this axis! Nominal positions for the X axis (principal axis) and for the Z axis (tool axis) are not required because you will not measure in these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



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

Plane

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



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

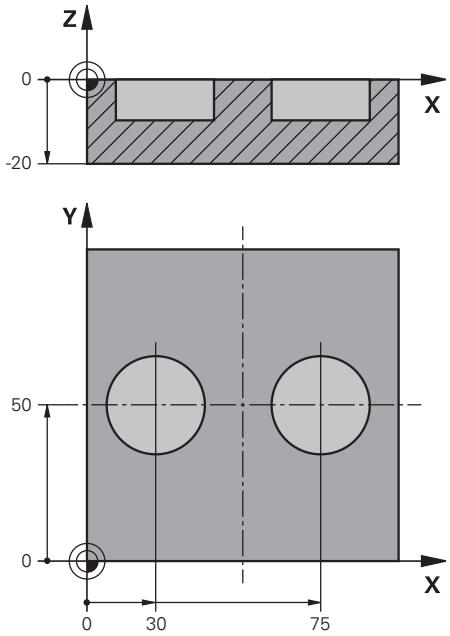
Evaluation of tolerances

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

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

Cycle run:

- With an error reaction parameter set to **Q309**=1, the control will check for scrap and rework. If you defined **Q309**=2, the control will only check for scrap.
- If the actual position determined by probing is erroneous, the control will interrupt execution of the NC program. A dialog window opens, showing all nominal and actual dimensions of the object
- You can then decide whether to continue machining or abort the NC program. To resume NC program operation, press **NC start**. To abort the program, press the **CANCEL** soft key**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
Q1100=+30 ;1ST POINT REF AXIS	Nominal position 1 of reference axis
Q1101= +50 ;1ST POINT MINOR AXIS	Nominal position 1 of minor axis
Q1102= -5 ;1ST POINT TOOL AXIS	Nominal position 1 of tool axis
QS1116="+10-1-0.5" ;DIAMETER 1	Diameter 1, tolerance specified
Q1103= +75 ;2ND POINT REF AXIS	Nominal position 2 of reference axis
Q1104=+50 ;2ND POINT MINOR AXIS	Nominal position 2 of minor axis
QS1105= -5 ;2ND POINT TOOL AXIS	Nominal position 2 of tool axis
QS1117="+10-1-0,5" ;DIAMETER 2	Diameter 2, tolerance specified
... ;	
Q309=2 ;ERROR REACTION	
... ;	

Transferring the actual position

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

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

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

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

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

Application

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

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 59
- 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 64
- 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 65

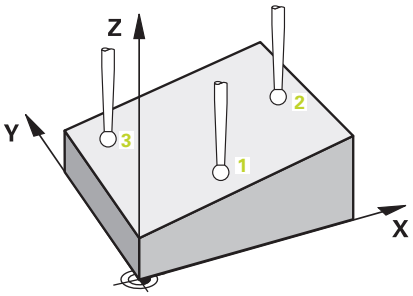
Cycle run

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

Please note while programming!

NOTICE

Danger of collision!

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

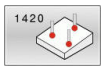
- Always move to clearance height between objects or touch points

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

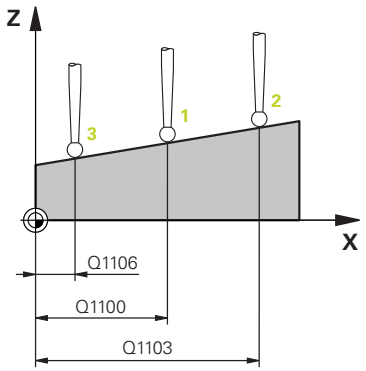
Aligning the rotary table axes:

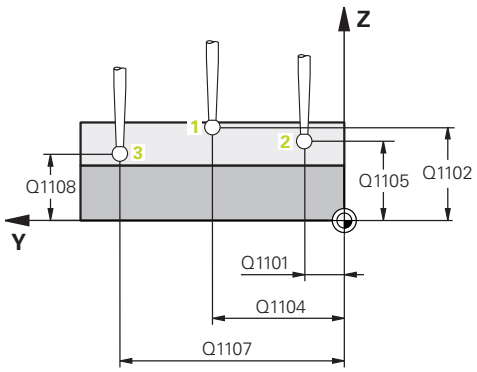
- Alignment with rotary table axes is only possible if two rotary table axes have been defined in the kinematics.
- To align the rotary table axes, (**Q1126** not equal to 0), the rotation must be saved (**Q1121** not equal to 0). Otherwise, an error message will be displayed because it is not possible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

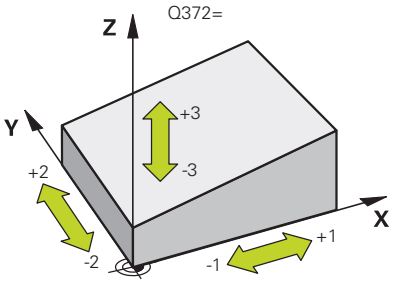
Cycle parameters

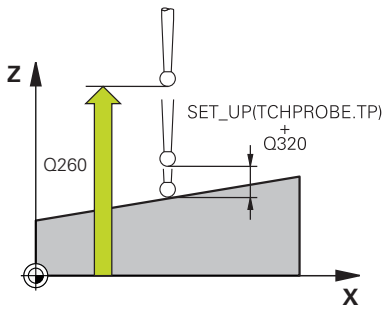


- **Q1100 1st noml. position of ref. axis** (absolute): Nominal position of the first touch point in the principal axis of the working plane.
  Input range: –99999.9999 to 99999.9999
- **Q1101 1st noml. position of minor axis** (absolute): Nominal position of the first touch point in the secondary axis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









- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
  Input range: –99999.9999 to 99999.9999
- **Q1125 Traverse to clearance height?**: 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)

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 59
- 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 64
- 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 65

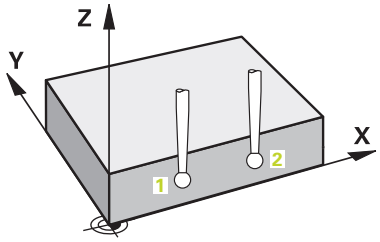
Cycle run

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

Please note while programming!

NOTICE

Danger of collision!

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

- Always move to clearance height between objects or touch points

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

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

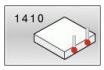
- If the current coordinates of the rotary axes and the defined tilt angles (3-D ROT menu) match, the working plane is consistent. Thus, the basic rotation is calculated in the input coordinate system (I-CS), based on the tool axis.
- If the current coordinates of the rotary axes and the defined tilt 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.

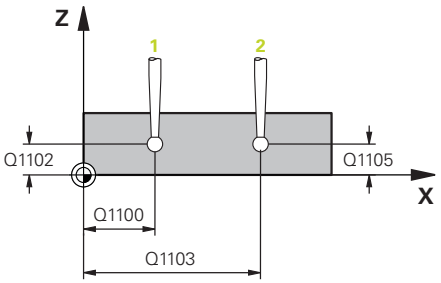
Aligning the rotary table axes:

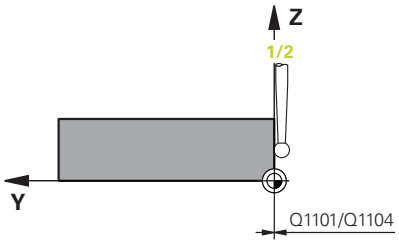
- Alignment with rotary table axes is only possible if the measured rotation can be compensated using a rotary table axis. This must be the first rotary table axis, seen from the workpiece.
- To align the rotary table axes, (**Q1126** not equal to 0), the rotation must be saved (**Q1121** not equal to 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

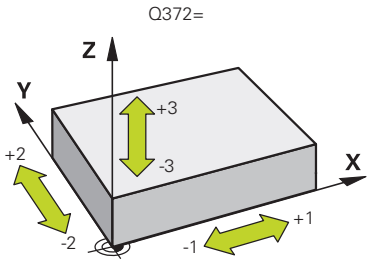
Cycle parameters

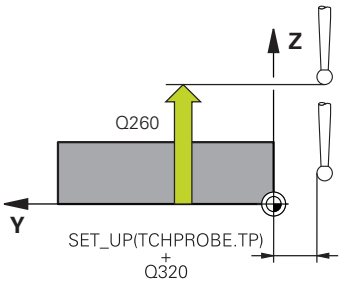


- **Q1100 1st noml. position of ref. axis** (absolute): Nominal position of the first touch point in the principal axis of the working plane.
  Input range: –99999.9999 to 99999.9999
- **Q1101 1st noml. position of minor axis** (absolute): Nominal position of the first touch point in the secondary axis 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)

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**:

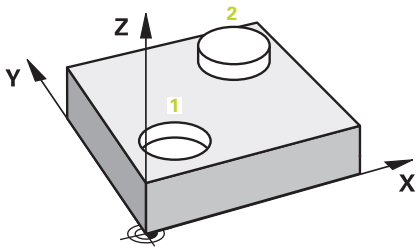
- If the probing position relative to the current datum is unknown, you can execute the cycle in semi-automatic mode

  Further information: "Semi-automatic mode", Page 59
- 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 64
- 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 65

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

Please note while programming!

NOTICE

Danger of collision!

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

- Always move to clearance height between objects or touch points

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

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

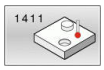
- If the current coordinates of the rotary axes and the defined tilt angles (3-D ROT menu) match, the working plane is consistent. Thus, the basic rotation is calculated in the input coordinate system (I-CS), based on the tool axis.
- If the current coordinates of the rotary axes and the defined tilt 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.

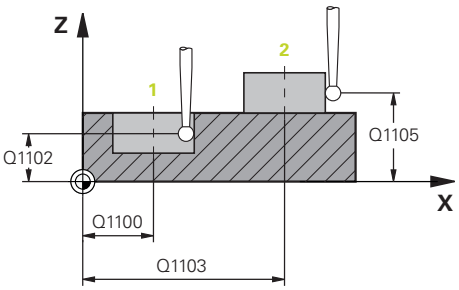
Aligning the rotary table axes:

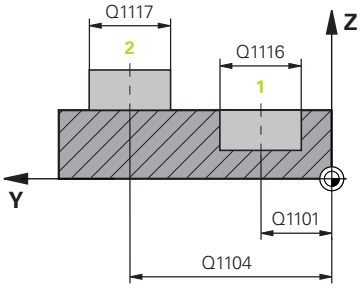
- Alignment with rotary table axes is only possible if the measured rotation can be compensated using a rotary table axis. This must be the first rotary table axis, seen from the workpiece.
- To align the rotary table axes, (**Q1126** not equal to 0), the rotation must be saved (**Q1121** not equal to 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

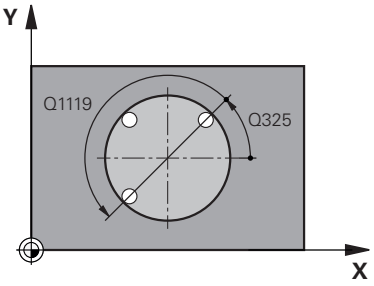
Cycle parameters

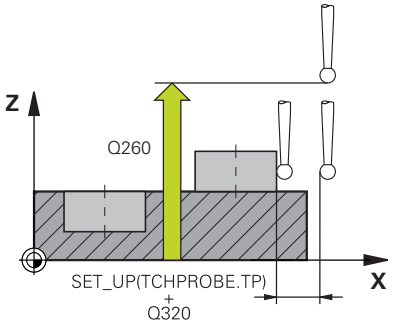


- **Q1100 1st noml. position of ref. axis** (absolute): Nominal position of the first touch point in the principal axis of the working plane.
  Input range: –99999.9999 to 99999.9999
- **Q1101 1st noml. position of minor axis** (absolute): Nominal position of the first touch point in the secondary axis 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

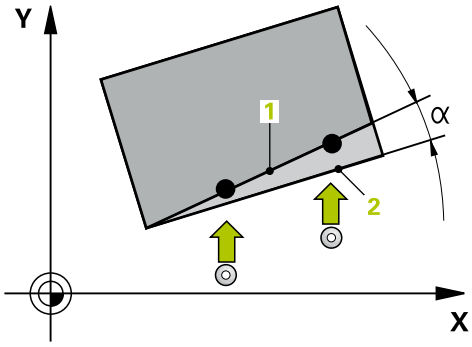
```
5 TCH PROBE 1410  PROBING TWO CIRCLES
   Q1100=+0   ;1ST POINT REF AXIS
   Q1101=+0   ;1ST POINT MINOR AXIS
   Q1102=+0   ;1ST POINT TOOL AXIS
   Q1116=0    ;DIAMETER 1
   Q1103=+0   ;2ND POINT REF AXIS
   Q1104=+0   ;2ND POINT MINOR AXIS
   Q1105=+0   ;2ND POINT TOOL AXIS
   Q1117=+0   ;DIAMETER 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 # (see figure at right). This enables you to measure the basic rotation against any straight line **1** of the workpiece and to establish the reference to the actual 0° direction **2**.



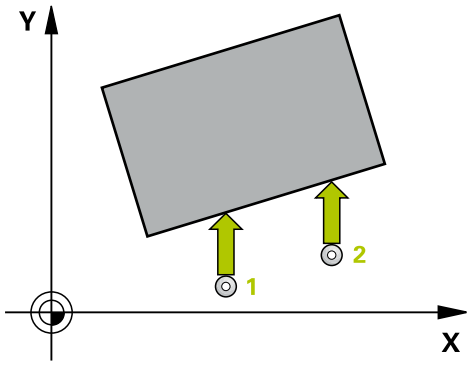
> These cycles do not work with 3-D Rot! In such a case, use Cycles **14xx**. **Further information:** "Touch probe cycles 14xx: fundamentals", Page 57

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

Application

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

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 47), 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!

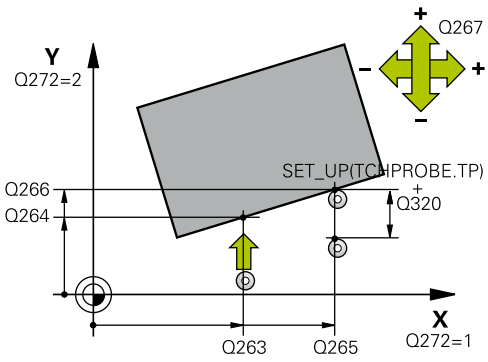
When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

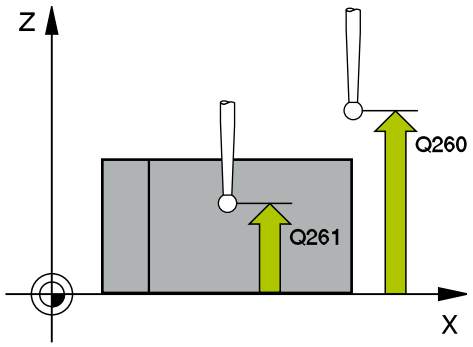
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- 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)

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

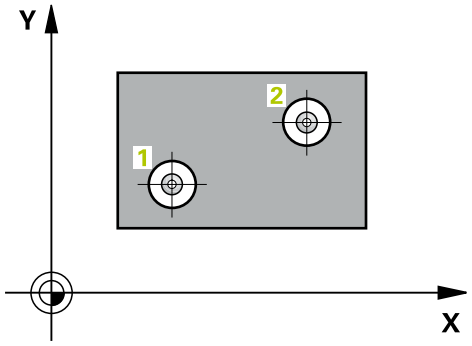
1 The control positions the touch probe at rapid traverse speed (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 47), to the programmed center point of the first hole **1**.

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

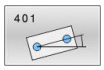
Danger of collision!

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

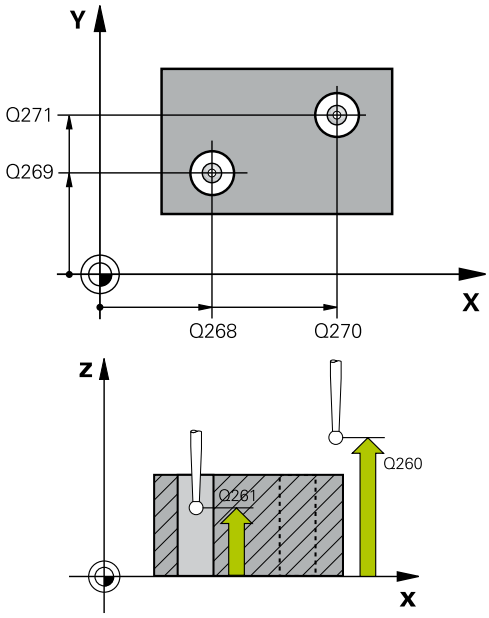
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- 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)

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

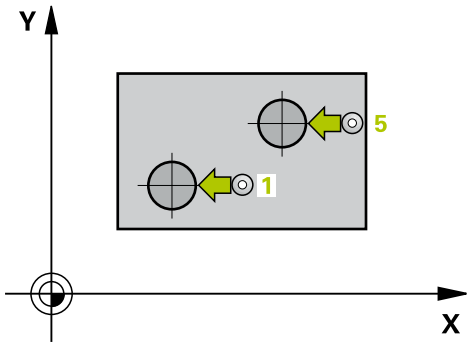
1 The control positions the touch probe at rapid traverse speed (value from FMAX column), using positioning logic (see "Executing touch probe cycles", Page 47), to the programmed center point of the first stud **1**.

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

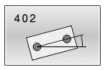
Danger of collision!

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

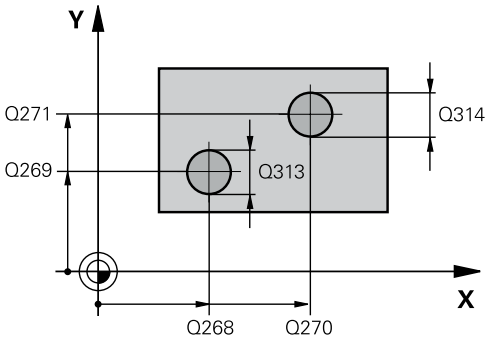
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

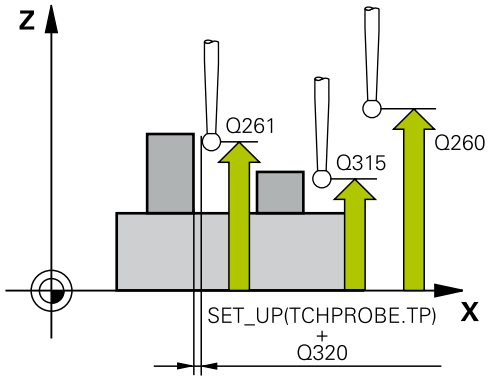
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The control will reset an active basic rotation at the beginning of the cycle.
- If you want to compensate the misalignment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

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.

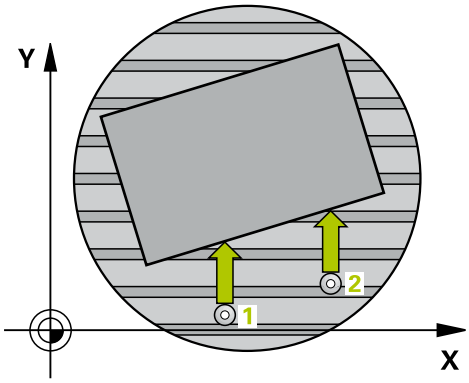
Cycle run

1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 47), 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° to +90°.

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

NOTICE

Danger of collision!

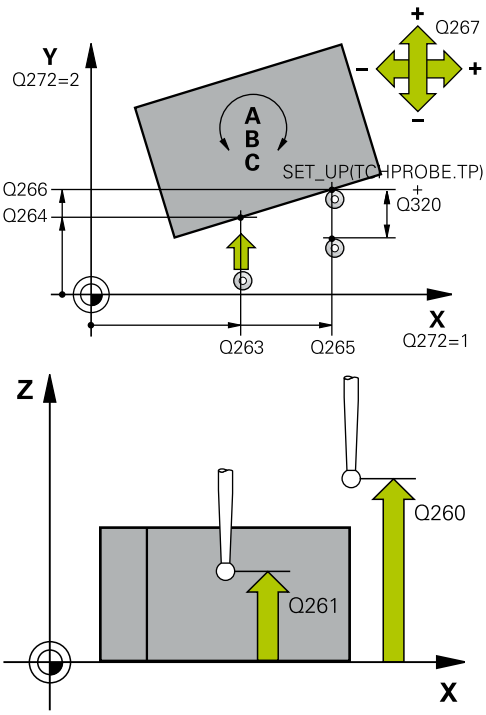
When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- 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.

Cycle parameters

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



Example

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

- **Q312 Axis for compensating movement?**:
  Specify the rotary axis in which the control is to compensate the measured misalignment:
  0: Automatic mode—the control uses the active kinematics to determine the rotary axis to be aligned. In Automatic mode the first rotary axis of the table (as viewed from the workpiece) is used as compensation axis. This is the recommended setting!
  4: Compensate misalignment with rotary axis A
  5: Compensate misalignment with rotary axis B
  6: Compensate misalignment with rotary axis C
- **Q337 Set to zero after alignment?**: Define whether the control should set the angle of the aligned rotary axis to 0 in the preset table or in the datum table after the alignment.
  0: Do not set the angle of the rotary axis to 0 in the table after alignment
  1: Set the angle of the rotary axis to 0 in the table after alignment
- **Q305 Number in table?** Specify the number of the row in the preset table in which the control is to enter the basic rotation.
  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

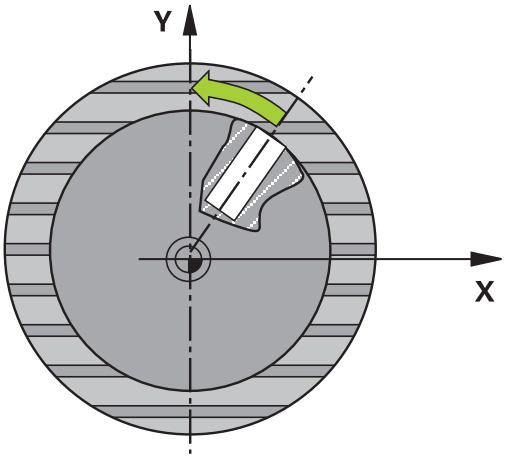
4.11 ROT. IN C AXIS (Cycle 405, DIN/ISO: G405

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

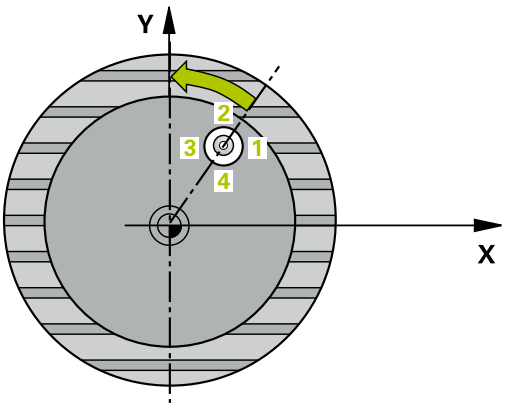
1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic "Executing touch probe cycles", at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of the touch probe 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!

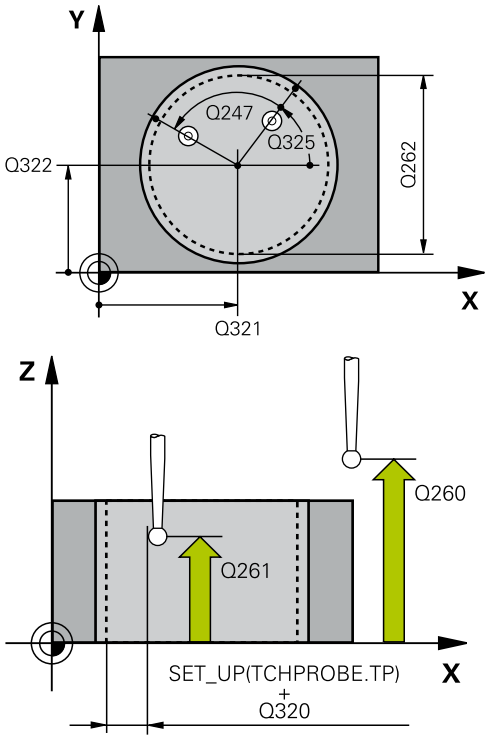
When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- 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)

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.

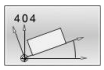
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

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

Example

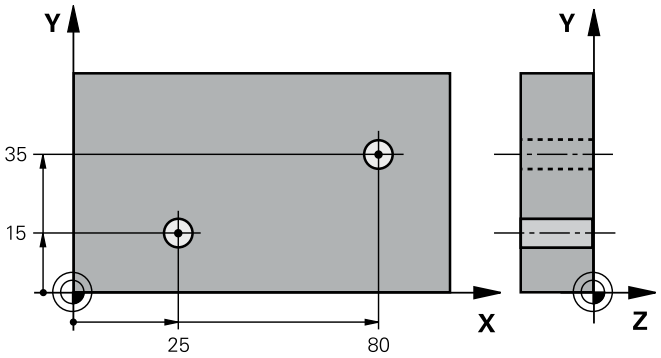
```
5 TCH PROBE 404  SET BASIC
    ROTATION
    Q307=+0     ;PRESET ROTATION ANG.
    Q305=-1     ;NUMBER IN TABLE
```

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

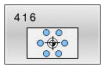
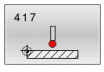
The control offers twelve cycles for automatically finding presets and handling them as follows:

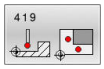
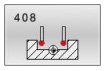
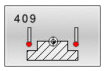
- Setting the determined values directly as display values
- Writing the determined values to the preset table
- Writing the determined values to a datum table

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) ■ Measuring the inside length and width of a rectangle ■ and defining the center as preset	111
	PRESET OUTS. RECTAN (Cycle 411, DIN/ISO: G411) ■ Measuring the outside length and width of a rectangle ■ and defining the center as preset	115
	PRESET INSIDE CIRCLE (Cycle 412, DIN/ISO: G412) ■ Measuring any four points on the circle inside ■ and defining the center as preset	119
	PRESET OUTS. CIRCLE (Cycle 413, DIN/ISO: G413) ■ Measuring any four points on circle outside ■ and defining the center as preset	124
	PRESET OUTSIDE CORNER (Cycle 414, DIN/ISO: G414) ■ Measuring two straight lines on the outside ■ and defining the intersection of the lines as preset	129
	PRESET INSIDE CORNER (Cycle 415, DIN/ISO: G415) ■ Measuring two straight lines on the inside ■ and defining the intersection of the lines as preset	134
	PRESET CIRCLE CENTER (Cycle 416, DIN/ISO: G416) ■ Measuring any three holes on a circular hole pattern ■ and defining the circle center as preset	139
	PRESET IN TS AXIS (Cycle 417, DIN/ISO: G417) ■ Measuring any position in the touch-probe axis ■ and defining any position as preset	144

Soft key	Cycle	Page
	PRESET FROM 4 HOLES (Cycle 418, DIN/ISO: G418) ■ Measuring 2 holes on each line crosswise ■ and defining the intersection of the lines as preset	146
	PRESET IN ONE AXIS (Cycle 419, DIN/ISO: G419) ■ Measuring any position in a selectable axis ■ and defining any position in a selectable axis as preset	151
	SLOT CENTER PRESET (Cycle 408, DIN/ISO: G408) ■ Measuring the width of an inside slot ■ and defining slot center as datum	154
	RIDGE CENTER PRESET (Cycle 409, DIN/ISO: G409) ■ Measuring the width of an outside ridge ■ and defining the ridge center as preset	158

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

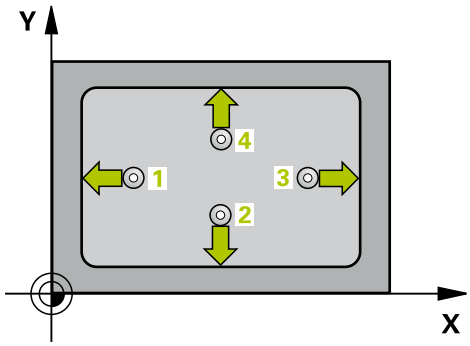
The control saves the measurement results of the respective probing cycle in the globally effective Q parameters **Q150** to **Q160**. You can use these parameters in your NC program. Note the table of result parameters listed with every cycle description.

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

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 110)
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!

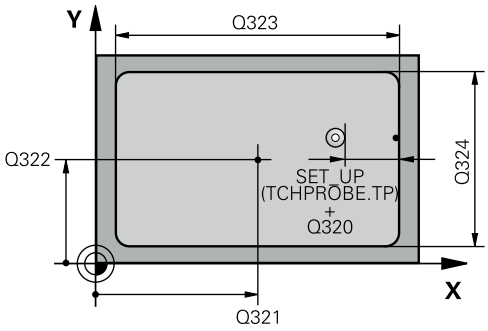
To prevent a collision between touch probe and workpiece, enter **low** estimates for the lengths of the first and second sides. If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center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

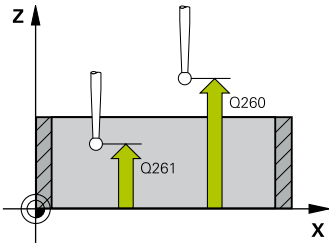
- Before a cycle definition you must have programmed a tool call to define the touch probe axis.

- 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
```

