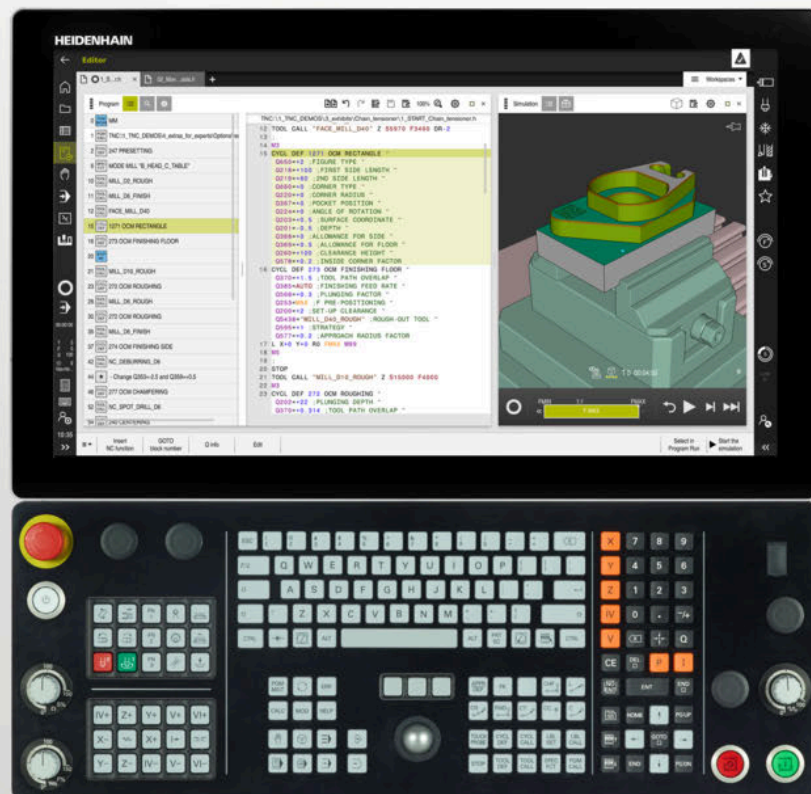




HEIDENHAIN



TNC7

User's Manual
Measuring Cycles for
Workpieces and Tools

NC Software
817620-16
817621-16
817625-16

English (en)
01/2022

Contents

1	About the User's Manual.....	19
2	About the product.....	25
3	Working with Touch Probe Cycles.....	41
4	Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment.....	55
5	Touch Probe Cycles: Automatic Preset Measurement.....	119
6	Touch Probe Cycles: Automatic Workpiece Inspection.....	203
7	Touch Probe Cycles: Special Functions.....	261
8	Touch Probe Cycles: Calibration.....	279
9	Touch Probe Cycles: Automatic Kinematics Measurement.....	297
10	Touch Probe Cycles: Automatic Tool Measurement.....	339
11	Special Cycles.....	365

1	About the User's Manual.....	19
1.1	Target group: Users.....	20
1.2	Available user documentation.....	21
1.3	Types of notes used.....	22
1.4	Notes on using NC programs.....	23
1.5	Contact to the editorial staff.....	23

2	About the product.....	25
2.1	The TNC7.....	26
2.2	Proper and intended operation.....	26
2.3	Intended place of operation.....	27
2.4	Safety precautions.....	28
2.5	Software.....	30
2.5.1	Software options.....	31
2.5.2	Feature Content Level.....	37
2.5.3	Information on licensing and use.....	37
2.6	Comparison between TNC 640 and TNC7.....	38

3	Working with Touch Probe Cycles.....	41
3.1	General information about touch probe cycles.....	42
3.1.1	Method of function.....	42
3.1.2	Notes.....	43
3.1.3	Touch probe cycles in the Manual Operation and Electronic Handwheel modes.....	43
3.1.4	Touch probe cycles for automatic operation.....	43
3.1.5	Available cycle groups.....	47
3.2	Before you start working with touch probe cycles!.....	50
3.2.1	General information.....	50
3.2.2	Executing touch probe cycles.....	50
3.3	Program defaults for cycles.....	51
3.3.1	Entering GLOBAL DEF definitions.....	51
3.3.2	Using GLOBAL DEF information.....	52
3.3.3	Global data valid everywhere.....	53
3.3.4	Global data for probing functions.....	54