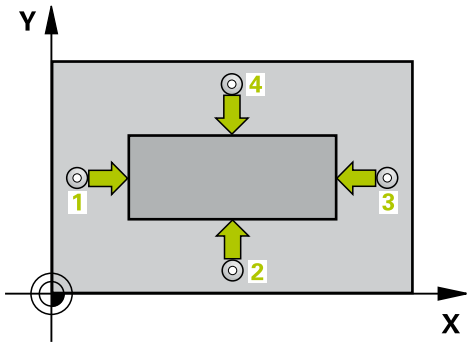
- **Q331 New preset in reference axis?** (absolute): Coordinate in the principal axis at which the control should set the pocket center. Default setting = 0.
  Input range: –99999.9999 to 99999.9999
- **Q332 New preset in minor axis?** (absolute): Coordinate in the secondary axis at which the control 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 110)
  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)

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 47), 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 110)

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!

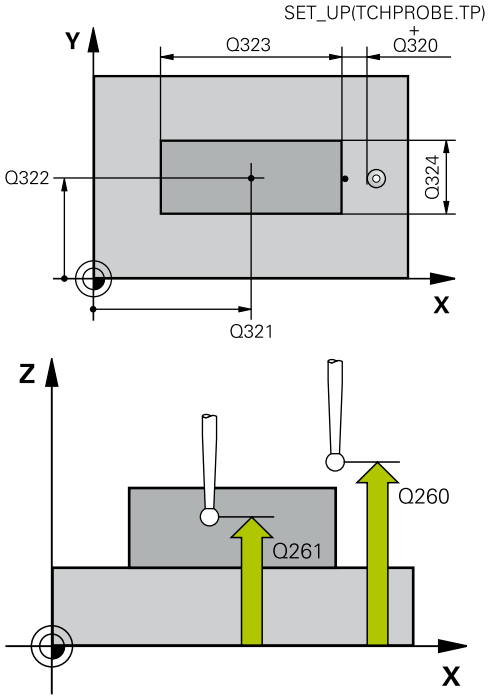
To prevent a collision between touch probe and workpiece, enter **high** estimates for the lengths of the 1st and 2nd sides.

- 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

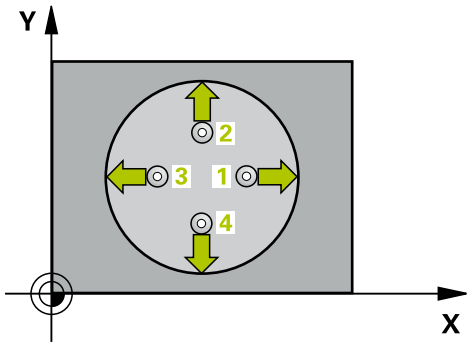
- **Q331 New preset in reference axis?** (absolute): Coordinate in the principal axis at which the control should set the stud center. Default setting = 0.
  Input range: –99999.9999 to 99999.9999
- **Q332 New preset in minor axis?** (absolute): Coordinate in the secondary axis at which the control should 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 110)
  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)

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 47), 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 110), 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.

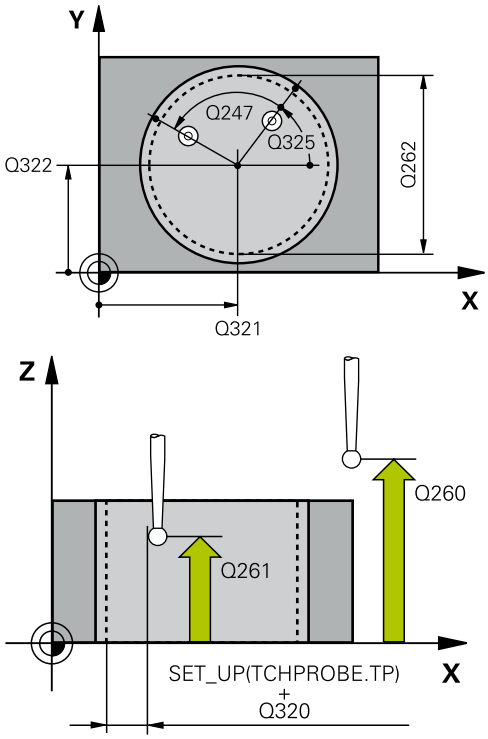
- Positioning of the touch points
- Before a cycle definition you must have programmed a tool call to define the touch probe axis.

- 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 110)
  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).

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

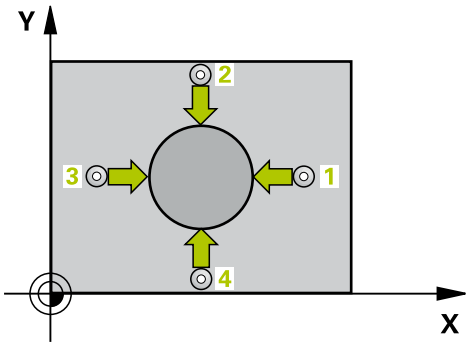
- **Q381 Probe in TS axis? (0/1)**: Specify whether the control should also set the preset in the touch probe axis:
  0: Do not set the preset in the touch probe axis
  1: Set the preset in the touch probe axis
- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the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

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

Application

Touch probe cycle **413** finds the center of a circular stud and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.

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 47), 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 110), 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.

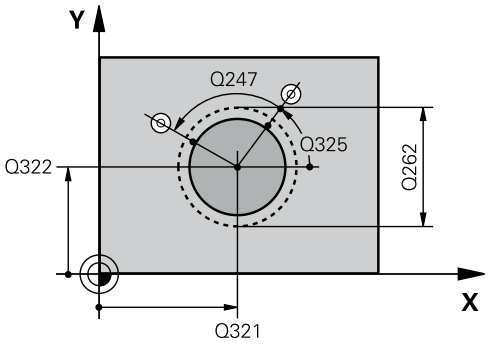
- Before a cycle definition you must have programmed a tool call to define the touch probe axis.

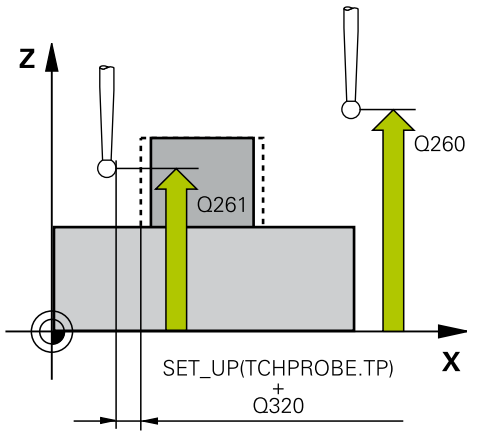
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The smaller the stepping angle **Q247**, the less accurately the control can calculate the preset. Minimum input value: 5°

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 110)
  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

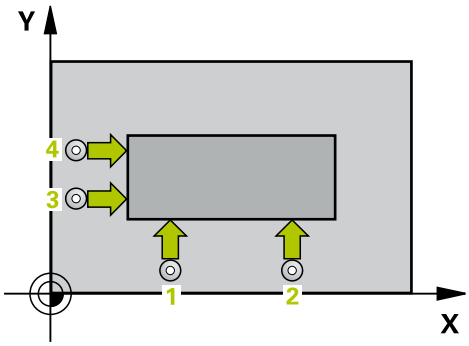
- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.
  Input range: –99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (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)

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 47), 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 110), 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.

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

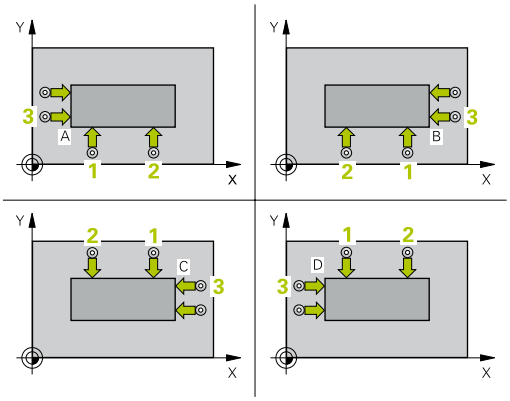
Please note while programming:

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.

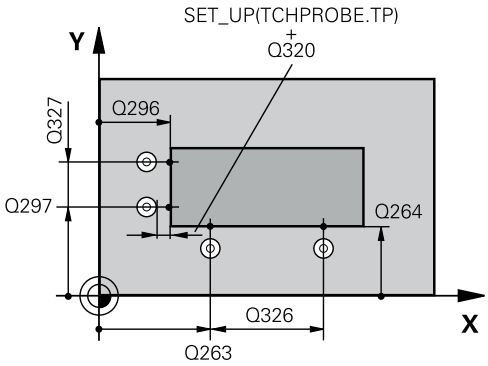


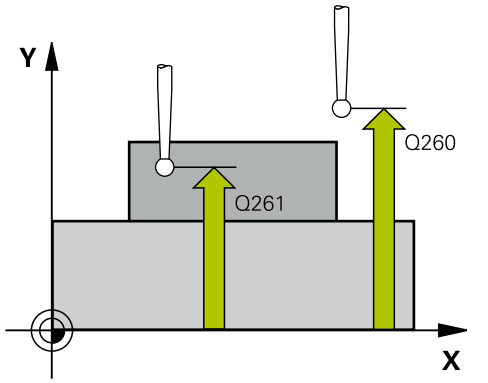
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- By defining the positions of the measuring points **1** and **3**, you also determine the corner at which the control sets the preset (see figure at the right and table below).

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

Cycle parameters

- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane.
  Input range: –99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working plane.
  Input range -99999.9999 to 99999.9999
- **Q326 Spacing in 1st axis?** (incremental): Distance between the first and second measuring points in the principal axis of the working plane.
  Input range: 0 to 99999.9999
- **Q296 3rd measuring point in 1st axis?** (absolute): Coordinate of the third touch point in the principal axis of the working plane.
  Input range: –99999.9999 to 99999.9999
- **Q297 3rd measuring point in 2nd axis?** (absolute): Coordinate of the third touch point in the secondary axis of the working plane.
  Input range: –99999.9999 to 99999.9999
- **Q327 Spacing in 2nd axis?** (incremental): Distance between third and fourth measuring points in the secondary axis of the working plane.
  Input range: 0 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
  Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table.
  Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
  Input range: –99999.9999 to 99999.9999
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move 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 110)
  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

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

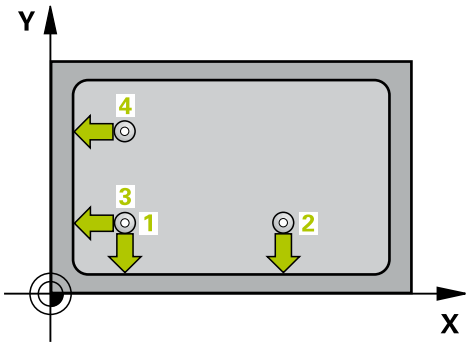
- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.
  Input range: –99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?**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

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

Application

Touch probe cycle **415** finds the intersection of two lines and defines it as the preset. If desired, the control can also write the point of intersection coordinates to a datum table or the preset table.

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 110), 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!

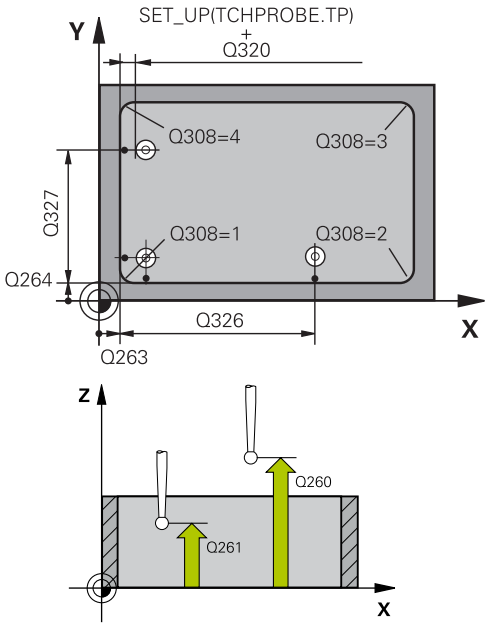
When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- 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 110)
  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

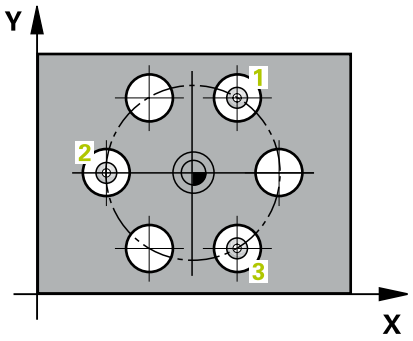
- **Q382 Probe TS axis: Coord. 1st axis?** (absolute): Coordinate of the probe point in the principal axis of the working plane at which point the preset is to be set in the touch probe axis. Only effective if **Q381** = 1.
  Input range: –99999.9999 to 99999.9999
- **Q383 Probe TS axis: Coord. 2nd axis?** (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)

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 47), 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 110), and saves the actual values in the Q parameters listed below.
8 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

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

Please note while programming:

NOTICE

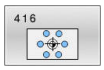
Danger of collision!

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

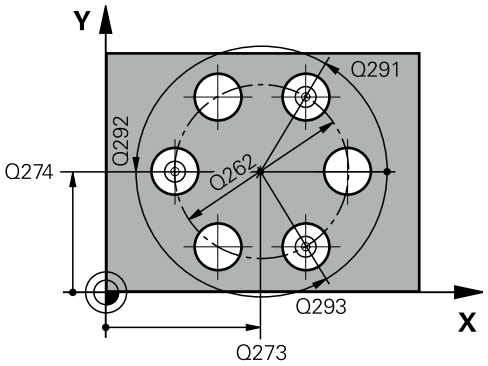
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

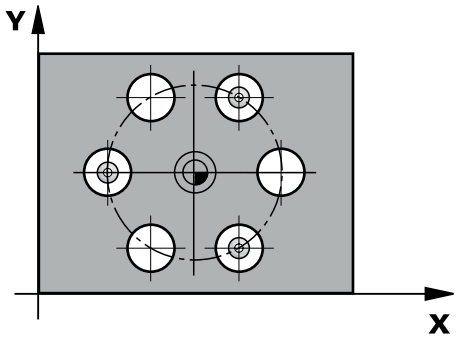
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

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 110)
  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)

Application

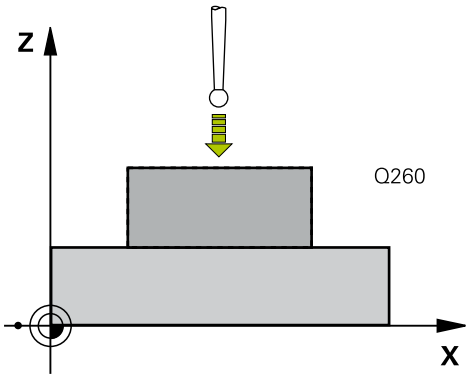
Touch probe cycle **417** measures any coordinate in the touch probe axis and defines it as the preset. If desired, the control can also write the measured coordinates to a datum table or preset table.