4	Touch Probe Cycles: Automatic Measurement of Workpiece Misalignment.....	55
4.1	Overview.....	56
4.2	Fundamentals of touch probe cycles 14xx.....	57
4.2.1	Functionalities common to touch probe cycles 14xx for measuring rotations.....	57
4.2.2	Semi-automatic mode.....	58
4.2.3	Evaluation of tolerances.....	64
4.2.4	Transferring the actual position.....	66
4.3	Cycle 1420 PROBING IN PLANE.....	67
4.3.1	Cycle parameters.....	70
4.4	Cycle 1410 PROBING ON EDGE.....	73
4.4.1	Cycle parameters.....	77
4.5	Cycle 1411 PROBING TWO CIRCLES.....	80
4.5.1	Cycle parameters.....	84
4.6	Cycle 1412 INCLINED EDGE PROBING.....	88
4.6.1	Cycle parameters.....	91
4.7	Touch probe cycles 4xx: fundamentals.....	94
4.7.1	Characteristics common to all touch probe cycles for measuring workpiece misalignment.....	94
4.8	Cycle 400 BASIC ROTATION.....	95
4.8.1	Cycle parameters.....	96
4.9	Cycle 401 ROT OF 2 HOLES.....	97
4.9.1	Cycle parameters.....	99
4.10	Cycle 402 ROT OF 2 STUDS.....	101
4.10.1	Cycle parameters.....	103
4.11	Cycle 403 ROT IN ROTARY AXIS.....	106
4.11.1	Cycle parameters.....	108
4.12	Cycle 405 ROT IN C AXIS.....	112
4.12.1	Cycle parameters.....	115
4.13	Cycle 404 SET BASIC ROTATION.....	117
4.13.1	Cycle parameters.....	117
4.14	Example: Determining a basic rotation from two holes.....	118

5	Touch Probe Cycles: Automatic Preset Measurement.....	119
5.1	Overview.....	120
5.2	Fundamentals of touch probe cycles 14xx for presetting.....	121
5.2.1	Characteristics common to all touch probe cycles 14xx for preset setting.....	121
5.3	Cycle 1400 POSITION PROBING.....	121
5.3.1	Cycle parameters.....	123
5.4	Cycle 1401 CIRCLE PROBING.....	125
5.4.1	Cycle parameters.....	127
5.5	Cycle 1402 SPHERE PROBING.....	129
5.5.1	Cycle parameters.....	132
5.6	Fundamentals of touch probe cycles 4xx for preset setting.....	134
5.6.1	Characteristics common to all touch probe cycles 4xx for preset setting.....	134
5.7	Cycle 410 PRESET INSIDE RECTAN.....	136
5.7.1	Cycle parameters.....	138
5.8	Cycle 411 PRESET OUTS. RECTAN.....	141
5.8.1	Cycle parameters.....	143
5.9	Cycle 412 PRESET INSIDE CIRCLE.....	147
5.9.1	Cycle parameters.....	149
5.10	Cycle 413 PRESET OUTS. CIRCLE.....	153
5.10.1	Cycle parameters.....	155
5.11	Cycle 414 PRESET OUTS. CORNER.....	159
5.11.1	Cycle parameters.....	161
5.12	Cycle 415 PRESET INSIDE CORNER.....	164
5.12.1	Cycle parameters.....	167
5.13	Cycle 416 PRESET CIRCLE CENTER.....	170
5.13.1	Cycle parameters.....	173
5.14	Cycle 417 PRESET IN TS AXIS.....	177
5.14.1	Cycle parameters.....	179
5.15	Cycle 418 PRESET FROM 4 HOLES.....	181
5.15.1	Cycle parameters.....	183
5.16	Cycle 419 PRESET IN ONE AXIS.....	186
5.16.1	Cycle parameters.....	187

5.17	Cycle 408 SLOT CENTER PRESET.....	189
5.17.1	Cycle parameters.....	191
5.18	Cycle 409 RIDGE CENTER PRESET.....	194
5.18.1	Cycle parameters.....	196
5.19	Example: Presetting at center of a circular segment and on top surface of workpiece.....	199
5.20	Example: Presetting on top surface of workpiece and at center of a bolt hole circle.....	200