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 47), 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 110), and saves the actual values in the Q parameters listed below.



Parameter number	Meaning
Q160	Actual value of measured point

Please note while programming:

NOTICE

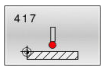
Danger of collision!

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

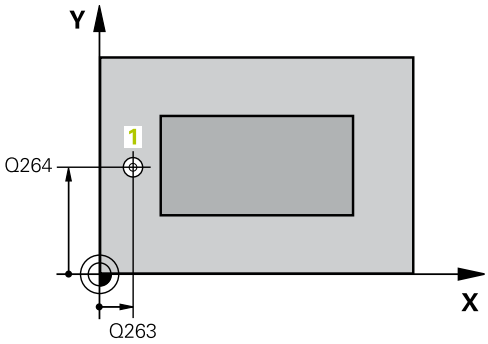
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

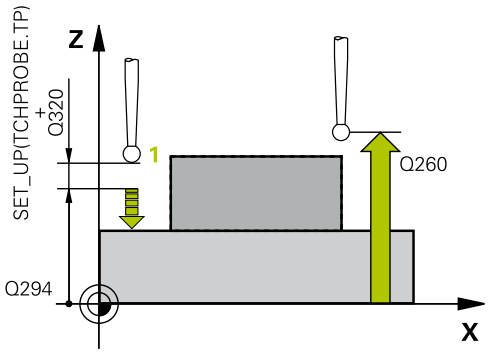
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The control then sets the preset in this axis.

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 110)
  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

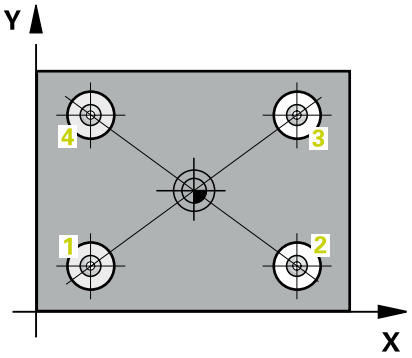
```
5 TCH PROBE 417  PRESET IN TS AXIS
    Q263=+25  ;1ST POINT 1ST AXIS
    Q264=+25  ;1ST POINT 2ND AXIS
    Q294=+25  ;1ST POINT 3RD AXIS
    Q320=0    ;SET-UP CLEARANCE
    Q260=+50  ;CLEARANCE HEIGHT
    Q305=0    ;NUMBER IN TABLE
    Q333=+0   ;PRESET
    Q303=+1   ;MEAS. VALUE TRANSFER
```

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

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 47), 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 110). The control calculates the preset as the intersection of the lines connecting the centers of holes **1**/**3** and **2**/**4** and saves the actual values in the Q parameters listed below.
7 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

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

Please note while programming:

NOTICE

Danger of collision!

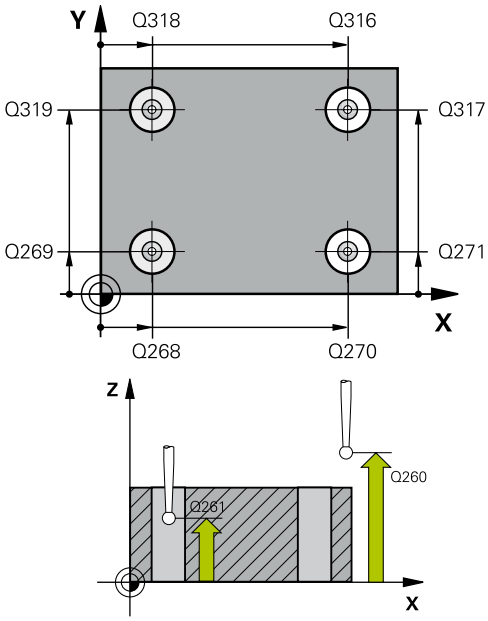
When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

- 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 110)
  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)

Application

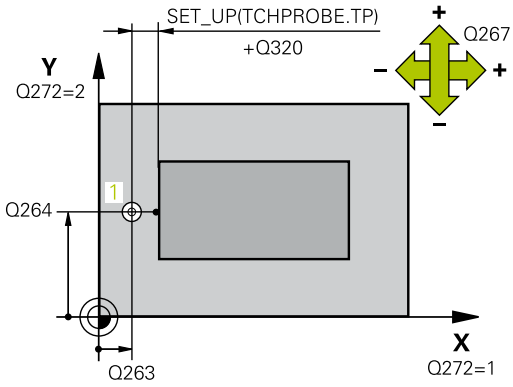
Touch probe cycle **419** measures any coordinate in the a selectable axis and defines it as the preset. If desired, the control can also write the measured coordinates to a datum table or preset table.

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 47), 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 110)



Please note while programming:

NOTICE

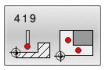
Danger of collision!

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.

Cycle parameters

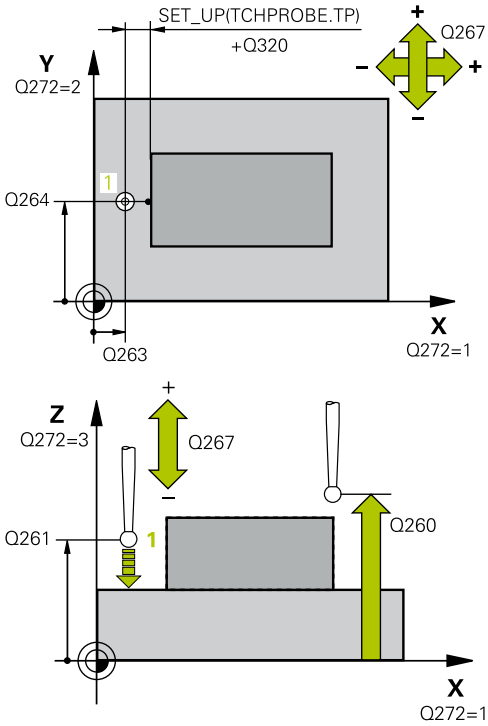


- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane.
  Input range: –99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working plane.
  Input range -99999.9999 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
  Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table.
  Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
  Input range: –99999.9999 to 99999.9999
- **Q272 Meas. axis (1/2/3, 1=ref. axis)?**: Axis in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
  3: Touch probe axis = measuring axis

Axis assignment

Active touch probe axis: Q272 = 3	Associated reference axis: Q272 = 1	Associated minor axis: Q272 = 2
Z	X	Y
Y	Z	X
X	Y	Z

- **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
  –1: Negative traverse direction
  +1: Positive traverse direction



Example

```
5 TCH PROBE 419  PRESET IN ONE AXIS
  Q263=+25  ;1ST POINT 1ST AXIS
  Q264=+25  ;1ST POINT 2ND AXIS
  Q261=+25  ;MEASURING HEIGHT
  Q320=0    ;SET-UP CLEARANCE
  Q260=+50  ;CLEARANCE HEIGHT
  Q272=+1   ;MEASURING AXIS
  Q267=+1   ;TRAVERSE DIRECTION
  Q305=0    ;NUMBER IN TABLE
  Q333=+0   ;PRESET
  Q303=+1   ;MEAS. VALUE TRANSFER
```

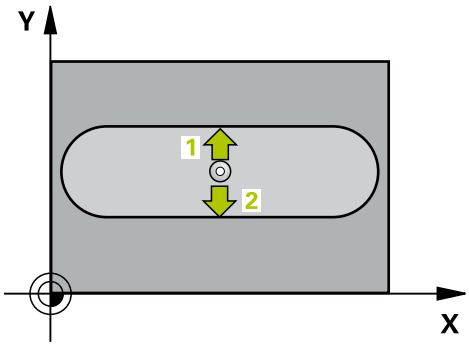
- **Q305 Number in table?**: Indicate the number of the row in the preset table / datum table in which the control saves the coordinates.
  If **Q303 = 1**, the control will write the data to the preset table. If the active preset changes, this change will immediately become effective. Otherwise, the 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 110)
  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)

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 47), 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 110), and saves the actual values in the Q parameters listed below.
5 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q166	Actual value of measured slot width
Q157	Actual value of the centerline

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles **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!

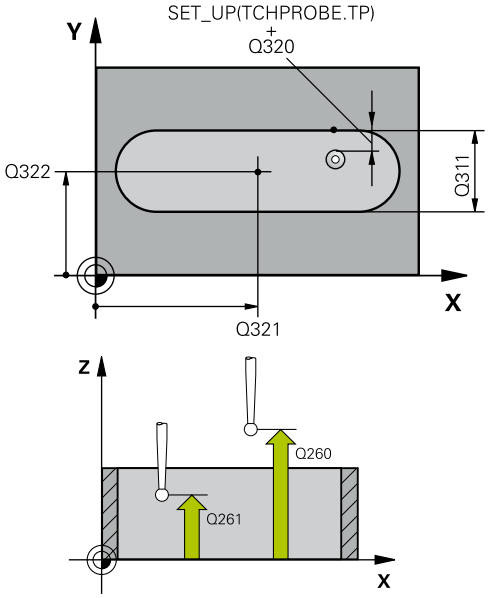
To prevent a collision between touch probe and workpiece, enter a **low** estimate for the slot width. If the slot width and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the slot. In this case, the touch 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
```

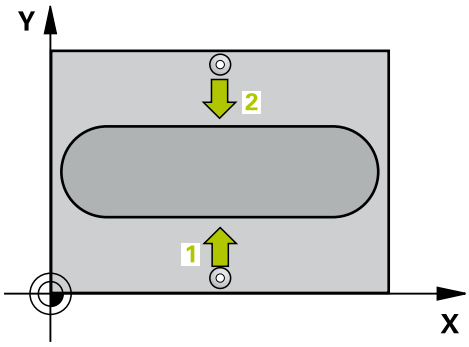
- **Q405 New preset?** (absolute): Coordinate in the measuring axis at which the control should set the calculated slot center. Default setting = 0.
  Input range: –99999.9999 to 99999.9999
- **Q303 Meas. value transfer (0,1)?**: Specify whether the determined preset is to be saved in the preset table 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)

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 47), 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 110), and saves the actual values in the Q parameters listed below.

5 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

Parameter number	Meaning
Q166	Actual value of measured ridge width
Q157	Actual value of the centerline

Please note while programming:

NOTICE

Danger of collision!

When running touch probe cycles **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

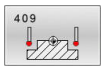
Danger of collision!

To prevent a collision between touch probe and workpiece, enter a **high** estimate for the ridge width.

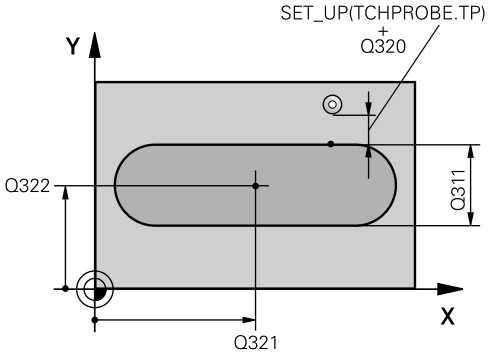
- Before the cycle definition, you must have programmed a tool call to define the touch probe axis.

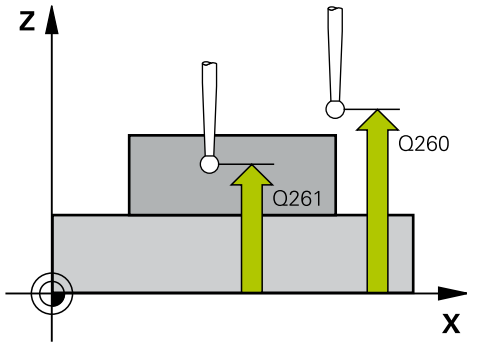
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.

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



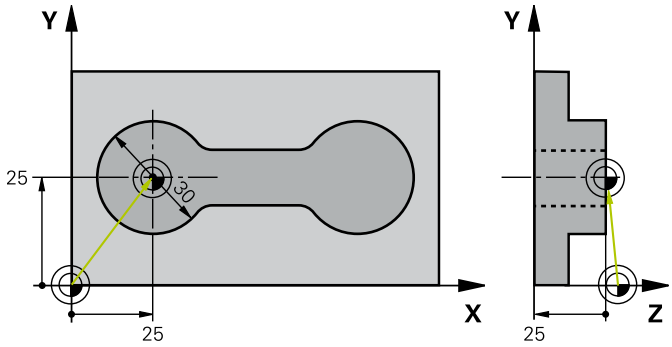


Example

```
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

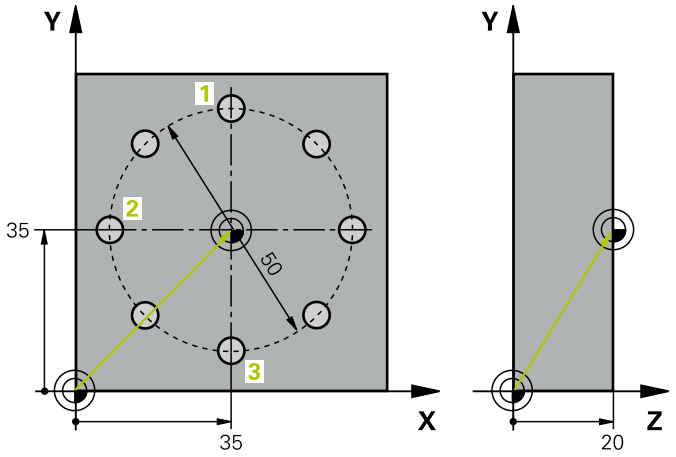
5.14 Example: Presetting in center of a circular segment and on top surface of workpiece



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.



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

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

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.

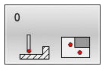
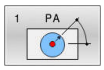
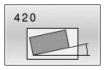
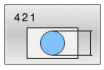
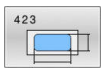
NOTICE

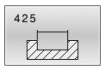
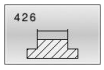
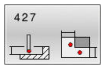
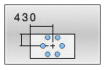
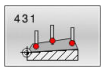
Danger of collision!

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) ■ Measuring a coordinate in a selectable axis	172
	POLAR PRESET (Cycle 1) ■ Measuring a point ■ Probing direction via angle	173
	MEASURE ANGLE (Cycle 420, DIN/ISO: G420) ■ Measuring an angle in the working plane	174
	MEASURE HOLE (Circle 421, DIN/ISO: G421) ■ Measuring the position of a hole ■ Measuring the diameter of a hole ■ Nominal-to-actual value comparison, if applicable	177
	MEASURE CIRCLE OUTSIDE (Cycle 422, DIN/ISO: G422) ■ Measuring the position of a circular stud ■ Measuring the diameter of a circular stud ■ Nominal-to-actual value comparison, if applicable	182
	MEASURE RECTANGLE INSIDE (Cycle 423, DIN/ISO: G423) ■ Measuring the position of a rectangular pocket ■ Measuring the length and width of a rectangular pocket ■ Nominal-to-actual value comparison, if applicable	187
	MEASURE RECTANGLE OUTSIDE (Cycle 424, DIN/ISO: G424) ■ Measuring the position of a rectangular stud ■ Measuring the length and width of a rectangular stud ■ Nominal-to-actual value comparison, if applicable	191

Soft key	Cycle	Page
	MEASURE INSIDE WIDTH (Cycle 425, DIN/ISO: G425) ■ Measuring the position of a slot ■ Measuring the width of a slot ■ Nominal-to-actual value comparison, if applicable	194
	MEASURE RIDGE WIDTH (Cycle 426, DIN/ISO: G426) ■ Measuring the position of a ridge ■ Measuring the width of a ridge ■ Nominal-to-actual value comparison, if applicable	197
	MEASURE COORDINATE (Cycle 427, DIN/ISO: G427) ■ Measuring any coordinate in a selectable axis ■ Nominal-to-actual value comparison, if applicable	200
	MEAS. BOLT HOLE (Cycle 430, DIN/ISO: G430) ■ Measuring the center point of a bolt hole circle ■ Measuring the diameter of a bolt hole circle ■ Nominal-to-actual value comparison, if applicable	203
	MEASURE PLANE (Cycle 431, DIN/ISO: G431) ■ Finding the angle of a plane by measuring three points	206

Recording the results of measurement

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

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

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

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

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

Measuring log for Probing Cycle 421 Hole Measuring

Date: 30-06-2005
Time: 6:55:04
Measuring program: TNC:\GEH35712\CHECK1.H

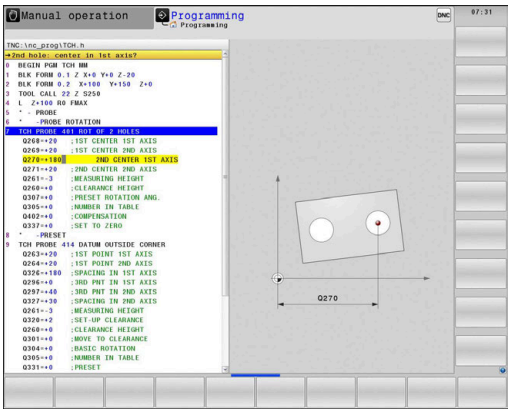
Nominal values:	
Center in reference axis:	50.0000
Center in minor axis:	65.0000
Diameter:	12.0000
Given limit values:	
Maximum limit for center in reference axis:	50.1000
Minimum limit for center in reference axis:	49.9000
Maximum limit for center in minor axis:	65.1000
Minimum limit for center in minor axis:	64.9000
Maximum dimension for hole:	12.0450
Minimum dimension for hole:	12.0000
Actual values:	
Center in reference axis:	50.0810
Center in minor axis:	64.9530
Diameter:	12.0259
Deviations:	
Center in reference axis:	0.0810
Center in minor axis:	-0.0470
Diameter:	0.0259
Further measuring results: Measuring height:	-5.0000

End of measuring log

Measurement results in Q parameters

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

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



Classification of results

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

Class of results	Parameter value
Measurement results are within tolerance	**Q180** = 1
Rework is required	**Q181** = 1
Scrap	**Q182** = 1

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

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

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

Tolerance monitoring

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

Tool monitoring

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

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

Tool compensation

Requirements:

- Active tool table
- Tool monitoring must be switched on in the cycle: Set **Q330** unequal to 0 or enter a tool name. To enter the tool name, press the associated soft key. The control no longer displays the right single quotation mark.

- HEIDENHAIN recommends using this function only if the tool to be compensated for is the one that was used to machine the contour as well as if any necessary reworking will also be done with this tool.
- If you perform several compensation measurements, the control adds the respective measured deviation to the value stored in the tool table.

Milling tool: If you refer to a milling cutter in parameter **Q330**, the appropriate values are compensated in the following way: The control basically always compensates the tool radius in column DR of the tool table, even if the measured deviation is within the specified tolerance. You can inquire whether re-working is necessary via parameter **Q181** in the NC program (**Q181**=1: rework required).

Turning tool: (applies only to Cycles **421**, **422**, **427**) If you reference a turning tool in parameter **Q330**, the appropriate values in row DZL and DXL, respectively, will be compensated. The control also monitors the breakage tolerance, which is defined in column LBREAK. You can inquire whether re-working is necessary via parameter **Q181** in the NC program (**Q181**=1: rework required).

If you want to compensate the values for an indexed tool with a tool name, program the following:

- **QS0** = "TOOL NAME"
- **FN18: SYSREAD Q0 = ID990 NR10 IDX0**; specify the number of the **QS** parameter in **IDX**
- **Q0**= **Q0** +0.2; add the index of the basic tool number
- In the cycle: **Q330** = **Q0**; use the indexed tool

Tool breakage monitoring

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)

Application

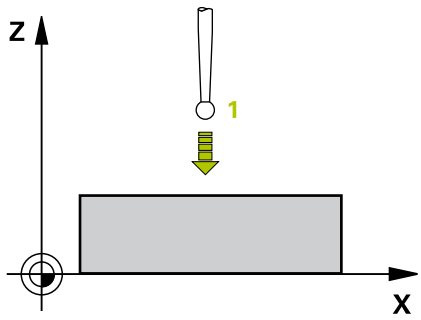
The touch probe cycle measures any position on the workpiece in a selectable axis direction.

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

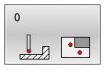
Danger of collision!

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)

Application

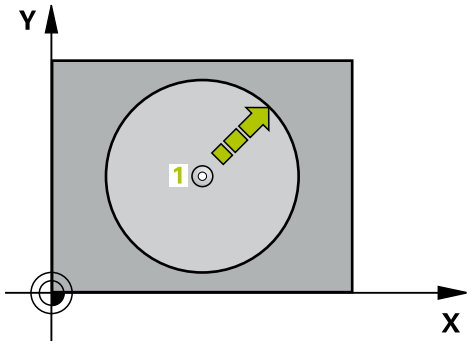
Touch probe cycle **1** measures any position on the workpiece in any probing direction.

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

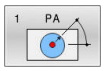
Danger of collision!

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

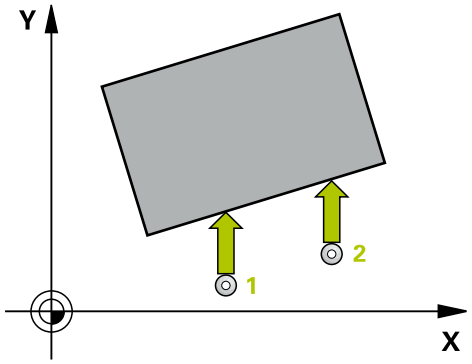
```
67 TCH PROBE 1.0  POLAR PRESET
68 TCH PROBE 1.1  X ANGLE: +30
69 TCH PROBE 1.2  X+5 Y+0 Z-5
```

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

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 47), at the programmed touch point **1**. The sum of **Q320**, **SET_UP** and the ball-tip radius is taken into account for probe movements in any probing direction. When the probe movement starts, the center of the ball tip will be offset by this sum in the direction opposite the probing direction.

2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).

3 The touch probe then moves to the next touch point **2** and probes again.

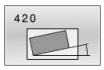
4 The control returns the touch probe to the clearance height and saves the measured angle in the following Q parameter:

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

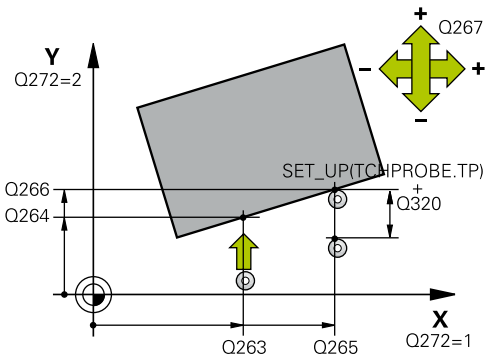
Please note while programming:

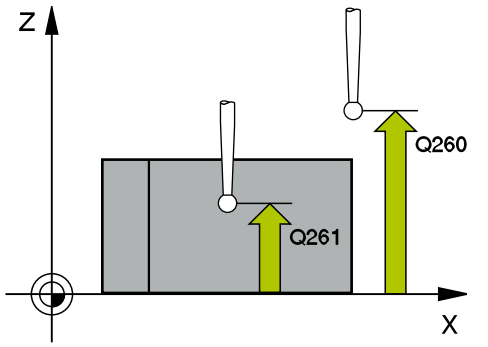
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- If touch probe axis = measuring axis, you can measure the angle in the direction of the A axis or B axis:
  - If you want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

Cycle parameters



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





Example

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

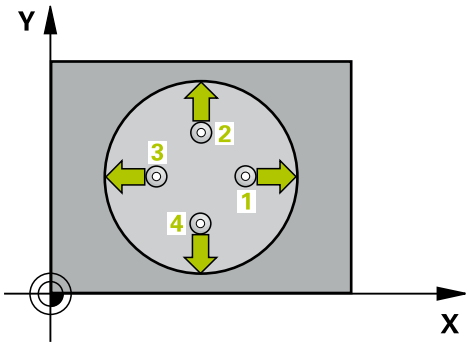
- **Q301 Move to clearance height (0/1)?**: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Q281 Measuring log (0/1/2)?**: Define whether the control should create 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)

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 47), 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:

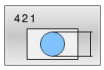
Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis
Q163	Deviation from diameter

Please note while programming:

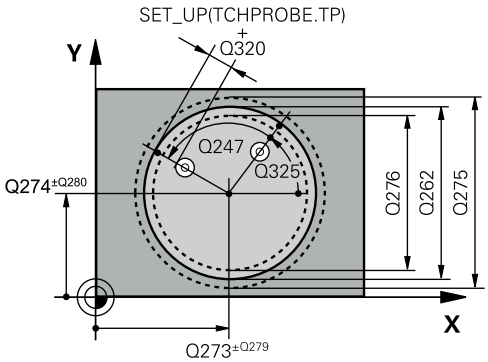
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°.
- If parameter **Q330** references a milling tool, the information in parameters **Q498** and **Q531** has no effect

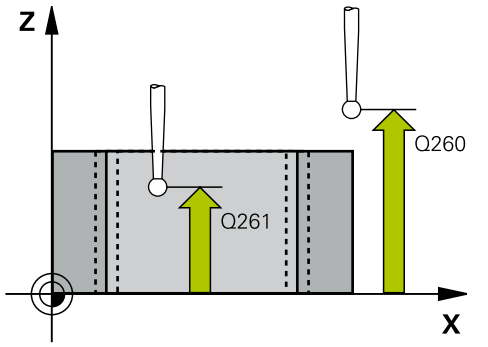
  If parameter **Q330** references a turning tool, the following applies:
  - Parameters **Q498** and **Q531** have to be defined
  - The information in parameters **Q498**, **Q531**, for example from Cycle **800**, has to match this information
  - If the control compensates the position of the turning tool, the corresponding values in rows **DZL** and **DXL**, respectively, will be compensated.
  - The control also monitors the breakage tolerance, which is defined in column **LBREAK**.

Cycle parameters



- **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
```

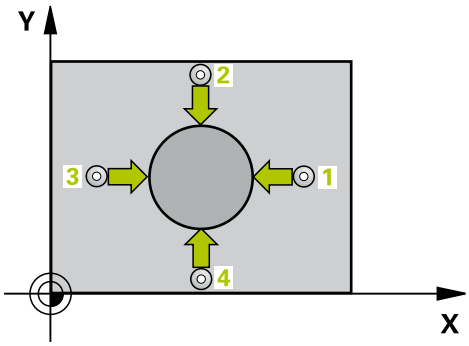
- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 170); alternatively tool name with a maximum of 16 characters
  0: Monitoring inactive
  >0: Number or name of the tool that the control used for machining. It is possible to take over the 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
- **Q498 Reverse tool (0=no/1=yes)?**: Only relevant if you specified a turning tool in parameter **Q330**. For proper monitoring of the turning tool, the control requires the exact working condition. Therefore, enter the following:
  1: Turning tool is mirrored (rotated by 180°), e.g. by Cycle **800** and parameter **Reverse the tool Q498**=1
  0: Turning tool corresponds to the description in the turning tool table (toolturn.trn), no modification, e.g. by Cycle **800** and parameter **Reverse the tool Q498**=0
- **Q531 Angle of incidence?**: Only relevant if you specified a turning tool in parameter **Q330**. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining, e.g. from Cycle **800**, **Angle of incidence? Q531**.
  Input range: –180° to +180°

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

Application

Touch probe cycle **422** measures the center point and diameter of a circular stud. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

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 47), 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

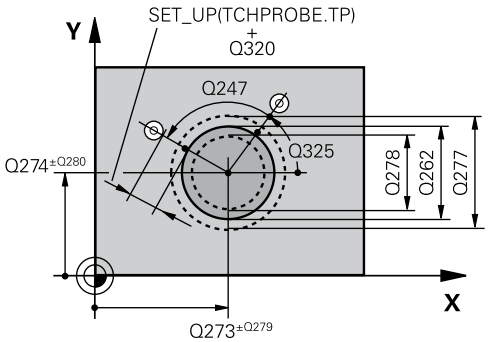
Please note while programming:

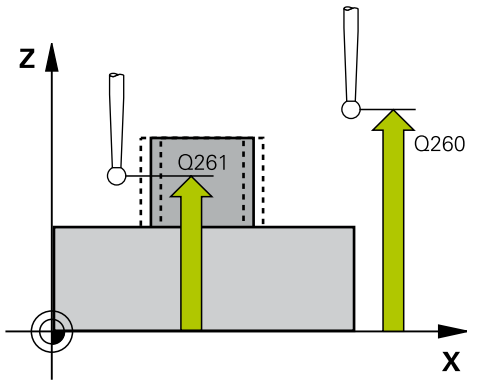
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°.
- If parameter **Q330** references a milling tool, the information in parameters **Q498** and **Q531** has no effect

  If parameter **Q330** references a turning tool, the following applies:
  - Parameters **Q498** and **Q531** have to be defined
  - The information in parameters **Q498**, **Q531**, for example from Cycle **800**, has to match this information
  - If the control compensates the position of the turning tool, the corresponding values in rows **DZL** and **DXL**, respectively, will be compensated.
  - The control also monitors the breakage tolerance, which is defined in column **LBREAK**.

Cycle parameters

- **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 170).
  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

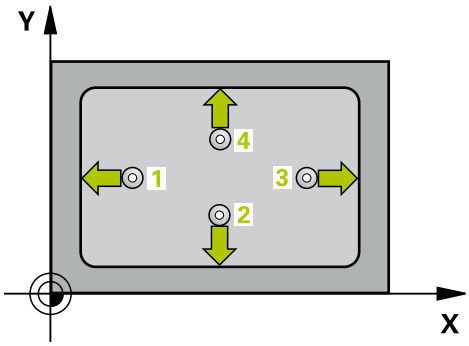
- **Q365 Type of traverse? Line=0/arc=1**: Definition of the path function with which the tool is to move between the measuring points if "traverse to clearance height" is active (**Q301**=1):
  0: Move in a straight line between machining operations
  1: Move in a circular arc on the pitch circle diameter between machining operations
- **Q498 Reverse tool (0=no/1=yes)?**: Only relevant if you specified a turning tool in parameter **Q330**. For proper monitoring of the turning tool, the control requires the exact working condition. Therefore, enter the following:
  1: Turning tool is mirrored (rotated by 180°), e.g. by Cycle **800** and parameter **Reverse the tool Q498**=1
  0: Turning tool corresponds to the description in the turning tool table (toolturn.trn), no modification, e.g. by Cycle **800** and parameter **Reverse the tool Q498**=0
- **Q531 Angle of incidence?**: Only relevant if you specified a turning tool in parameter **Q330**. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining, e.g. from Cycle **800**, **Angle of incidence? Q531**.
  Input range: –180° to +180°

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

Application

Touch probe cycle **423** finds the center, length, and width of a rectangular pocket. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

Cycle run



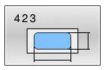
1 The control positions the touch probe at rapid traverse (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 47), at touch point **1**. The control calculates the touch points from the data in the cycle and the set-up clearance from the **SET_UP** column of 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

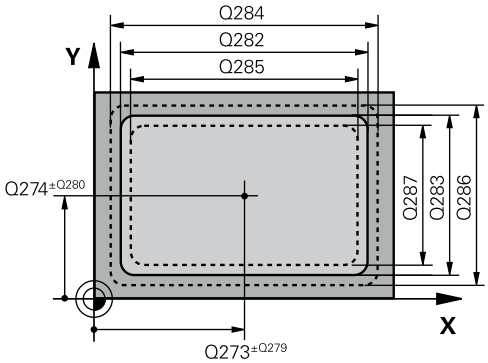
Please note while programming:

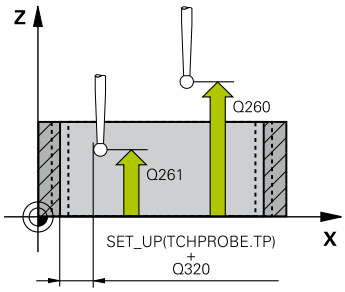
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, 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 170).
  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

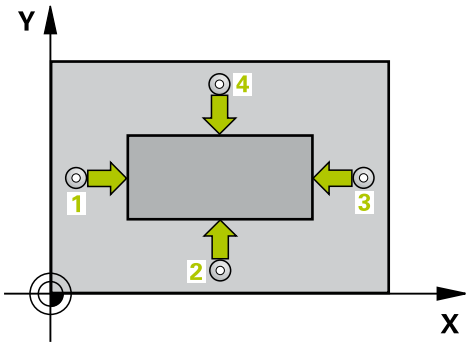
```
Q330=0      ;TOOL
```

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

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 47), 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:

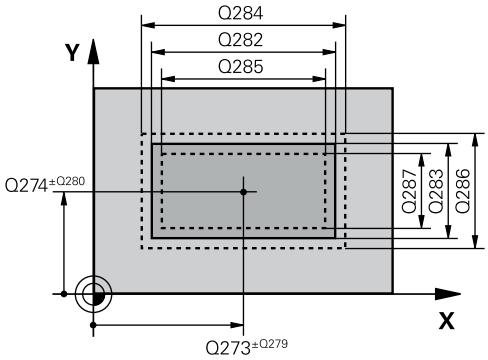
Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of side length in the reference axis
Q155	Actual value of side length in the minor axis
Q161	Deviation at center of reference axis
Q162	Deviation at center of minor axis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

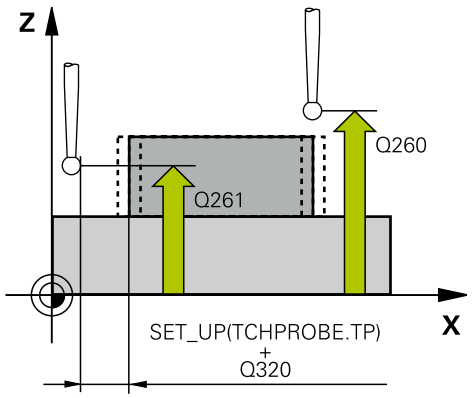
Please note while programming:

- 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

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

- **Q286 Max. size limit 2nd side length?**: Maximum permissible width of the stud.
  Input range: 0 to 99999.9999
- **Q287 Min. size limit 2nd side length?**: Minimum permissible width of the stud.
  Input range: 0 to 99999.9999
- **Q279 Tolerance for center 1st axis?**: Permissible position deviation in the principal axis of the working plane.
  Input range: 0 to 99999.9999
- **Q280 Tolerance for center 2nd axis?**: Permissible position deviation in the secondary axis of the working plane.
  Input range: 0 to 99999.9999
- **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control will save the **log file named TCHPR424.TXT** by default in the directory that also contains the associated .h file.
  2: Interrupt the program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.
- **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  0: Do not interrupt program run, do not display an error message
  1: Interrupt program run and display an error message
- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 170); 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

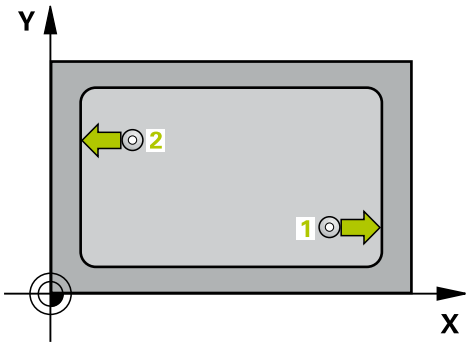
Q279=0.1	;TOLERANCE 1ST CENTER
Q280=0.1	;TOLERANCE 2ND CENTER
Q281=1	;MEASURING LOG
Q309=0	;PGM STOP TOLERANCE
Q330=0	;TOOL

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

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.

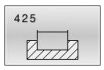
4 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

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

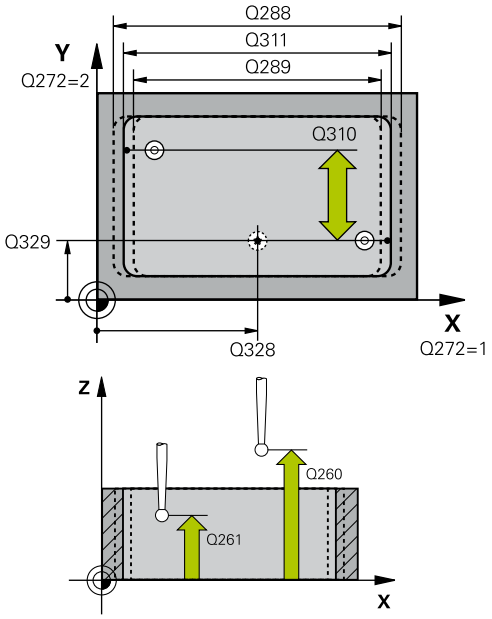
Please note while programming:

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



Example

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

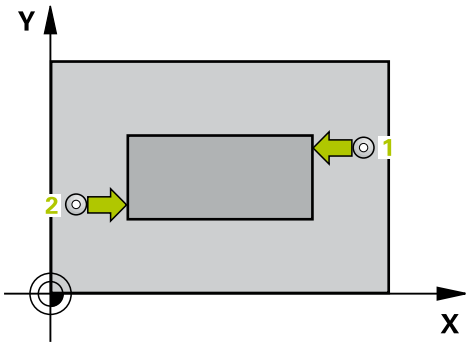
- **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  0: Do not interrupt program run, do not display an error message
  1: Interrupt program run and display an error message
- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 170); 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)

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 47), 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.

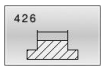
4 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

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

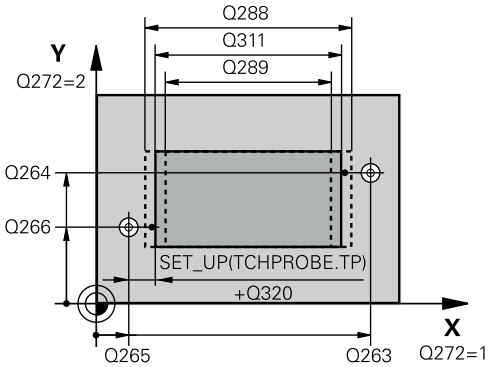
Please note while programming:

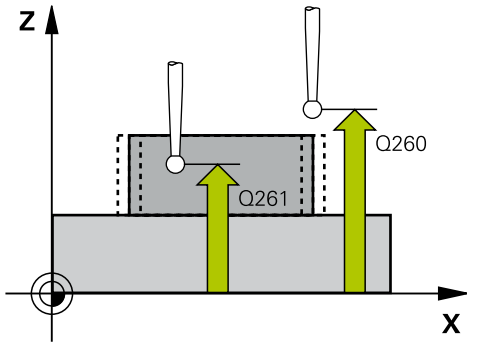
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane.
  Input range: –99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working plane.
  Input range -99999.9999 to 99999.9999
- **Q265 2nd measuring point in 1st axis?** (absolute): Coordinate of the second touch point in the principal axis of the working plane.
  Input range: –99999.9999 to 99999.9999
- **Q266 2nd measuring point in 2nd axis?** (absolute): Coordinate of the second touch point in the secondary axis of the working plane.
  Input range: –99999.9999 to 99999.9999
- **Q272 Measuring axis (1=1st / 2=2nd)?**: Axis in the working plane in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
  Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table.
  Input range: 0 to 99999.9999
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
  Input range: –99999.9999 to 99999.9999
- **Q311 Nominal length?** : Nominal value of the length to be measured.
  Input range: 0 to 99999.9999
- **Q288 Maximum limit of size?**: Maximum permissible length.
  Input range: 0 to 99999.9999
- **Q289 Minimum limit of size?**: Minimum permissible length.
  Input range: 0 to 99999.9999
- **Q281 Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control saves the **log file named TCHPR426.TXT** in the same folder as the associated NC program.
  2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.





Example