6	Touch Probe Cycles: Automatic Workpiece Inspection.....	203
6.1	Fundamentals.....	204
6.1.1	Overview.....	204
6.1.2	Recording the results of measurement.....	205
6.1.3	Measurement results in Q parameters.....	207
6.1.4	Classification of results.....	207
6.1.5	Tolerance monitoring.....	207
6.1.6	Tool monitoring.....	207
6.1.7	Reference system for measurement results.....	208
6.2	Cycle 0 REF. PLANE.....	209
6.2.1	Cycle parameters.....	210
6.3	Cycle 1 POLAR PRESET.....	210
6.3.1	Cycle parameters.....	211
6.4	Cycle 420 MEASURE ANGLE.....	212
6.4.1	Cycle parameters.....	213
6.5	Cycle 421 MEASURE HOLE.....	215
6.5.1	Cycle parameters.....	217
6.6	Cycle 422 MEAS. CIRCLE OUTSIDE.....	221
6.6.1	Cycle parameters.....	223
6.7	Cycle 423 MEAS. RECTAN. INSIDE.....	227
6.7.1	Cycle parameters.....	229
6.8	Cycle 424 MEAS. RECTAN. OUTS.....	231
6.8.1	Cycle parameters.....	233
6.9	Cycle 425 MEASURE INSIDE WIDTH.....	236
6.9.1	Cycle parameters.....	237
6.10	Cycle 426 MEASURE RIDGE WIDTH.....	240
6.10.1	Cycle parameters.....	241
6.11	Cycle 427 MEASURE COORDINATE.....	243
6.11.1	Cycle parameters.....	245
6.12	Cycle 430 MEAS. BOLT HOLE CIRC.....	248
6.12.1	Cycle parameters.....	250
6.13	Cycle 431 MEASURE PLANE.....	253
6.13.1	Cycle parameters.....	255

6.14	Programming Examples.....	257
6.14.1	Example: Measuring and reworking a rectangular stud.....	257
6.14.2	Example: Measuring a rectangular pocket and recording the results.....	259

7	Touch Probe Cycles: Special Functions.....	261
7.1	Fundamentals.....	262
7.1.1	Overview.....	262
7.2	Cycle 3 MEASURING.....	263
7.2.1	Cycle parameters.....	264
7.3	Cycle 4 MEASURING IN 3-D.....	265
7.3.1	Cycle parameters.....	267
7.4	Cycle 444 PROBING IN 3-D.....	268
7.4.1	Cycle parameters.....	272
7.5	Cycle 441 FAST PROBING.....	274
7.5.1	Cycle parameters.....	275
7.6	Cycle 1493 EXTRUSION PROBING.....	276
7.6.1	Cycle parameters.....	278

8	Touch Probe Cycles: Calibration.....	279
8.1	Fundamentals.....	280
8.1.1	Overview.....	280
8.1.2	Calibrating a touch trigger probe.....	281
8.1.3	Displaying calibration values.....	281
8.2	Cycle 461 TS CALIBRATION OF TOOL LENGTH.....	282
8.2.1	Cycle parameters.....	283
8.3	Cycle 462 CALIBRATION OF A TS IN A RING.....	284
8.3.1	Cycle parameters.....	286
8.4	Cycle 463 TS CALIBRATION ON STUD.....	287
8.4.1	Cycle parameters.....	289
8.5	Cycle 460 CALIBRATION OF TS ON A SPHERE (option 17).....	290
8.5.1	Cycle parameters.....	294

9	Touch Probe Cycles: Automatic Kinematics Measurement.....	297
9.1	Fundamentals (option 48).....	298
9.1.1	Overview.....	298
9.1.2	Fundamentals.....	299
9.1.3	Requirements.....	300
9.1.4	Notes.....	301
9.2	Cycle 450 SAVE KINEMATICS (option 48).....	302
9.2.1	Cycle parameters.....	304
9.2.2	Log function.....	305
9.3	Cycle 451 MEASURE KINEMATICS (option 48).....	305
9.3.1	Positioning direction.....	307
9.3.2	Machines with Hirth-coupled axes.....	308
9.3.3	Example calculation of measuring positions for an A axis:.....	308
9.3.4	Choice of number of measuring points.....	309
9.3.5	Choice of the calibration sphere position on the machine table.....	309
9.3.6	Notes on various calibration methods.....	310
9.3.7	Notes on the accuracy.....	311
9.3.8	Notes on various calibration methods.....	312
9.3.9	Backlash.....	312
9.3.10	Notes.....	313
9.3.11	Cycle parameters.....	314
9.3.12	Various modes (Q406).....	318
9.3.13	Log function.....	320
9.4	Cycle 452 PRESET COMPENSATION (option 48).....	320
9.4.1	Cycle parameters.....	324
9.4.2	Adjustment of interchangeable heads.....	327
9.4.3	Drift compensation.....	329
9.4.4	Log function.....	331
9.5	Cycle 453 KINEMATICS GRID (option 48), (option 52).....	331
9.5.1	Various modes (Q406).....	333
9.5.2	Choice of the calibration sphere position on the machine table.....	333
9.5.3	Notes.....	333
9.5.4	Cycle parameters.....	335
9.5.5	Log function.....	337

10 Touch Probe Cycles: Automatic Tool Measurement.....	339
10.1 Fundamentals.....	340
10.1.1 Overview.....	340
10.1.2 Differences between Cycles 30 to 33 and Cycles 480 to 483.....	341
10.1.3 Setting machine parameters.....	341
10.1.4 Entries in the tool table for milling and turning tools.....	343
10.2 Cycle 30 or 480 CALIBRATE TT.....	344
10.2.1 Cycle parameters.....	346
10.3 Cycle 31 or 481 CAL. TOOL LENGTH.....	346
10.3.1 Cycle parameters.....	348
10.4 Cycle 32 or 482 CAL. TOOL RADIUS.....	349
10.4.1 Cycle parameters.....	351
10.5 Cycle 33 or 483 MEASURE TOOL.....	353
10.5.1 Cycle parameters.....	355
10.6 Cycle 484 CALIBRATE IR TT.....	356
10.6.1 Cycle parameters.....	359
10.7 Cycle 485 MEASURE LATHE TOOL (option 50).....	360
10.7.1 Cycle parameters.....	364

11 Special Cycles.....	365
11.1 Fundamentals.....	366
11.1.1 Overview.....	366
11.2 Cycle 13 ORIENTATION.....	367
11.2.1 Cycle parameters.....	368

1

**About the
User's Manual**

1.1 Target group: Users

A user is anyone who uses the control to perform at least one of the following tasks:

- Operating the machine
 - Setting up tools
 - Setting up workpieces
 - Machining workpieces
 - Eliminating possible errors during program run
- Creating and testing NC programs
 - Creating NC programs at the control or externally using a CAM system
 - Using the Simulation mode to test the NC programs
 - Eliminating possible errors during program test

The depth of information in the User's Manual results in the following qualification requirements on the user:

- Basic technical understanding, e.g. ability to read technical drawings and spatial imagination
- Basic knowledge in the field of metal cutting, e.g. meaning of material-specific parameters
- Safety instructions, e.g. possible dangers and their avoidance
- Training on the machine, e.g. axis directions and machine configuration



HEIDENHAIN offers separate information products for other target groups:

- Leaflets and overview of the product program for potential buyers
- Service Manual for service technicians
- Technical Manual for machine manufacturers

Additionally, HEIDENHAIN provides users and lateral entrants with a wide range of training opportunities in the field of NC programming

HEIDENHAIN training portal

In line with the target group, this User's Manual only contains information on the operation and use of the control. The information products for other target groups contain information on further product life phases.

1.2 Available user documentation

User's Manual

HEIDENHAIN refers to this information product as User's Manual, regardless of the output or transport medium. Well-known designations with the same meaning include operator's manual and operating instructions.

The User's Manual for the control is available in the variants below:

- As a printed version, sub-divided into the modules below:
 - The **Setting up and running** User's Manual contains all information needed for setting up the machine and for running NC programs.
ID: 1358774-xx
 - The **Programming and testing** User's Manual contains all information needed for creating and testing NC programs. Touch probe and machining cycles are not included.
ID for Klartext programming: 1358773-xx
 - The **machining cycles** User's Manual contains all functions of the machining cycles.
ID: 1358775-xx
 - The **Measuring cycles for Workpieces and Tools** User's Manual contains all functions of the touch probe cycles.
ID: 1358777-xx

- As PDF files, sub-divided according to the printed versions or as complete PDF, containing all modules

TNCguide

- As HTML file for use as an integrated product help **TNCguide** directly on the control

TNCguide

The User's Manual supports you in the safe handling of the control according to its intended use.