```
5 TCH PROBE 426  MEASURE RIDGE
  WIDTH
  Q263=+50   ;1ST POINT 1ST AXIS
  Q264=+25   ;1ST POINT 2ND AXIS
  Q265=+50   ;2ND PNT IN 1ST AXIS
  Q266=+85   ;2ND PNT IN 2ND AXIS
  Q272=2     ;MEASURING AXIS
  Q261=-5    ;MEASURING HEIGHT
  Q320=0     ;SET-UP CLEARANCE
  Q260=+20   ;CLEARANCE HEIGHT
  Q311=45    ;NOMINAL LENGTH
  Q288=45    ;MAXIMUM LIMIT
  Q289=44.95 ;MINIMUM LIMIT
  Q281=1     ;MEASURING LOG
  Q309=0     ;PGM STOP TOLERANCE
  Q330=0     ;TOOL
```

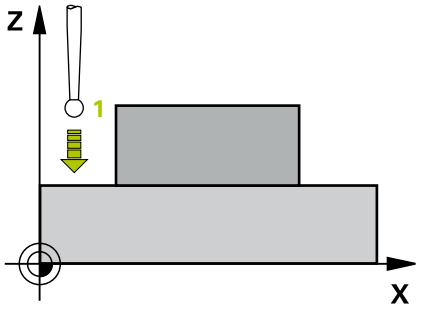
- **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  0: Do not interrupt program run, do not display an error message
  1: Interrupt program run and display an error message
- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 170); 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)

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:

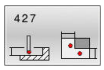
Parameter number	Meaning
Q160	Measured coordinate

Please note while programming:

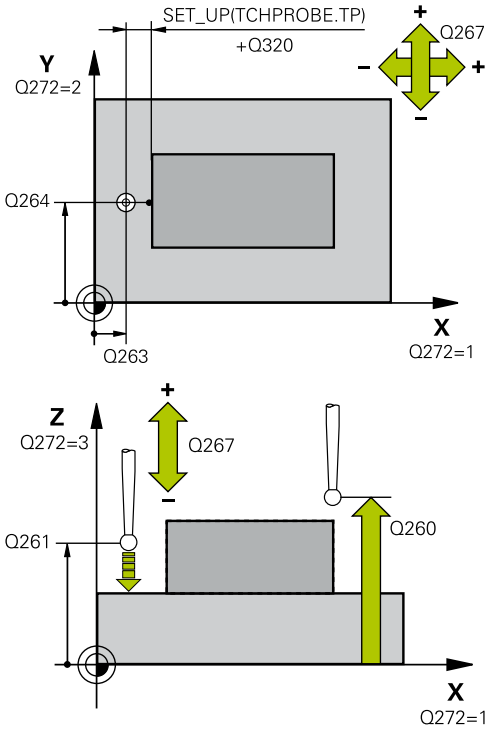
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- If an axis of the active working plane is defined as the measuring axis (**Q272** = 1 or 2), the control will perform a tool radius compensation. The control determines the direction of compensation from the defined traversing direction (**Q267**).
- If the touch probe axis is defined as the measuring axis (**Q272** = 3), the control will perform a tool length compensation.
- If parameter **Q330** references a milling tool, the information in parameters **Q498** and **Q531** has no effect

  If parameter **Q330** references a turning tool, the following applies:
  - Parameters **Q498** and **Q531** have to be defined
  - The information in parameters **Q498**, **Q531**, for example from Cycle **800**, has to match this information
  - If the control compensates the position of the turning tool, the corresponding values in rows **DZL** and **DXL**, respectively, will be compensated.
  - The control also monitors the breakage tolerance, which is defined in column **LBREAK**.

Cycle parameters



- **Q263 1st measuring point in 1st axis?** (absolute): Coordinate of the first touch point in the principal axis of the working plane.
  Input range: –99999.9999 to 99999.9999
- **Q264 1st measuring point in 2nd axis?** (absolute): Coordinate of the first touch point in the secondary axis of the working plane.
  Input range -99999.9999 to 99999.9999
- **Q261 Measuring height in probe axis?** (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
  Input range: –99999.9999 to 99999.9999
- **Q320 Set-up clearance?** (incremental): Define an additional distance between touch point and ball tip. **Q320** is added to the value of **SET_UP** in the touch probe table.
  Input range: 0 to 99999.9999
- **Q272 Meas. axis (1/2/3, 1=ref. axis)?**: Axis in which the measurement is to be made:
  1: Principal axis = measuring axis
  2: Secondary axis = measuring axis
  3: Touch probe axis = measuring axis
- **Q267 Trav. direction 1 (+1=+ / -1=-)?**: Direction in which the probe is to approach the workpiece:
  –1: Negative traverse direction
  +1: Positive traverse direction
- **Q260 Clearance height?** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
  Input range: –99999.9999 to 99999.9999
- **Q281Measuring log (0/1/2)?**: Define whether the control should create a measuring log:
  0: Do not create a measuring log
  1: Create a measuring log: The control saves the **log file named TCHPR427.TXT** in the same folder as the associated NC program.
  2: Interrupt the program run and display the measuring log on the screen.Resume the NC program run with **NC Start**.
- **Q288 Maximum limit of size?**: Maximum permissible measured value.
  Input range: –99999.9999 to 99999.9999
- **Q289 Minimum limit of size?**: Minimum permissible measured value.
  Input range: –99999.9999 to 99999.9999



Example

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

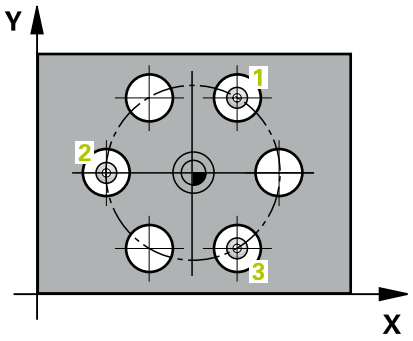
- **Q309 PGM stop if tolerance exceeded?**: Define whether in the event of a violation of tolerance limits, the control is to interrupt program run and display an error message:
  0: Do not interrupt program run, do not display an error message
  1: Interrupt program run and display an error message
- **Q330 Tool for monitoring?**: Define whether the control is to monitor the tool (see "Tool monitoring", Page 170); 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
- **Q498 Reverse tool (0=no/1=yes)?**: Only relevant if you specified a turning tool in parameter **Q330**. For proper monitoring of the turning tool, the control requires the exact working condition. Therefore, enter the following:
  1: Turning tool is mirrored (rotated by 180°), e.g. by Cycle **800** and parameter **Reverse the tool Q498**=1
  0: Turning tool corresponds to the description in the turning tool table (toolturn.trn), no modification, e.g. by Cycle **800** and parameter **Reverse the tool Q498**=0
- **Q531 Angle of incidence?**: Only relevant if you specified a turning tool in parameter **Q330**. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining, e.g. from Cycle **800**, **Angle of incidence? Q531**.
  Input range: –180° to +180°

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

Application

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

Cycle run



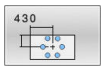
1 The control positions the touch probe at rapid traverse speed (value from **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 47), to the programmed center point of the first hole **1**.
2 Then the probe moves to the entered measuring height and probes four points to 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

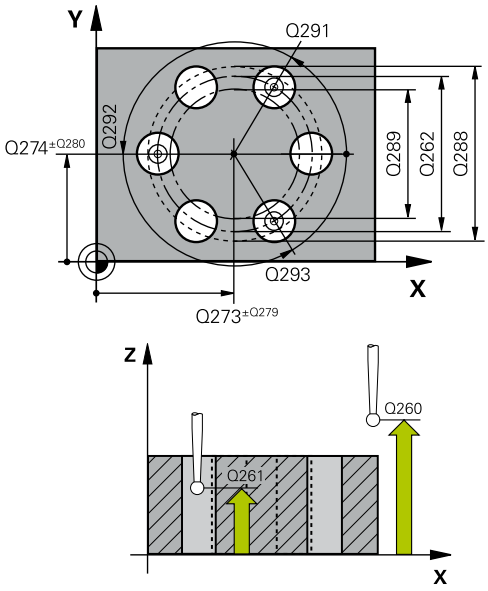
Please note while programming:

- 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 170); 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)

Application

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

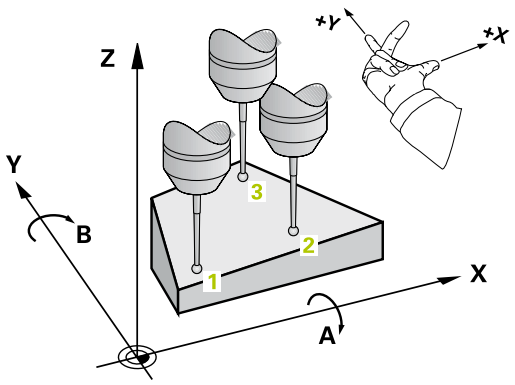
Cycle run

1 The control positions the touch probe at rapid traverse (value from the **FMAX** column), using positioning logic (see "Executing touch probe cycles", Page 47), 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

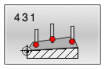
Danger of collision!

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.

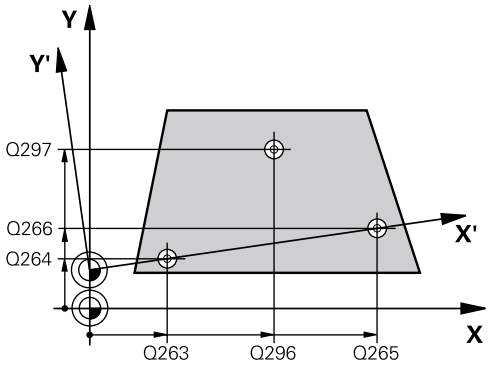
- Make sure to program **SYM** (**SEQ**) + or **SYM** (**SEQ**) -

- 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.

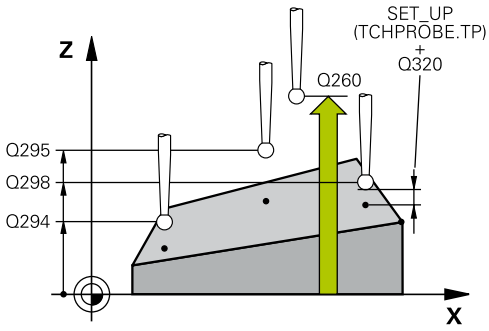
Cycle parameters



- **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

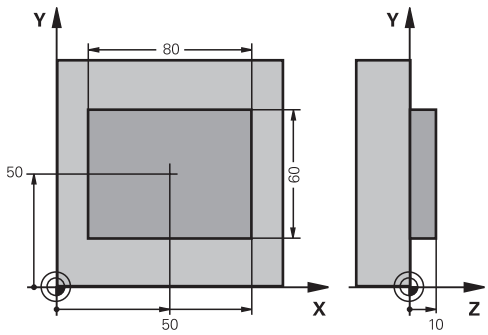
```
5 TCH PROBE 431  MEASURE PLANE
    Q263=+20   ;1ST POINT 1ST AXIS
    Q264=+20   ;1ST POINT 2ND AXIS
    Q294=-10   ;1ST POINT 3RD AXIS
    Q265=+50   ;2ND PNT IN 1ST AXIS
    Q266=+80   ;2ND PNT IN 2ND AXIS
    Q295=+0    ;2ND PNT IN 3RD AXIS
    Q296=+90   ;3RD PNT IN 1ST AXIS
    Q297=+35   ;THIRD 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

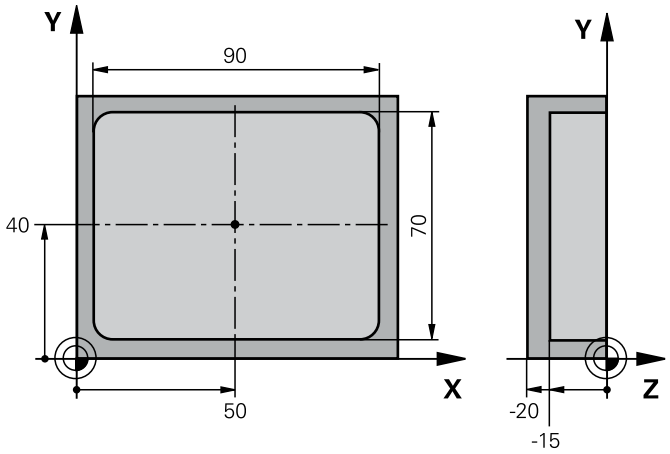
- Rough the rectangular stud with 0.5 mm finishing allowance
- Measure the rectangular stud
- Finish the rectangular stud, taking the measured values into account



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.

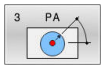
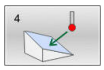
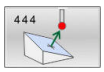
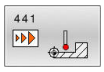
NOTICE

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) ■ Touch probe cycle for defining OEM cycles	215
	MEASURING IN 3-D (Cycle 4) ■ Measuring any position	217
	PROBING IN 3-D (Cycle 444, ISO: G444) ■ Measuring any position ■ Determining the deviation from the nominal coordinates	219
	FAST PROBING (Cycle 441, DIN/ISO: G441) ■ Touch probe cycle for defining various touch probe parameters	224

7.2 MEASURING (Cycle 3)

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.

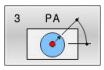
Please note while programming:

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)

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

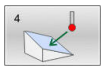
Danger of collision!

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

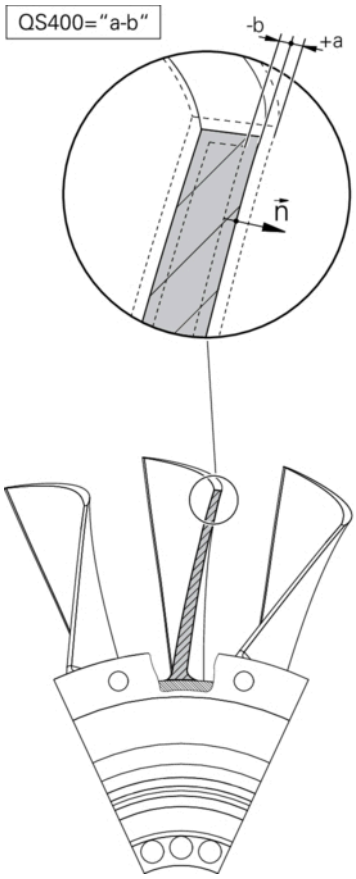
Example

```
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 PROBING IN 3-D (Cycle 444, ISO: G444)

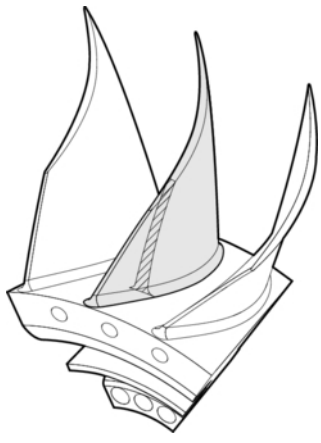
Application

Refer to your machine manual!
This function must be enabled and adapted by the machine tool builder.



Cycle **444** checks one specific point on the surface of a component. This cycle is used, for example, to measure free-form surfaces of molded parts. It can be determined whether a point on the surface of the component lies in an undersize or oversize range compared to a nominal coordinate. The operator can subsequently perform further machining steps, such as reworking.

Cycle **444** probes any point in three dimensions, and determines the deviation to a nominal coordinate. A normal vector, defined in parameters **Q581**, **Q582**, and **Q583**, is used for this purpose. The normal vector is perpendicular to an imagined surface in which the nominal coordinate is located. The normal vector points away from the surface, and does not determine the probing path. It is advisable to determine the normal vector with the help of a CAD or CAM system. A tolerance range **QS400** defines the permissible deviation between the actual and nominal coordinate along the normal vector. This way you define, for example, that the program is to be interrupted if an undersize is detected. Additionally, the control outputs a log and the deviations are stored in the Q parameters listed below.



Cycle run

1. Starting from the current position, the touch probe traverses to a point on the normal vector that is at the following distance from the nominal coordinate: Distance = ball-tip radius + **SET_UP** value from the tchprobe.tp table (TNC:\table \tchprobe.tp) + **Q320** Pre-positioning takes a clearance height into account. For more information on the probing logic, see "Executing touch probe cycles", Page 47
2. The touch probe then approaches the nominal coordinate. The probing distance is defined by DIST, not by the normal vector! The normal vector is only used for the correct calculation of the coordinates.
3. After the control has saved the position, the touch probe is retracted and stopped. The control saves the measured coordinates of the contact point in Q parameters.
4. Finally, the control retracts the touch probe by the value that you defined in parameter **MB** in the direction opposite to the probing direction.

Result parameters

The control stores the probing results in the following parameters:

Parameter number	Meaning
Q151	Measured position in principal axis
Q152	Measured position in secondary axis
Q153	Measured position in tool axis
Q161	Measured deviation in reference axis
Q162	Measured deviation in secondary axis
Q163	Measured deviation in tool axis
Q164	Measured 3-D deviation ■ Less than 0: Undersize ■ Greater than 0: Oversize
Q183	Workpiece status: ■ - 1 = undefined ■ 0 = good ■ 1 = Rework ■ 2 = Scrap

Log function

Once probing has finished, the control generates a log in HTML format. The log includes the results from the principal, secondary, and tool axes as well as the 3-D deviation. The control saves the log in the same folder in which the *.h file is located (as long as no path has been configured for FN16).

The log contains the following data on the principal, secondary, and tool axes:

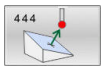
- Actual probing direction (as a vector in the input system). The value of the vector corresponds to the configured probing path
- Defined nominal coordinate
- If a tolerance **QS400** was defined: Upper and lower allowances are output, as well as the determined deviation along the normal vector
- Ascertained actual coordinate
- Colored display of the values (green for "good," orange for "rework," red for "scrap")

Please note while programming!

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.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- In order to obtain exact results from the touch probe being used, you need to perform 3#D calibration before executing Cycle **444**. Software option 92, **3D#ToolComp**, is required for 3#D calibration.
- Cycle **444** generates a measuring log in HTML format.
- An error message is output if Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, or Cycle **26 AXIS-SPECIFIC SCALING** is active before Cycle **444** is run.
- For probing, an active TCPM will be taken into account. While the TCPM is active, probing of positions is even possible if the position resulting from the **Tilt working plane** function is inconsistent with the current position of the rotary axes.
- If your machine is equipped with a controlled spindle, you should activate the angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3-D touch probe.
- Cycle **444** references all coordinates to the input system.
- The control writes the measured values to return parameters, see "Application", Page 219.
- The workpiece status good/rework/scrap is set via Q parameter **Q183**, independent of parameter **Q309** (see "Application", Page 219).

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
- **Q581 Surface-normal in ref. axis?** Enter here the surface normal in the direction of the principal axis. The surface normal of a point is normally output by a CAD/CAM system.
  Input range: –10 to 10
- **Q582 Surface-normal in minor axis?** Enter here the surface normal in the direction of the secondary axis. The surface normal of a point is normally output by a CAD/CAM system.
  Input range: –10 to 10
- **Q583 Surface-normal in tool axis?** Enter here the surface normal in the direction of the tool axis. The surface normal of a point is normally output by a CAD/CAM system.
  Input range: –10 to 10
- **Q320 Set-up clearance?** (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

```
4 TCH PROBE 444 PROBING IN 3-D
    Q263=+0    ;1ST POINT 1ST AXIS
    Q264=+0    ;1ST POINT 2ND AXIS
    Q294=+0    ;1ST POINT 3RD AXIS
    Q581=+1    ;NORMAL IN REF. AXIS
    Q582=+0    ;NORMAL IN MINOR AXIS
    Q583=+0    ;NORMAL IN TOOL AXIS
    Q320=+0    ;SAFETY CLEARANCE
    Q260=100   ;CLEARANCE HEIGHT
    QS400="1-1";TOLERANCE
    Q309=+0    ;ERROR REACTION
```

- **QS400 Tolerance value?** Specify a tolerance range that will be monitored by the cycle. The tolerance defines the deviation permitted along the surface normals. This deviation is determined between the nominal coordinate and the actual coordinate of the workpiece. (The surface normal is defined by **Q581** through **Q583**, and the nominal coordinate is defined by **Q263**, **Q264**, and **Q294**.) The tolerance value is distributed over the axes, depending on the normal vector:
  Example: QS400 ="0.4-0.1" means: upper allowance = nominal coordinate +0.4, lower allowance = nominal coordinate –0.1. The following tolerance range thus results for the cycle: "nominal coordinate + 0.4" to "nominal coordinate - 0.1".
  Example: QS400 ="0.4" means: upper allowance = nominal coordinate +0.4, lower allowance = nominal coordinate. The following tolerance range thus results for the cycle: "nominal coordinate + 0.4" to "nominal coordinate".
  Example: QS400 ="-0.1" means: upper allowance = nominal coordinate, lower allowance = nominal coordinate –0.1. The following tolerance range thus results for the cycle: "nominal coordinate" to "nominal coordinate –0.1".
  Example: QS400 =" " means: No tolerance band.
  Example: QS400 ="0" means: No tolerance band.
  Example: QS400 ="0.1+0.1" means: No tolerance band.
- **Q309 Reaction to tolerance error?** Specify whether the control is to interrupt the program run and display a message if a deviation is detected:
  0: If the tolerance is exceeded, do not the interrupt program run, do not display an error message
  1: If the tolerance is exceeded, interrupt the program run and display an error message
  2: If the value of the determined actual coordinate along the surface normal vector is less than the nominal coordinate, the control displays a message and interrupts the NC program run. However, there will be no error message if the value of the determined actual coordinate is higher than that of the nominal coordinate.

7.5 FAST PROBING (Cycle 441, DIN/ISO: G441)

Application

You can use touch probe cycle **441** to globally specify various touch probe parameters (e.g. the positioning feed rate) for all subsequently used touch probe cycles.

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
```

7.6 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.

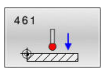
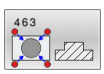
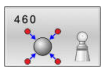
The control provides calibration cycles for calibrating the length and the radius:

Proceed as follows:

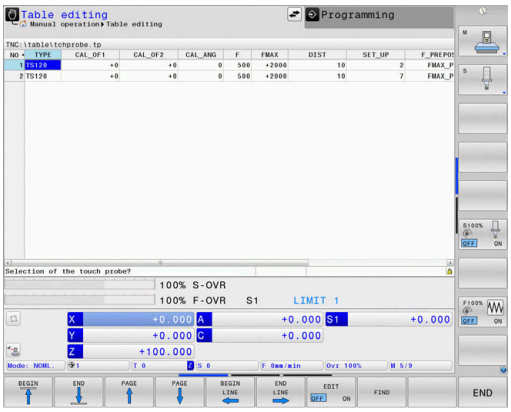
- Press the **TOUCH PROBE** key.

- 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) ■ Calibrating the length	228
	CALIBRATION OF A TS IN A RING (Cycle 462, DIN/ISO: G462) ■ Measuring the radius using a ring gauge ■ Measuring the center offset using a ring gauge	230
	TS CALIBRATION ON STUD (Cycle 463, DIN/ISO: G463) ■ Measuring the radius using a stud or a calibration pin ■ Measuring the center offset using a stud or a calibration pin	233
	CALIBRATION OF TS ON A SPHERE (Cycle 460, DIN/ISO: G460) ■ Measuring the radius using a calibration sphere ■ Measuring the center offset using a calibration sphere	236

7.7 Displaying calibration values



The control saves the effective length and effective radius of the touch probe in the tool table. The control saves the touch probe center offset to the touch probe table in the columns **CAL_OF1** (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

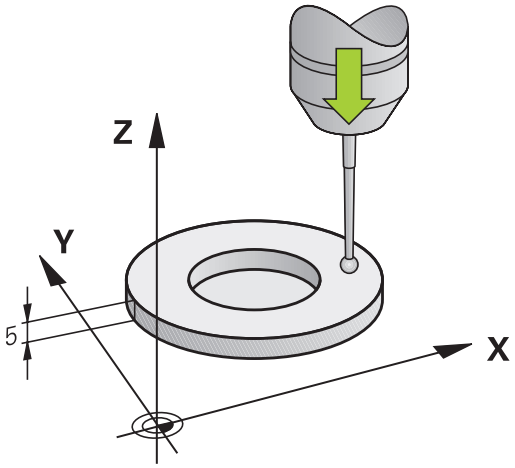
7.8 TS CALIBRATION OF TOOL LENGTH (Cycle 461, DIN/ISO: G461)

Application

Refer to your machine manual!

Before starting the calibration cycle, you must set the preset in the spindle axis so that Z=0 on the machine table; you must also pre-position the touch probe above the calibration ring.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.



Cycle 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

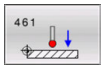
Danger of collision!

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

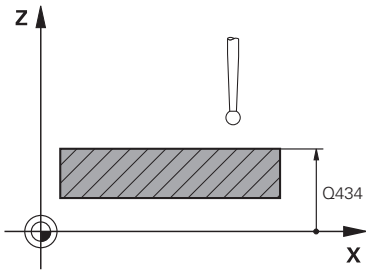
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- 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



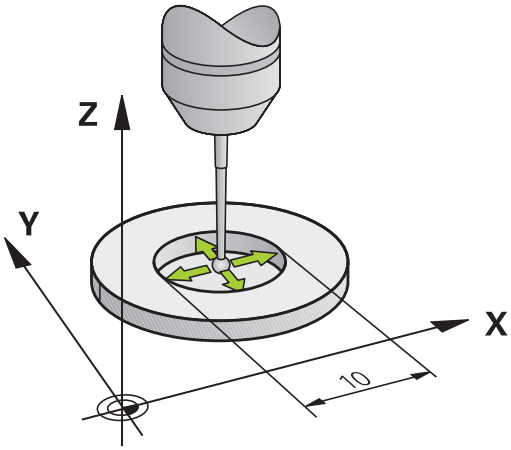
Example

```
5 TCH PROBE 461  TS CALIBRATION OF
    TOOL LENGTH
    Q434=+5     ;PRESET
```

7.9 CALIBRATION OF A TS IN A RING (Cycle 462, DIN/ISO: G462)

Application

Refer to your machine manual!



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

When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run, the control finds the center point of the calibration ring or pin (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is determined. If the touch probe allows probing from opposite orientations, the center offset is determined during another run.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.

The orientation of the touch probe determines the calibration routine:

- No orientation possible, or orientation in only one direction: The control executes one approximate and one fine measurement, and then ascertains the effective ball-tip radius (column R in tool.t).
- Orientation possible in two directions (e.g. HEIDENHAIN touch probes with cable): The control executes one approximate and one fine measurement, rotates the touch probe by 180°, and then executes four more probing routines. The center offset (CAL_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations.
- Any orientation possible (e.g. HEIDENHAIN infrared touch probes): Probing operation: see "Orientation possible in two directions").

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

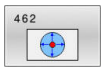
Danger of collision!

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

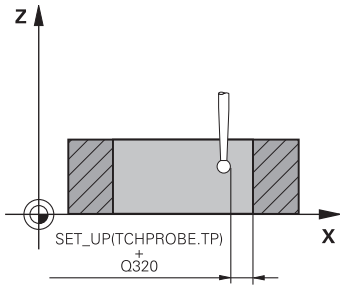
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- 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
```

7.10 TS CALIBRATION ON STUD (Cycle 463, DIN/ISO: G463)

Application

Refer to your machine manual!

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

When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run the control finds the midpoint of the calibration ring or pin (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is ascertained. If the touch probe allows probing from opposite orientations, the center offset is determined during another run.

A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, TCHPRAUTO.html will contain all the measuring logs.

The orientation of the touch probe determines the calibration routine:

- No orientation possible, or orientation in only one direction: The control executes one approximate and one fine measurement, and then ascertains the effective ball-tip radius (column R in tool.t).
- Orientation possible in two directions (e.g. HEIDENHAIN touch probes with cable): The control executes one approximate and one fine measurement, rotates the touch probe by 180°, and then executes four more probing routines. The center offset (CAL_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations.
- Any orientation possible (e.g. HEIDENHAIN infrared touch probes): Probing operation: see "Orientation possible in two directions").

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

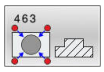
Danger of collision!

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

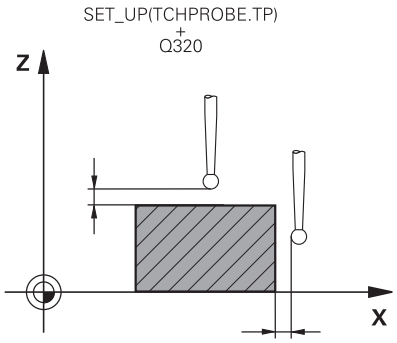
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- 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



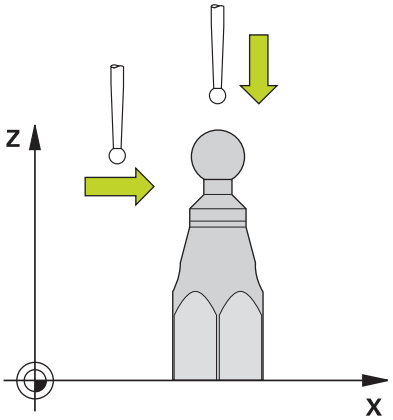
Example

```
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
```

7.11 CALIBRATION OF TS ON A SPHERE (Cycle 460, DIN/ISO: G460)

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.

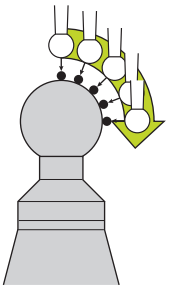
With Cycle **460** you can calibrate a triggering 3#D touch probe automatically using an exact calibration sphere.

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. This 3#D calibration is necessary if you want to achieve a very high accuracy with Cycle **444** 3#D Probing (see "PROBING IN 3-D (Cycle 444, ISO: G444)", Page 219).

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

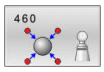
Danger of collision!

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

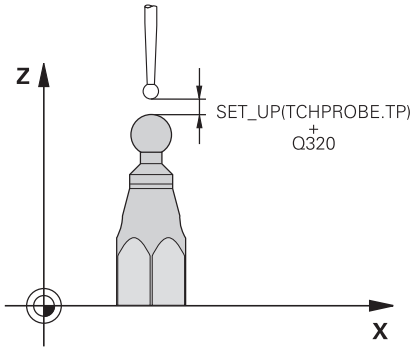
- The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- Reset any coordinate transformations beforehand.

- 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 use Cycle **444**, you should first perform a 3#D calibration.
- If you program **Q455**=1 to 30, a table will be stored under TNC:\system\3D-ToolComp*.
- If there is already a reference to a calibration table (entry in DR2TABLE), this table will be overwritten.
- If there is no reference to a calibration table (entry in DR2TABLE), then, in dependence of the tool number, a reference and the associated table will be created.

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

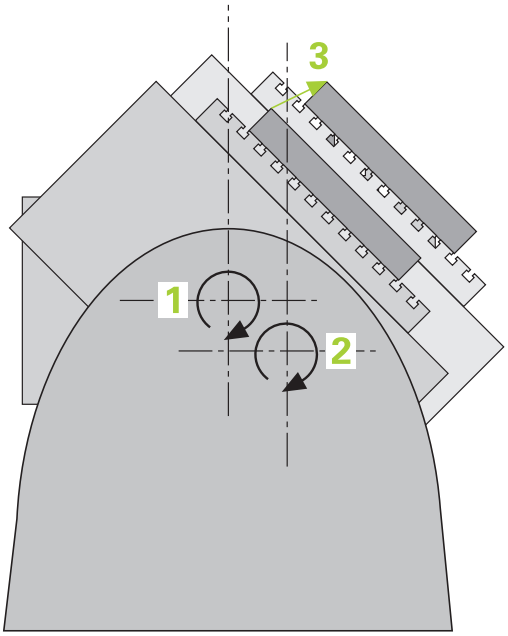
Touch Probe Cycles: Automatic Kinematics Measurement

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.

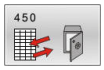


The **KinematicsOpt** function of the control is an important component that helps you to really meet these complex requirements: a 3-D touch probe cycle measures the rotary axes on your machine fully automatically, regardless of whether they are realized as tables or spindle heads. For this purpose, 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
	SAVE KINEMATICS (Cycle 450, DIN/ISO: G450, option 48) ■ Storing the active machine kinematic configuration ■ Restoring previously saved kinematic configuration	246
	MEASURE KINEMATICS (Cycle 451, DIN/ISO: G451, option 48) ■ Automatic checking of the machine kinematic configuration ■ Optimizing the machine kinematic configuration	249
	PRESET COMPENSATION (Cycle 452, DIN/ISO: G452, option 48) ■ Automatic checking of the machine kinematic configuration ■ Optimizing the kinematic transformation chain of the machine	263
	KINEMATICS GRID (Cycle 453, DIN/ISO: G453, option 48) ■ Automatic checking depending on the tilting axis position of the machine kinematic configuration ■ Optimizing the machine kinematic configuration	274

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 **mStrobeRotAxPos** machine parameter (no. 204803), you have to position the rotary axes to 0° (ACTUAL system) before starting one of the KinematicsOpt cycles (except for **450**).

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.

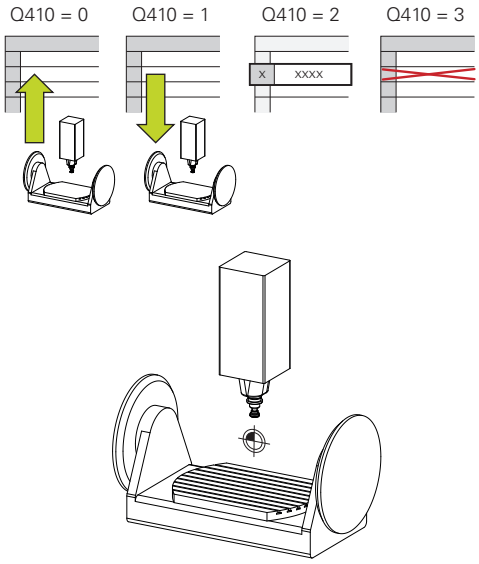
If machine parameters were changed through the KinematicsOpt cycles, the control must be restarted. Otherwise the changes could be lost in certain circumstances.

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.

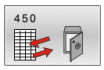


Please note while programming:

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.

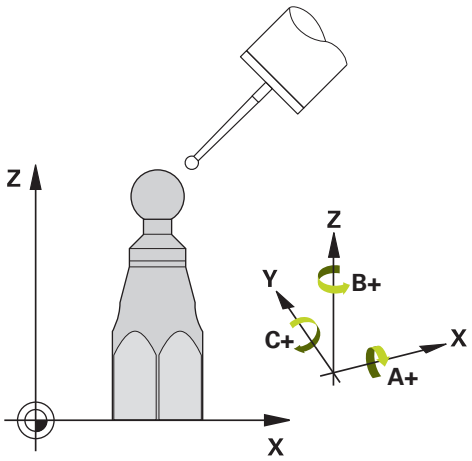
Operating notes:

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

Machines with Hirth-coupled axes

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

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

- Note the documentation of the machine tool builder

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° - (-30°)) / (4 – 1) = 120 / 3 = 40°

Measuring position 1 = **Q411** + 0 * stepping angle = –30° –> –30°

Measuring position 2 = **Q411** + 1 * stepping angle = +10° –> 9°

Measuring position 3 = **Q411** + 2 * stepping angle = +50° –> 51°

Measuring position 4 = **Q411** + 3 * stepping angle = +90° –> 90°

Choice of number of measuring points

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

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

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

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

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

Choice of the calibration sphere position on the machine table

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

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

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 262).

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

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

- **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 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:

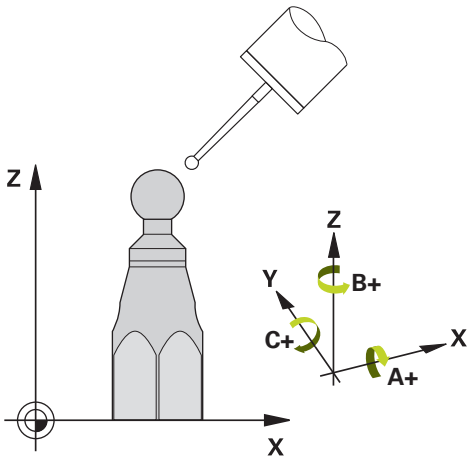
- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Mode used (0=Check/1=Optimize position/2=Optimize pose)
- Active kinematic number
- Entered calibration sphere radius
- For each measured rotary axis:
  - Starting angle
  - End angle
  - Angle of incidence
  - 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 249). 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

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

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

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

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

Drift compensation

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

Log function

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

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

Notes on log data

(see "Log function", Page 262)

8.6 KINEMATICS GRID (Cycle 453, DIN/ISO: G453, option 48)

Application

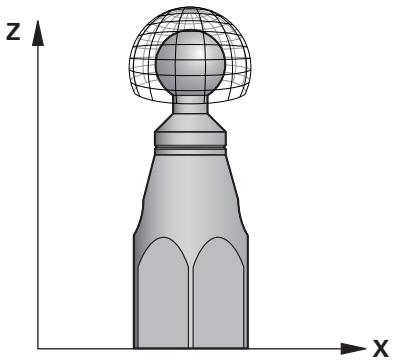
Refer to your machine manual!

KinematicsOpt (software option 48) is required.

KinematicsComp (software option 52) is required.

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

To use this cycle, your machine tool builder needs to create and configure a compensation table (*.kco) first and enter some more settings.



Even if your machine was already optimized regarding positioning errors (e.g. via Cycle **451**), residual errors at the Tool Center Point (**TCP**) during tilting of the rotary axes may remain. Such errors occur particularly with swivel-head machines. These can result, for example, from component errors (e.g. a bearing error) with head rotation axes.

Cycle **453 KINEMATICS GRID** enables these errors to be detected and compensated in accordance with the tilting axis positions. Options 48 (**KinematicsOpt**) and 52 (**KinematicsComp**) are required. With this cycle and using a 3-D TS touch probe, you measure a HEIDENHAIN calibration sphere that you have attached to the machine table. The cycle then moves the touch probe automatically to positions in a grid-line arrangement around the calibration sphere. The machine tool builder defines these tilting axis positions. You can arrange the positions in up to three dimensions. (Each dimension is a rotary axis.) After the probing process on the sphere, compensation of the errors can be performed using a multi-dimensional table. The machine tool builder defines this compensation table (*.kco) and specifies its storage location.

When using Cycle **453**, run it at different positions in the workspace. This allows you to check immediately if a compensation with Cycle **453** has the desired positive effect on the machine's accuracy. Only when the desired improvements are achieved with the same compensation values at several positions is such a type of compensation suitable for the respective machine. If this is not the case, then the errors are to be sought outside the rotary axes.

Perform the measurement with Cycle **453** in an optimized condition regarding the rotary axis positioning errors. For this purpose, use e.g. Cycle **451** beforehand.

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

The control then optimizes the accuracy of your machine. For this purpose, it automatically saves the compensation values resulting

from a measurement in a compensation table (*.kco). (This applies to mode **Q406**=1.)

Cycle run

1 Clamp the calibration sphere and check for potential collisions.
2 In Manual mode of operation, set the preset to the center of the sphere or, if you defined **Q431=1** or **Q431=3**: Manually position the touch probe above the calibration sphere in the touch probe axis and at the center of the sphere in the working plane.
3 Select one of the Program Run operating modes and start the NC program
4 The cycle is executed in accordance with the setting in **Q406** (–1=Delete mode / 0=Test mode / 1=Compensate mode)

During presetting, the programmed radius of the calibration sphere will only be monitored for the second measurement. The reason is that if pre-positioning with respect to the calibration sphere is inaccurate and you then start presetting, the calibration sphere will be probed twice.

Various modes (Q406)

Delete mode Q406 = -1

- The axes are not moved
- The control writes all values to the compensation table (*.kco), setting them to "0". The result is that no further compensations will be effective for the currently selected kinematics.

Test mode Q406 = 0

- The control probes the calibration sphere.
- The results are saved to a log in html format that is stored in the directory as the current NC program

Compensate mode Q406 = 1

- The control probes the calibration sphere.
- The control writes the deviations to the compensation table (*.kco). The table is updated and the compensation settings are immediately effective.
- The results are saved to a log in html format that is stored in the directory as the current NC program

Choice of the calibration sphere position on the machine table

In principle, you can fix the calibration sphere to any accessible position on the machine table and also on fixtures or workpieces. It is recommended to clamp the calibration sphere as closely as possible to the position intended for subsequent machining.

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

Please note while programming:

KinematicsOpt (software option 48) is required.
KinematicsComp (software option 52) is required.

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

Your machine tool builder defines the storage location of the compensation table (*.kco).

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°.

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

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.
- As with Cycles **451** and **452**, Cycle **453** ends with active 3D#ROT in automatic mode, matching the position of the rotary axes.
- Before defining the cycle, you must set the preset to the center of the calibration sphere and activate it, or you set input parameter **Q431** to 1 or 3, respectively.
- For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while probe monitoring is inactive.
- Programming in inches: The control always records the log data and results of measurement in millimeters.
- If you have activated 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.

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

Cycle parameters

- **Q406 Mode (-1/0/+1)** : Define whether the control should write the values to the compensation table (*.kco) with 0, check the currently existing deviations, or perform a compensation. A log (*.html) is created.
  -1: Delete values in the compensation table (*.kco). The compensation values for TCP positioning errors are set to 0 in the compensation table (*.kco). The control will not perform any probing. No results will be output to the log (*.html).
  0: Check TCP positioning errors. The control measures the TCP positioning errors in accordance with the rotary axis positions but does not write values to the compensation table (*.kco). The control displays the standard and maximum deviation in a log (*.html).
  1: Compensate TCP positioning errors. The control measures the TCP positioning errors in accordance with the rotary axis positions and writes the deviations to the compensation table (*.kco). The compensations are then immediately effective. The control displays the standard and maximum deviation in a log (*.html).
- **Q407 Radius of calib. sphere?** Enter the exact radius of the calibration sphere to be used.
  Input range: 0.0001 to 99.9999
- **Q320 Set-up clearance?** (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

Probing with Cycle 453

```
4 TOOL CALL "TCH PROBE" Z
6 TCH PROBE 453 KINEMATICS GRID
    Q406=0      ;MODE
    Q407=12.5   ;SPHERE RADIUS
    Q320=0      ;SET-UP CLEARANCE
    Q408=0      ;RETR. HEIGHT
    Q253=750    ;F PRE-POSITIONING
    Q380=0      ;REFERENCE ANGLE
    Q423=4      ;NO. OF PROBE POINTS
    Q431=0      ;PRESET
```

- **Q253 Feed rate for pre-positioning?** Specify the traversing speed of the tool during pre-positioning in mm/min.
  Input range: 0.0001 to 99999.9999; alternatively **FMAX**, **FAUTO**, **PREDEF**
- **Q380 Ref. angle in ref. axis?** (absolute): Enter the reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis.
  Input range: 0 to 360.0000
- **Q423 Number of probes?** Define the number of probe measurements with which the control is to measure the calibration sphere in the plane. 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

Log function

After running Cycle **453**, the control will create a log **(TCHPR453.html)** and save it in the folder where the current NC program resides. It contains the following data:

- Date and time of protocol creation
- Path of the NC program from which the cycle was run
- Number and name of the currently active tool
- Mode
- Measured data: Standard deviation and maximum deviation
- Information at which position in degrees (°) the maximum deviation occurred
- Number of measuring positions

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) ■ Calibrating the tool touch probe	287
		Measuring tool length (Cycle 31 or 481, DIN/ISO: G481) ■ Measuring the tool length	290
		Measuring a tool radius (Cycle 32 or 482, ISO: G482) ■ Measuring the tool radius	294
		Measuring tool length and radius (Cycle 33 or 483, ISO: G483) ■ Measuring the tool length and radius	298
		CALIBRATE IR TT (Cycle 484, ISO: G484) ■ Calibrating the tool touch probe (e.g., infrared tool touch probe)	302
		MEASURE LATHE TOOL (Cycle 485, ISO: G485, option 50) ■ Measurement of turning tools	305

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, and 485** can be hidden with the optional **hideMeasureTT** machine parameter (no. 128901).

Programming and operating notes:

- Before you start working with the touch probe cycles, check all machine parameters defined in **ProbeSettings** > **CfgTT** (no. 122700) and **CfgTTRoundStylus** (no. 114200) or **CfgTTRectStylus** (no. 114300).
- When measuring a stationary tool, the Control will use the feed rate for probing defined in the **probingFeed** machine parameter (no. 122709).

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

The spindle speed is calculated as follows:

n = **maxPeriphSpeedMeas** / (r • 0.0063) where

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

The probing feed rate is calculated as follows:
v = measuring tolerance • n with

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

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

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

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

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

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

Tool radius	Measuring tolerance
Up to 30 mm	**measureTolerance1**
30 to 60 mm	2 • **measureTolerance1**
60 to 90 mm	3 • **measureTolerance1**
90 to 120 mm	4 • **measureTolerance1**

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

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

Measuring tolerance = (r • **measureTolerance1**)/ 5 mm) where

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

Entries in the tool table for milling and turning tools

Abbr.	Inputs	Dialog
CUT	Number of teeth (20 teeth maximum)	**Number of teeth?**
LTOL	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: length?**
RTOL	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Wear tolerance: radius?**
DIRECT.	Cutting direction of the tool for measuring a rotating tool	**Cutting direction (M3 = -)?**
R-OFFS	Tool length measurement: Tool offset between stylus center and tool center. Default setting: No value entered (offset = tool radius)	**Tool offset: radius?**
L-OFFS	Radius measurement: Tool offset between upper edge of stylus and lower edge of tool in addition to **offsetToolAxis**. Default: 0	**Tool offset: length?**
LBREAK	Permissible deviation from tool length L for breakage detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Breakage tolerance: length?**
RBREAK	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the control locks the tool (status **L**). Input range: 0 to 0.9999 mm	**Breakage tolerance: radius?**

Input examples for common tool types

Tool type	CUT	R-OFFS	L-OFFS
Drill	No function	0: No offset required because tool tip is to be measured	
End mill	4: four cutting edges	R: Offset required because the tool diameter is greater than the contact plate diameter of the TT	0: No additional offset required during radius measurement. Offset from **offsetToolAxis** (no. 122707) used.
Spherical cutter with a diameter of 10 mm	4: four cutting edges	0: No offset required because the south pole of the ball is to be measured.	5: At a diameter of 10 mm, the tool radius will be defined as offset. If this is not the case, the diameter of the spherical cutter will be measured too far down. So the tool diameter will not be correct.

9.2 CALIBRATE TT (Cycle 30 or 480, ISO: G480)

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 283). 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)

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 293

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)

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 283). 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 297

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)

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 283). 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 301

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)

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
```

9.7 MEASURE LATHE TOOL (Cycle 485, ISO: G485, option 50)

Application

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

Cycle **485 MEASURE LATHE TOOL** is available for the measurement of lathe tools using the tool touch probe from HEIDENHAIN. The control measures the tool in a fixed programmed sequence.

Cycle run

1 The control positions the lathe tool to the clearance height
2 The lathe tool is oriented based on the entries in **TO** and **ORI**
3 The control moves the tool to the measuring position in the principal axis; traverse movement is interpolated in the principal and secondary axes
4 Then the lathe tool moves to the measuring position in the tool axis
5 The tool is measured. Depending on the definition of **Q340**, either tool dimensions are changed or the tool is locked
6 The measuring result is transferred to the result parameter **Q199**
7 After the measurement has been performed, the control positions the tool in the tool axis to the clearance height

Result parameter Q199:

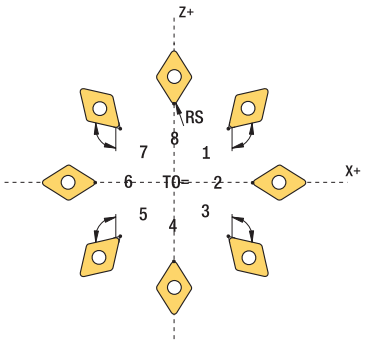
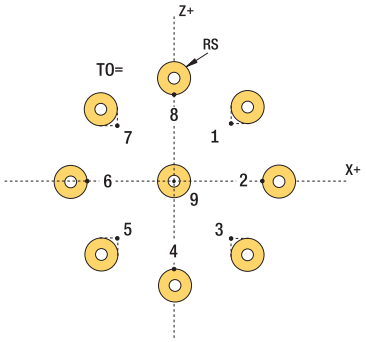
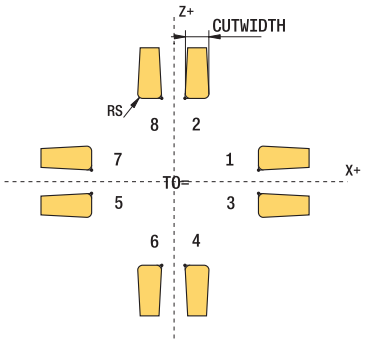
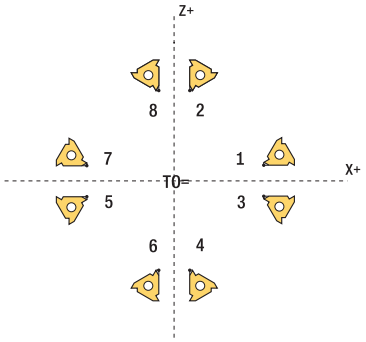
Result	Meaning
0	Tool dimensions within the tolerance **LTOL** / **RTOL** Tool is not locked
1	Tool dimensions outside the tolerance **LTOL** / **RTOL** Tool is locked
2	Tool dimensions outside the tolerance **LBREAK** / **RBREAK** Tool is locked

The cycle uses the following entries from toolturn.trn:

Abbr.	Entries	Dialog
ZL	Tool length 1 (**Z** direction)	**Tool length 1?**
XL	Tool length 2 (**X** direction)	**Tool length 2?**
DZL	Delta value of tool length 1 (**Z** direction), is added to **ZL**	**Tool length oversize 1**
DXL	Delta value of tool length 2 (**X** direction), is added to **XL**	**Tool length oversize 2**
RS	Cutting edge radius: if contours were programmed with radius compensation **RL** or **RR**, the control takes the cutting edge radius into account in turning cycles, and performs cutting radius compensation	**Cutting edge radius?**
TO	Tool orientation: from the tool orientation, the control determines the position of the tool tip and, depending on the selected tool type, additional information such as the tool angle direction, position of the tool reference point, etc. This information is necessary, for example, for calculating the cutting radius compensation, milling cutter radius compensation, plunge angle, etc.	**Tool orientation?**
ORI	Spindle orientation angle: angle of the indexable insert to the main axis	**Angle of spindle orientation?**
TYPE	Type of turning tool: Roughing tool **ROUGH**, finishing tool **FINISH**, thread tool **THREAD**, recessing tool **RECESS**, button tool **BUTTON**, groove turning tool **RECTURN**	**Type of turning tool**

Further information: "Tool orientation (TO) that is supported for the following types of turning tools (TYPE)", Page 307

Tool orientation (TO) that is supported for the following types of turning tools (TYPE)

TYPE	Supported TO with possible limitations	Non-supported TO	
ROUGH, FINISH	■ 1 ■ 7 ■ 2, only **XL** ■ 3, only **XL** ■ 5, only **XL** ■ 6, only **XL** ■ 8, only **ZL**	■ 4 ■ 9	
BUTTON	■ 1 ■ 7 ■ 2, only **XL** ■ 3, only **XL** ■ 5, only **XL** ■ 6, only **XL** ■ 8, only **ZL**	■ 4 ■ 9	
RECESS, RECTURN	■ 1 ■ 7 ■ 8 ■ 2 ■ 3, only **XL** ■ 5, only **XL**	■ 4 ■ 6 ■ 9	
THREAD	■ 1 ■ 7 ■ 8 ■ 2 ■ 3, only **XL** ■ 5, only **XL**	■ 4 ■ 6 ■ 9	

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

NOTICE

Danger of collision!

If the tool data **ZL** / **DZL** and **XL** / **DXL** deviate by more than ±2 mm from the real tool data, then there is a danger of collision.

- Enter the approximate tool data closer than ±2 mm
- Run the cycle carefully

The cycle depends on the optional machine parameter **CfgTTRectStylus** (no. 114300). Refer to your machine manual.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before you begin the cycle, you must run a **TOOL CALL** with the tool axis **Z**.
- If you define **YL** and **DYL** with a value outside of ±5 mm, the tool won't reach tool touch probe.
- The cycle does not support **SPB-INSERT** (angular offset). You must enter the value 0 in **SPB-INSERT**, otherwise the control will generate an error message.

Cycle parameters

- **Q340 Tool measurement mode (0-2)?**: usage of the measured values:
  0: The measured values are entered in **ZL** and **XL**. If values are already entered in the tool table, they will be overwritten. **DZL** and **DXL** will be reset to **0**. TL will not be changed
  1: The measured values **ZL** and **XL** are compared with the values from the tool table. These values will not be changed. The control then calculates the deviations of **ZL** and **XL**, and enters these in **DZL** and **DXL**. If the delta values are larger than the permissible wear or breakage tolerance, the control locks the tool (**TL** = Tool Locked). In addition, the deviation is also entered in Q parameters **Q115** and **Q116**
  2: The measured values **ZL** and **XL** as well as **DZL** and **DXL** are compared with the values from the tool table, but are not changed. If the values are larger than the permissible wear or breakage tolerance, the control locks the tool (**TL** = Tool Locked).
- **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

Example

```
6 TOOL CALL  12 Z
7 TCH PROBE 485  MEASURE LATHE
    TOOL
    Q340=+1     ;CHECK
    Q260=+100 ;CLEARANCE HEIGHT
```

10

VSC: camera-based setup control (option 136)

10.1 Camera-based Visual Setup Control (VSC) (option 136)

Fundamentals

In order to use Visual Setup Control you need the following components:

- Software: Option 136 Visual Setup Control (VSC)
- Hardware: Camera system from HEIDENHAIN

Application

Refer to your machine manual!
This function must be enabled and adapted by the machine tool builder.

Camera-based setup control (Option 136 Visual Setup Control) can be used to monitor the current setup situation before and during machining and compare it to a safe desired condition. After setup, simple cycles for automatic monitoring are available.

A camera system takes reference photos of the current working space. With Cycles **600 GLOBAL WORKING SPACE** or **601 LOCAL WORKING SPACE**, the control generates an image of the working space and compares the image with previously created reference images. These cycles can highlight irregularities in the working space. The operator decides whether the NC program is interrupted in the event of an error or continues to run.

Using VSC offers the following advantages:

- The control can recognize elements (e.g. tools, fixtures, etc.) that are in the working space once the program has started
- If you always want to clamp a workpiece at the same position (e.g. hole at top right), the control can check the setup situation
- For documentation purposes you can generate an image of the current workspace (e.g. of a clamping situation that is rarely used)

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

Terms

The following terms are used in connection with VSC:

Term	Explanation
Reference image	A reference image shows a situation in the working space that you regard as safe. Therefore only create reference images showing safe, non-hazardous situations.
Mid-value image	The control creates a mid-value image taking into account all reference images. The control compares new images with the mid-value image as part of evaluation.
Images with error	If you generate an image showing a poor situation (such as an incorrectly clamped workpiece), you can generate an "image of error" It is not advisable to highlight an error image as a reference image.
Monitoring area	Denotes an area that you highlight with the mouse. When evaluating new images, the control only refers to this area. Parts of images outside the monitoring area have no effect on the results of the monitoring process. Several monitoring areas can be defined. Monitoring areas are not linked to images.
Error	Area on an image containing a deviation from the desired position. Errors always refer to the image for which they were saved (image of error) or to the image most recently evaluated.
Monitoring phase	No further reference images are produced in the monitoring phase. You can use the cycle for automatic monitoring of your working space. In this phase, the control only issues a warning if it finds a deviation when comparing images.

Managing monitoring data

In **Manual operation** mode, you can manage images from Cycles **600** and **601**.

To manage the monitoring data, proceed as follows:

- Press the **CAMERA** soft key

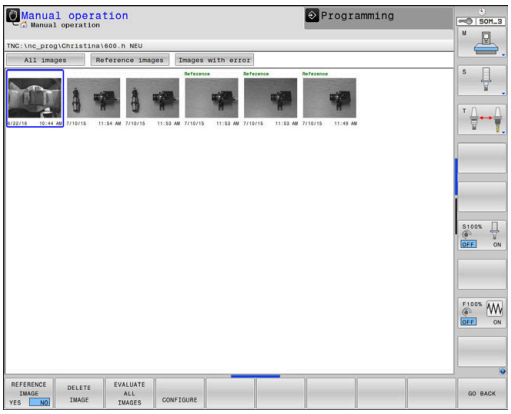
- Press the **MONITORING DATA MANAGEMENT** soft key
- The control shows a list of the NC programs being monitored.

- Press the **OPEN** soft key
- The control shows a list of monitoring points.
- Edit the desired data

Selecting data

You can select the buttons with the mouse. These interfaces make it easier to search and show results in a manageable way.

- **All images**: Display all images in this monitoring file
- **Reference images**: Only display reference images
- **Images with error**: Display all images where you have highlighted an error



Features of the monitoring data management

Soft key	Function
REFERENCE IMAGE YES NO	Mark selected image as a reference image A reference image shows a situation in the working space that you regard as safe. All reference images are taken into account for the evaluation. If you add or remove an image as a reference image, this has an effect on the results of image evaluation.
DELETE IMAGE	Delete image currently selected
EVALUATE ALL IMAGES	Carry out automatic image evaluation The control carries out an image evaluation according to the reference images and the monitoring areas.
CONFIGURE	Change monitoring area or highlight an error
GO BACK	Go back to the previous screen If you change the configuration, the control carries out an image evaluation.

Overview

The control provides two cycles you can use to define visual setup control in the **Programming** mode of operation:

TOUCH PROBE

- The soft-key row shows all available touch probe functions divided into groups.

MONITORING WITH CAMERA

- Press the **MONITORING WITH CAMERA** soft key.

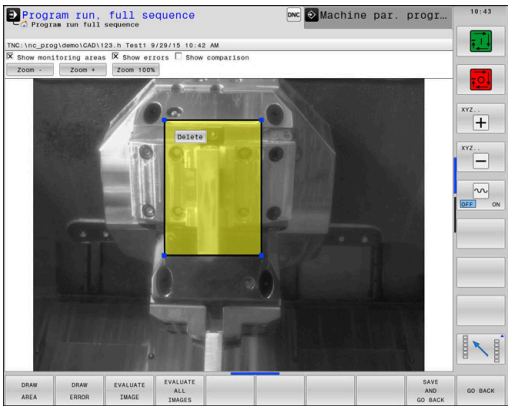
Soft key	Cycle	Page
600	GLOBAL WORKING SPACE (Cycle 600, ISO: G600, option 136) ■ Monitoring the working space of the machine tool ■ Generating an image of the current working space from a position determined by the machine tool builder ■ Image comparison with prepared reference images	319
601	LOCAL WORKING SPACE (Cycle 601, ISO: G601, option 136) ■ Monitoring the working space of the machine tool ■ Generating an image of the current working space from the position of the spindle at the point in time of the cycle call ■ Image comparison with prepared reference images	324

Configuration

You can change your settings regarding the monitoring area and errors at any time. When you press the **CONFIGURE** soft key, the soft-key display changes and you can change your settings.

Soft key	Function
CONFIGURE	Change settings for the monitoring area and sensitivity If you make a change in this menu, the results of the image evaluation may change.
DRAW AREA	Draw new monitoring area If you add a new monitoring area, or change/ delete monitoring areas already set, this will have an effect on the image evaluation. The same monitoring area applies to all reference images.
DRAW ERROR	Draw new error
EVALUATE IMAGE	The control checks if or how the new settings affect this image
EVALUATE ALL IMAGES	The control checks if or how the new settings affect all images
SHOW AREAS	The control shows all drawn monitoring areas
SHOW COMPARISON	The control compares the momentary image with the mean image
SAVE AND GO BACK	Save current image and return to the previous screen If you change the configuration, the control carries out an image evaluation.
GO BACK	Discard images and return to the previous screen

Defining the monitoring area



The monitoring area must be defined in the **Program Run, Full Sequence/Single Block** operating modes. The control will prompt you to define a monitoring area. The control will display this prompt on the screen after you have started the cycle for the first time in the **Program run, Full Sequence/Single Block** operating mode.

A monitoring area consists of one or more windows. If you define multiple windows, they may overlap. The control will only take these areas of the image into account. Errors outside the monitoring area will not be detected. The monitoring area is not linked to the images, but only to the monitoring file specified in **QS600**. A monitoring area always applies to all images of a monitoring file. A change to the monitoring area affects all images.

Drawing the monitoring area or error image:

Proceed as follows:

DRAW AREA

- ▶ Press the **DRAW AREA** or **DRAW ERROR** soft key
- ▶ Draw a rectangle around the area to be monitored
- > The control indicates the selected area with a frame.
- ▶ Use the available buttons to resize the image as desired

or

- ▶ You can define more windows by pressing the **DRAW AREA** or **DRAW ERROR** soft key and repeat the steps at the desired position
- ▶ Lock the defined area in place by double-clicking.
- > The area is protected from accidental shifting.

SAVE AND GO BACK

- ▶ Press the **SAVE AND GO BACK** soft key
- > The control will save the current image and return to the previous screen.

Deleting drawn areas

Proceed as follows:

- ▶ Select the area to be deleted
- > The control indicates the selected area with a frame.
- ▶ Press the **Delete** button

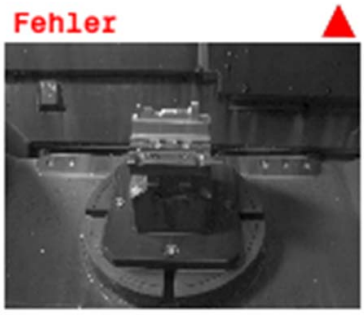
The status display at the top of the image will display information about the minimum number of reference images, the current number of reference images, and the current number of error images.

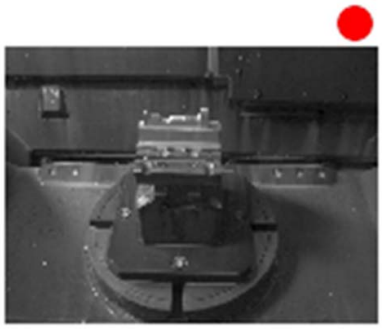
Result of the image evaluation

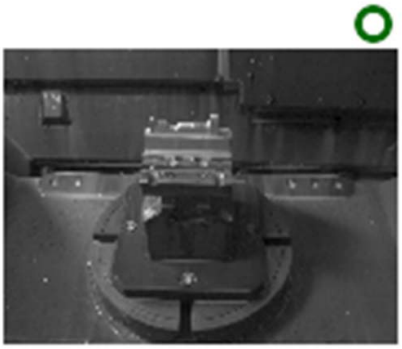
The result of the image evaluation depends on the monitoring area and the reference images. When evaluating all images, each image is evaluated according to the current configuration and the results are compared with the data last saved.

If you change the monitoring area or add/delete reference images, the images may be tagged with the following symbol:

- **Triangle:** You changed the monitoring area or the sensitivity. This has an impact on all of your reference images and the mean image, respectively. As a result of your change to the configuration, the control can no longer detect errors that had been saved for this image! The system has become less sensitive. If you would like to proceed, confirm the reduced sensitivity of the system. The new settings will be applied.
- **Solid circle:** You changed the monitoring area or the sensitivity. This has an impact on all of your reference images and the mean image, respectively. As a result of your change to the configuration, the control can now detect errors that it was unable to detect before. The system has become more sensitive. If you would like to proceed, confirm the increased sensitivity of the system. The new settings will be applied.
- **Empty circle:** No error message: All deviations saved in the image were detected. The system, therefore, has basically kept its previous sensitivity.



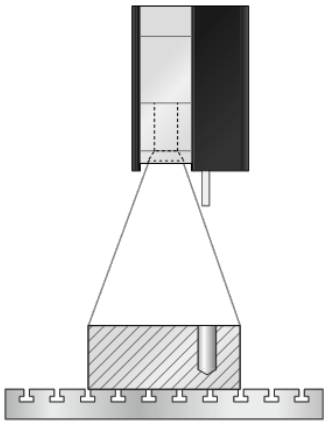




10.2 GLOBAL WORKING SPACE (Cycle 600, ISO: G600, option 136)

Application

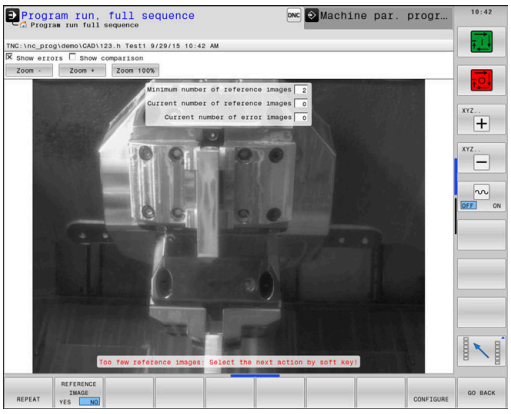
Refer to your machine manual!
This function must be enabled and adapted by the machine tool builder.



With Cycle **600** GLOBAL WORKING SPACE, you monitor the working space of your machine tool. The control will generate an image of the current working space from a position determined by your machine tool builder. Then, the control will match the image against previously generated reference images and enforce a program stop, if required. You can program this cycle to suit a particular application and specify one or several monitoring areas. Cycle **600** becomes effective as soon as it has been defined, and does not have to be called. Before you can use camera monitoring, you must first generate reference images and define a monitoring area.
Further information: "Creating reference images", Page 320
Further information: "Monitoring phase", Page 321

Creating reference images

Cycle run



1 The machine tool builder mounts the camera onto the main spindle. The main spindle moves to a position defined by the machine tool builder
2 After this position has been reached, the control will automatically open the camera cap
3 Once you run this cycle for the first time in **Program Run, Full Sequence/Single Block**, the control interrupts the current NC program and shows the image taken by the camera.
4 A message is displayed, indicating that there is no reference image to be evaluated
5 Press the **REFERENCE IMAGE YES** soft key
6 Then, the following message is displayed at the bottom of the screen: **Monitoring point not configured: Draw areas!**
7 Press the **CONFIGURE** soft key and define the monitoring area
Further information: "Defining the monitoring area", Page 317
8 These steps are repeated until the control has saved a sufficient number of reference images. The number of reference images can be specified in the cycle with parameter **Q617**
9 Conclude the procedure by pressing the **GO BACK** soft key. The control resumes program run
10 Finally, the control closes the camera cap
11 Press **NC Start** and run your NC program as usual

After you have defined the monitoring area, you can press the following soft keys:

GO BACK

- ▶ Press the **GO BACK** soft key
- > The control will save the current image and return to the program run screen. If you change the configuration, the control carries out an image evaluation.
Further information: "Result of the image evaluation", Page 318

REPEAT

or

- ▶ Press the **REPEAT** soft key
- > The control will save the current image and return to the program run screen. If you change the configuration, the control carries out an image evaluation.
Further information: "Result of the image evaluation", Page 318

REFERENCE IMAGE YES NO

or

- ▶ Press the **REFERENCE IMAGE** soft key
- > The status display at the top right displays the word **Reference**. You have marked the current image as the reference image. Since a reference image can never be an error image at the same time, the **IMAGE OF ERROR** soft key turns gray.

or

- Press the **IMAGE OF ERROR** soft key

> The status display at the top right displays the word "Error". You have marked the current image as the error image. Since an image of an error can never be a reference image at the same time, the **REFERENCE IMAGES** soft key turns gray.

or

- Press the **CONFIGURE** soft key

> The soft key row changes. You can now change your previously entered settings with respect to the monitoring area and the sensitivity. If you make a change in this menu, this may affect all of your images.
> **Further information:** "Configuration", Page 316

Programming and operating notes:

- As soon as the control has generated at least one reference image, images will be evaluated and errors displayed. If no errors are detected, the following message is displayed: **Too few reference images: Select the next action by soft key!** This message disappears once the number of reference images defined in parameter **Q617** has been reached.
- The control will generate a mean-value image, taking into account all reference images. During evaluation, new images are compared with the mean-value image, taking the variance into account. Once the specified number of reference images has been reached, the cycle will continue without interruption.

Monitoring phase

Cycle run: Monitoring phase

1. The machine tool builder mounts the camera onto the main spindle. The main spindle moves to a position defined by the machine tool builder
2. After this position has been reached, the control will automatically open the camera cap
3. The control will generate an image of the current condition
4. The control will then compare the images using the mean image and the variance image.
   Further information: "Fundamentals", Page 312
5. Depending on whether the control detects an "error" (deviation), the control can now force a program cancellation. If parameter **Q309**=1 is set, the control will display the image on the screen upon detection of an error. If parameter **Q309**=0 is set, no image will be displayed on the screen and the program will not be aborted.
6. Finally, the control closes the camera cap

Please note while programming!

Your machine must be prepared for camera-based monitoring!

NOTICE

Danger of collision!

Danger of camera contamination if the camera cap is open (set with parameter **Q613**). This could result in blurred photos, or the camera might be damaged.

- Close the camera cap before continuing the machining process

NOTICE

Danger of collision!

Danger of collision if the camera is positioned automatically. The camera and your machine may be damaged.

- Refer to the machine manual to learn about the point the camera is pre-positioned to by the control. Your machine tool builder specifies the coordinates which Cycle **600** positions to.

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

In addition to the Reference Image attribute, you can assign the Error Image attribute to your images. This assignment may impact image evaluation.

Remember the following:

- Never mark a reference image as an error image at the same time

If you change the monitoring area, this has an impact on all images.

- Ideally, you define the monitoring area only once, i.e. at the beginning, and then make no or only little changes to it

The number of reference images will affect the accuracy of the image evaluation. A high number of reference images improves the quality of the evaluation.

- Specify a reasonable number of reference images in parameter **Q617**. (Approx. value: 10 images)
- You can also create more reference images than you have specified in **Q617**

Cycle parameters

- **QS600** (string parameter) **Name of monitoring point?**: Enter the name of your monitoring file.
- **Q616 Feed rate for positioning?**: Feed rate the control uses to position the camera. The control moves the camera to a position defined by the machine tool builder.
  Input range: 0.001 to 99999.999
- **Q309 PGM stop if tolerance exceeded?**: (0/1)
  Specify whether the control stops the program if an error is detected.
  0: The NC program does not stop if an error is detected. Even if not all reference images have been generated yet, no stop is performed. This means that the generated image will not be displayed on the screen. Parameter **Q601** is written to even if **Q309**=0.
  1: The NC program will stop after detecting an error, and the generated image will be displayed on the screen. If not enough reference images have been generated yet, every new image will be displayed on the screen until the control has generated enough reference images. If an error is detected, the control will display a message.
- **Q617 Number of reference images?**: Number of reference images that are required by the control for monitoring.
  Input range: 0 to 200

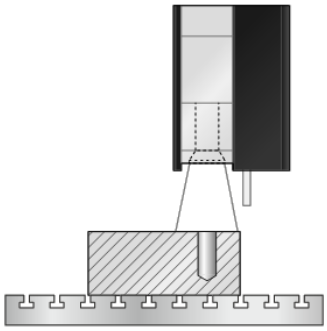
Example

```
4 TCH PROBE 600 GLOBAL WORKING SPACE
    QS600="OS";MONITORING POINT
    Q616=500  ;FEED RATE FOR POSITIONING
    Q309=1    ;PGM STOP TOLERANCE
    Q617=10   ;REFERENCE IMAGES
```

10.3 LOCAL WORKING SPACE (Cycle 601, ISO: G601, option 136)

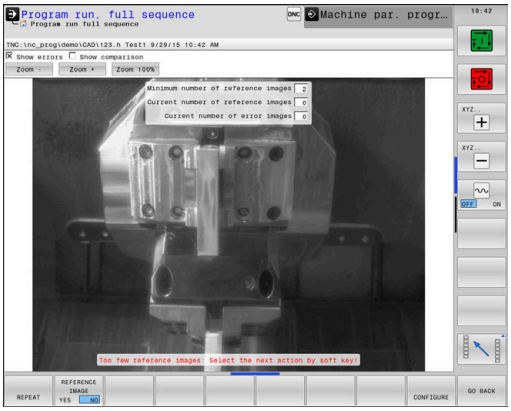
Application

Refer to your machine manual!
This function must be enabled and adapted by the machine tool builder.



With Cycle **601** LOCAL WORKING SPACE, you monitor the working space of your machine tool. The control will generate an image of the current working space from the position of the spindle at the moment the cycle is called. Then, the control will match the image against previously generated reference images and enforce a program stop, if required. You can program this cycle to suit a particular application and specify one or several monitoring areas. Cycle **601** becomes effective as soon as it has been defined, and does not have to be called. Before you can use camera monitoring, you must first generate reference images and define a monitoring area.
Further information: "Creating reference images", Page 324
Further information: "Monitoring phase", Page 326

Creating reference images



Cycle run

1 The machine tool builder mounts the camera onto the main spindle. The main spindle moves to the position programmed previously
2 The control automatically opens the camera cap
3 Once you run this cycle for the first time in **Program Run, Full Sequence/Single Block**, the control interrupts the current NC program and shows the image taken by the camera.
4 A message is displayed, indicating that there is no reference image to be evaluated
5 Press the **REFERENCE IMAGE YES** soft key
6 Then, the following message is displayed at the bottom of the screen: **Monitoring point not configured: Draw areas!**
7 Press the **CONFIGURE** soft key and define the monitoring area
Further information: "Defining the monitoring area", Page 317
8 These steps are repeated until the control has saved a sufficient number of reference images. The number of reference images can be specified in the cycle with parameter **Q617**
9 Conclude the procedure by pressing the **GO BACK** soft key. The control resumes program run
10 Finally, the control closes the camera cap
11 Press **NC Start** and run your NC program as usual

After you have defined the monitoring area, you can press the following soft keys:

- Press the **GO BACK** soft key
- The control will save the current image and return to the program run screen. If you change the configuration, the control carries out an image evaluation.
  Further information: "Result of the image evaluation", Page 318

or

- Press the **REPEAT** soft key
- The control will save the current image and return to the program run screen. If you change the configuration, the control carries out an image evaluation.
  Further information: "Result of the image evaluation", Page 318

or

- Press the **REFERENCE IMAGE** soft key
- The status display at the top right displays the word **Reference**. You have marked the current image as the reference image. Since a reference image can never be an error image at the same time, the **IMAGE OF ERROR** soft key turns gray.

or

- Press the **IMAGE OF ERROR** soft key
- The status display at the top right displays the word "Error". You have marked the current image as the error image. Since an image of an error can never be a reference image at the same time, the **REFERENCE IMAGES** soft key turns gray.

or

- Press the **CONFIGURE** soft key
- The soft key row changes. You can now change your previously entered settings with respect to the monitoring area and the sensitivity. If you make a change in this menu, this may affect all of your images. **Further information:** "Configuration", Page 316

Programming and operating notes:

- As soon as the control has generated at least one reference image, images will be evaluated and errors displayed. If no errors are detected, the following message is displayed: **Too few reference images: Select the next action by soft key!** This message disappears once the number of reference images defined in parameter **Q617** has been reached.
- The control will generate a mean-value image, taking into account all reference images. During evaluation, new images are compared with the mean-value image, taking the variance into account. Once the specified number of reference images has been reached, the cycle will continue without interruption.

Monitoring phase

The monitoring phase starts as soon as the control has generated enough reference images.

Cycle run: Monitoring phase

1 The machine tool builder mounts the camera onto the main spindle
2 The control automatically opens the camera cap
3 The control will generate an image of the current condition
4 The control will then compare the images using the mean image and the variance image.
5 Depending on whether the control detects an "error" (deviation), the control can now force a program cancellation. If parameter **Q309**=1 is set, the control will display the image on the screen upon detection of an error. If parameter **Q309**=0 is set, no image will be displayed on the screen and the program will not be aborted.
6 Depending on the setting of parameter **Q613**, the control either leaves the camera cap open or closes it.

Please note while programming!

Your machine must be prepared for camera-based monitoring!

NOTICE

Danger of collision!

Danger of camera contamination if the camera cap is open (set with parameter **Q613**). This could result in blurred images, or the camera might be damaged.

- Close the camera cap before continuing the machining process

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

In addition to the Reference Image attribute, you can assign the Error Image attribute to your images. This assignment may impact image evaluation.

Remember the following:

- Never mark a reference image as an error image at the same time

If you change the monitoring area, this has an impact on all images.

- Ideally, you define the monitoring area only once, i.e. at the beginning. and then make no or only little changes to it

The number of reference images will affect the accuracy of the image evaluation. A high number of reference images improves the quality of the evaluation.

- Specify a reasonable number of reference images in parameter **Q617**. (Approx. value: 10 images)
- You can also create more reference images than you have specified in **Q617**

Cycle parameters

- **QS600** (string parameter) **Name of monitoring point?**: Enter the name of your monitoring file.
- **Q309 PGM stop if tolerance exceeded?**: (0/1) Specify whether the control stops the program if an error is detected.
  0: The NC program does not stop if an error is detected. Even if not all reference images have been generated yet, no stop is performed. This means that the generated image will not be displayed on the screen. Parameter **Q601** is written to even if **Q309**=0.
  1: The NC program will stop after detecting an error, and the generated image will be displayed on the screen. If not enough reference images have been generated yet, every new image will be displayed on the screen until the control has generated enough reference images. If an error is detected, the control will display a message.
- **Q613 Keep camera cap open?**: (0/1) Specify whether the control should keep the camera cap open after monitoring.
  0: The control closes the camera cap after running Cycle **601**.
  1: The control keeps the camera cap open after running Cycle **601**. This function is only recommended if you would like to generate yet another image of the working space in a different position after the first call of Cycle **601**. In order to do so, program the new position in a linear block and then call Cycle **601** with a new monitoring point. Program **Q613**=0 before continuing with the cutting operation!
- **Q617 Number of reference images?**: Number of reference images that are required by the control for monitoring.
  Input range: 0 to 200

Example

```
4 TCH PROBE 601 LOCAL WORKING SPACE
   QS600="OS";MONITORING POINT
   Q309=+1    ;PGM STOP TOLERANCE
   Q613=0     ;KEEP CAMERA OPEN
   Q617=10    ;REFERENCE IMAGES
```

10.4 Possible queries

The VSC cycles enter a value into the parameter **Q601**.

The following values are possible:

- **Q601** = 1: no error
- **Q601** = 2: error
- **Q601** = 3: you have yet not defined a monitoring area or you did not save enough reference images
- **Q601** = 10: internal error (no signal, camera error, etc.)

You can use the parameter **Q601** for internal queries.

Further information: for If-then decisions, refer to the Conversational Programming User's Manual

The following is a programming example for a query:

0 BEGIN PGM 5MM	
1 BLK FORM CYLINDER Z R42 L150	Workpiece blank definition: cylinder
2 FUNCTION MODE MILL	Activate milling mode
3 TCH PROBE 601 LOCAL WORKING SPACE	Define Cycle 600
QS600 = OS ;MONITORING POINT	
Q309 = +0 ;PGM STOP TOLERANCE	
Q613 = +0 ;KEEP CAMERA OPEN	
Q617 = 10 ;REFERENCE IMAGES	
4 FN 9: IF Q601 EQU 1 GOTO LBL 20	If parameter Q601 = 1, then jump to LBL 20
5 FN 9: IF Q601 EQU 2 GOTO LBL 21	If parameter Q601 = 2, then jump to LBL 21
6 FN 9: IF Q601 EQU 3 GOTO LBL 22	If parameter Q601 = 3, then jump to LBL 22
7 FN 9: IF Q601 EQU 10 GOTO LBL 23	If parameter Q601 = 10, then jump to LBL 23
8 TOOL CALL "ZAHRADFRAESER_D75"	Call the tool
9 L X+... Y+... R0 FMAX	Program the machining operation
...	
...	
...	
57 LBL 21	Definition of LBL 21
58 STOP	Program stop, the operator can check the condition of the workspace
59 LBL 0	
60 END PGM 5MM	

11

Cycles: Special Functions

11.1 Fundamentals

Overview

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

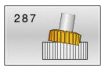
CYCL DEF

- Press the **CYCL DEF** key

SPECIAL CYCLES

- Press the **SPECIAL CYCLES** soft key

Soft key	Cycle	Page
9	**9 DWELL TIME** ■ Delay execution by the programmed dwell time	**Further information:** User's Manual Cycle Programming
12 PGM CALL	**12 PGM CALL** ■ Call any NC program	**Further information:** User's Manual Cycle Programming
13	**13 ORIENTATION** ■ Rotate spindle to a specific angle	334
32 T	**32 TOLERANCE** ■ Program the permissible contour deviation for jerk-free machining operations	**Further information:** User's Manual Cycle Programming
291	**291 COUPLG.TURNG.INTERP.** ■ Coupling of the tool spindle with the positions of the linear axes ■ Or, rescind the spindle coupling	**Further information:** User's Manual Cycle Programming
292	**292 CONTOUR.TURNG.INTRP.** ■ Coupling of the tool spindle with the positions of the linear axes ■ Create certain rotationally symmetric contours in the active working plane ■ Possible with tilted machining plane	**Further information:** User's Manual Cycle Programming
225 ABC	**225 ENGRAVING** ■ Engrave texts on a plane surface ■ Arranged in a straight line or along a circular arc	**Further information:** User's Manual Cycle Programming
232	**232 FACE MILLING** ■ Face mill a level surface in multiple infeeds ■ Selection of the milling plan	**Further information:** User's Manual Cycle Programming
285	**285 DEFINE GEAR** ■ Define the geometry of the gear wheel	**Further information:** User's Manual Cycle Programming
286	**286 GEAR HOBBING** ■ Definition of the tool data ■ Selection of the machining strategy and side ■ Possibility of using the entire cutting edge	**Further information:** User's Manual Cycle Programming

Soft key	Cycle	Page
	**287 GEAR SKIVING** ■ Definition of the tool data ■ Selection of the machining side ■ Definition of the first and last infeed ■ Definition of the number of cuts	**Further information:** User's Manual Cycle Programming
	238 MEASURE MACHINE STATUS ■ Determine the current machine status or test the measuring sequence	**Further information:** User's Manual Cycle Programming
	239 ASCERTAIN THE LOAD ■ Selection for a weighing run ■ Reset the load-dependent feedforward and controller parameters	**Further information:** User's Manual Cycle Programming
	18 THREAD CUTTING ■ With controlled spindle ■ Spindle stops at the bottom of the hole	**Further information:** User's Manual Cycle Programming

11.2 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.

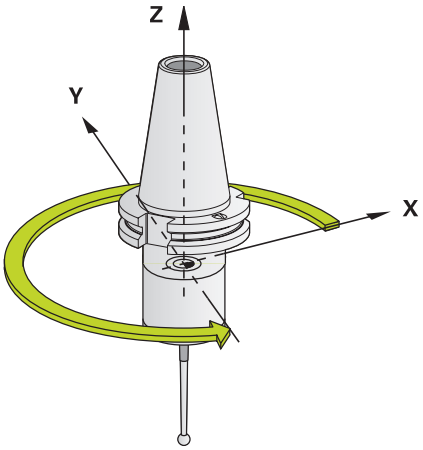
The control can control the main machine tool spindle and rotate it to a given angular position.

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.



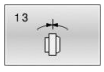
Example

```
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**, **FUNCTION MODE TURN**, and **FUNCTION DRESS** 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°

12

Tables of Cycles

12.1 Table of cycles

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

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

Touch probe cycles

Cycle number	Cycle name	DEF active	CALL active	Page
0	REF. PLANE	■		172
1	POLAR PRESET	■		173
3	MEASURING	■		215
4	MEASURING IN 3-D	■		217
30	CALIBRATE TT	■		287
31	CAL. TOOL LENGTH	■		290
32	CAL. TOOL RADIUS	■		294
33	MEASURE TOOL	■		298
400	BASIC ROTATION	■		84
401	ROT OF 2 HOLES	■		87
402	ROT OF 2 STUDS	■		91
403	ROT IN ROTARY AXIS	■		96
404	SET BASIC ROTATION	■		105
405	ROT IN C AXIS	■		101
408	SLOT CENTER PRESET	■		154
409	RIDGE CENTER PRESET	■		158
410	PRESET INSIDE RECTAN	■		111
411	PRESET OUTS. RECTAN	■		115
412	PRESET INSIDE CIRCLE	■		119
413	PRESET OUTS. CIRCLE	■		124
414	PRESET OUTS. CORNER	■		129
415	PRESET INSIDE CORNER	■		134
416	PRESET CIRCLE CENTER	■		139
417	PRESET IN TS AXIS	■		144
418	PRESET FROM 4 HOLES	■		146
419	PRESET IN ONE AXIS	■		151
420	MEASURE ANGLE	■		174
421	MEASURE HOLE	■		177
422	MEAS. CIRCLE OUTSIDE	■		182

Cycle number	Cycle name	DEF active	CALL active	Page
423	MEAS. RECTAN. INSIDE	■		187
424	MEAS. RECTAN. OUTS.	■		191
425	MEASURE INSIDE WIDTH	■		194
426	MEASURE RIDGE WIDTH	■		197
427	MEASURE COORDINATE	■		200
430	MEAS. BOLT HOLE CIRC	■		203
431	MEASURE PLANE	■		206
441	FAST PROBING	■		224
444	PROBING IN 3-D	■		219
450	SAVE KINEMATICS	■		246
451	MEASURE KINEMATICS	■		249
452	PRESET COMPENSATION	■		263
453	KINEMATICS GRID	■		274
460	CALIBRATION OF TS ON A SPHERE	■		236
461	TS CALIBRATION OF TOOL LENGTH	■		228
462	CALIBRATION OF A TS IN A RING	■		230
463	TS CALIBRATION ON STUD	■		233
480	CALIBRATE TT	■		287
481	CAL. TOOL LENGTH	■		290
482	CAL. TOOL RADIUS	■		294
483	MEASURE TOOL	■		298
484	CALIBRATE IR TT	■		302
485	MEASURE LATHE TOOL	■		305
600	GLOBAL WORKING SPACE	■		319
601	LOCAL WORKING SPACE	■		324
1410	PROBING ON EDGE	■		71
1411	PROBING TWO CIRCLES	■		77
1420	PROBING IN PLANE	■		66

Machining cycles

Cycle number	Cycle name	DEF active	CALL active	Page
13	ORIENTATION	■		334

Index

3

3D Touch Probes..... 42

A

About this manual..... 22
Automatic presetting
- Bolt circle..... 139
- Center of 4 holes..... 146
- Center of slot..... 154
- Circular pocket (hole)..... 119
- Circular stud..... 124
- Fundamentals..... 108
- Preset inside corner..... 134
- Preset outside corner..... 129
- rectangular pocket..... 111
- rectangular stud..... 115
- Ridge center..... 158
- Single axis..... 151
- Touch probe axis..... 144

B

Basic rotation..... 84
- Rot in rotary axis..... 96
- Rot of 2 holes..... 87
- Rot of 2 studs..... 91
- Set directly..... 105

C

Calibration cycles..... 226
- Calibration of TS..... 236
- Calibration of TS in a ring..... 230
- Calibration on stud..... 233
- TS Calibration of tool length. 228

Camera-based check fundamentals..... 312
Camera-based monitoring
- Global working space..... 319
- Local working space..... 324

Classification of results..... 169

D

Determine inclined workpiece position
- Probing in plane..... 66
- Probing on edge..... 71
- Probing two circles..... 77
- Set basic rotation..... 105
- Touch probe cycles 14xx fundamentals..... 57
- Touch probe cycles 4xx fundamentals..... 83

Determining inclined workpiece position
- Basic rotation..... 84
- Basic rotation over two holes. 87
- Basic rotation over two studs. 91
- Compensate basic rotation over a rotary axis..... 96
- Rotation of C axis..... 101

F

Fast probing..... 224
Feature content level..... 28

G

GLOBAL DEF..... 49

I

Inclined workpiece position checking
- Fundamentals..... 166
- Measure angle..... 174
- Measure bolt hole circle..... 203
- Measure circle..... 182
- Measure coordinate..... 200
- Measure hole..... 177
- Measure plane..... 206
- Measure rectangular pocket 187
- Measure rectangular stud.... 191
- Measure ridge from outside 197
- Measure slot width..... 194
- Polar preset..... 173
- Reference plane..... 172

K

Kinematic measurement
- Accuracy..... 254
- Backlash..... 256
- Prerequisites..... 244

Kinematics measurement
- Fundamentals..... 242
- Hirth coupling..... 252
- Kinematics grid..... 274
- Measure kinematics..... 249
- Preset compensation..... 263
- Save kinematics..... 246

KinematicsOpt..... 242

M

Measure circle from inside..... 177
Measure circle from outside.... 182
Measure inside width..... 194
Measure ridge width..... 197
Measure slot width..... 194
Measuring
- Angle..... 174
- Bolt hole circle..... 203
- Circle from outside..... 182
- Coordinate..... 200
- Holes..... 177
- Inside width..... 194
- Plane..... 206
- Rectangle from inside..... 187
- Rectangle from outside..... 191
- Ridge width..... 197
- using Cycle 3..... 215

Measuring in 3-D..... 217

O

Option..... 25

P

Positioning logic..... 48
Probing feed rate..... 46
Probing in 3-D..... 219

R

Recording measurement results.... 167
Rectangular pocket
- Measuring..... 187

Rectangular stud
- Measuring..... 191

Reference images..... 313

S

Software option..... 25
Spindle orientation..... 334

T

Table of cycles..... 336
- Touch probe cycles..... 336

Tolerance monitoring..... 169
Tool compensation..... 170
Tool measurement
- Fundamentals..... 282
- IR TT calibration..... 302
- Lathe tool, measurement..... 305
- Machine parameters..... 284
- Measure tool length and radius..... 298
- Tool length..... 290
- Tool radius..... 294
- TT calibration..... 287

Tool monitoring..... 170
Tool table..... 286
Touch probe cycles 14xx
- Evaluation of tolerances..... 64
- Fundamentals..... 57
- Probing in plane..... 66
- Probing on edge..... 71
- Probing two circles..... 77
- Semi-automatic mode..... 59
- Transfer an actual position..... 65

Touch probe data..... 53
Touch-probe table..... 52

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 32-5061
E-mail: info@heidenhain.de

Technical support FAX +49 8669 32-1000
Measuring systems ☏ +49 8669 31-3104
E-mail: service.ms-support@heidenhain.de
NC support ☏ +49 8669 31-3101
E-mail: service.nc-support@heidenhain.de
NC programming ☏ +49 8669 31-3103
E-mail: service.nc-pgm@heidenhain.de
PLC programming ☏ +49 8669 31-3102
E-mail: service.plc@heidenhain.de
APP programming ☏ +49 8669 31-3106
E-mail: service.app@heidenhain.de

www.heidenhain.de

www.klartext-portal.com
The Information Site for HEIDENHAIN Controls

Klartext App
The Klartext on Your Mobile Device

Google Play Store

Apple App Store

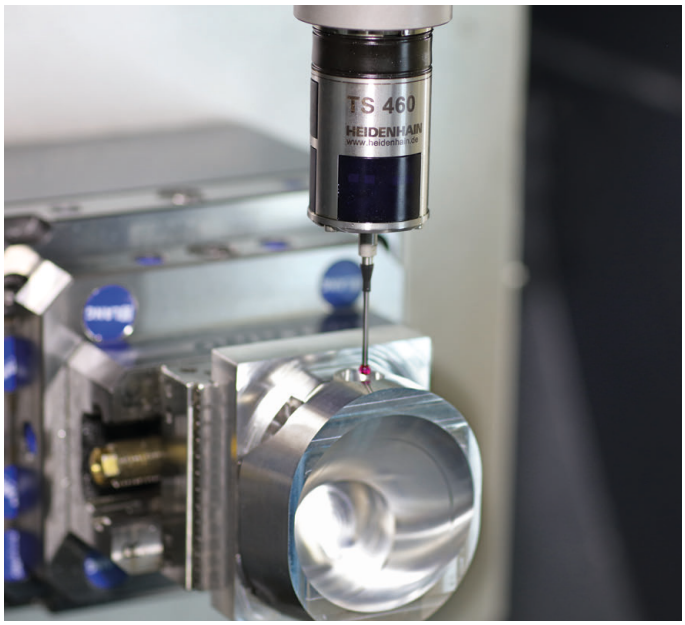
Touch probes from HEIDENHAIN

help you reduce non-productive time and improve the dimensional accuracy of the finished workpieces.

Workpiece touch probes

TS 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

Original document
1303409-20 · Ver00 · SW11 · 01/2021 · H · Printed in Germany