Further information: "Proper and intended operation", Page 26

Further information products for users

The following information products are available to you as users:

- **Overview of new and modified software functions** informs you about the innovations of specific software versions.
TNCguide
- **HEIDENHAIN brochures** inform you about products and services by HEIDENHAIN, e.g. software options of the control.
HEIDENHAIN brochures
- The **NC Solutions** database offers solutions for frequently occurring tasks.
HEIDENHAIN NC solutions

1.3 Types of notes used

Safety precautions

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

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

⚠ DANGER
Danger indicates hazards for persons. If you do not follow the avoidance instructions, the hazard will result in death or severe injury .
⚠ WARNING
Warning indicates hazards for persons. If you do not follow the avoidance instructions, the hazard could result in death or serious injury .
⚠ CAUTION
Caution indicates hazards for persons. If you do not follow the avoidance instructions, the hazard could result in minor or moderate injury .
NOTICE
Notice indicates danger to material or data. If you do not follow the avoidance instructions, the hazard could result in property damage .

Sequence of information in precautionary statements

All precautionary statements contain 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 manufacturer. This symbol also indicates machine-dependent functions. Possible hazards for the operator and the machine are described in the machine manual.



The book symbol represents a **cross reference** to external documentation, e.g. the documentation of your machine manufacturer 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

1.4 Notes on using NC programs

NC programs contained in this User's Manual are suggestions for solutions. The NC programs or individual NC blocks must be adapted before being used on a machine.

Change the following contents as needed:

- Tools
- Cutting parameters
- Feed rates
- Clearance height or safe position
- Machine-specific positions, e.g. with **M91**
- Paths of program calls

Some NC programs depend on the machine kinematics. Adapt these NC programs to your machine kinematics before the first test run.

In addition, test the NC programs using the simulation before the actual program run.



With a program test you determine whether the NC programs can be used with the available software options, the active machine kinematics and the current machine configuration.

1.5 Contact to the editorial staff**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

2

About the product

2.1 The TNC7

Every HEIDENHAIN control supports you with dialog-guided programming and finely detailed simulation. The TNC7 additionally offers you graphical or form-based programming to reach the desired result safe and sound.

Software options and optional hardware extensions can be used for flexibly increasing the range of functions and ease of use.

Such an extension provides e.g. the chance to perform turning and grinding in addition to milling and drilling processes.

Further information: User's Manual for Programming and Testing

The ease of use increases e.g. when using touch probes, handwheels or a 3D mouse.

Further information: User's Manual for Setup and Program Run

Definitions

Abbreviation	Definition
TNC	TNC is derived from the acronym CNC (computerized numerical control). The T (tip or touch) represents the possibility to enter NC programs directly at the control or to program them graphically using gestures.
7	The product number indicates the control generation. The range of functions depends on the enabled software options.

2.2 Proper and intended operation

The information about the proper and intended operation supports you as user in the safe handling of a product such as a machine tool.

The control is a machine component but not a complete machine. This User's Manual describes the use of the control. Before using the machine including the control, take the OEM documentation to inform yourself about the safety-related aspects, the necessary safety equipment as well as the requirements on the qualified personnel.



HEIDENHAIN sells controls designed for milling and turning machines as well as for machining centers with up to 24 axes. If you as a user face a different constellation, then contact the owner immediately.

HEIDENHAIN contributes additionally to enhancing your safety and that of your products, notably by taking into consideration the customer feedback. This results e.g. in function adaptations of the controls and safety precautions in the information products.



Contribute actively to increasing the safety by reporting any missing or misleading information.

Further information: "Contact to the editorial staff", Page 23

2.3 Intended place of operation

In accordance with DIN EN 50370-1 standard referring to electromagnetic compatibility (EMC), the control is approved for use in industrial environments.

Definitions

Guideline	Definition
DIN EN 50370-1:2006-02	This standard deals, among other things, with interference emissions and immunity to interference of machine tools.

2.4 Safety precautions

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

The following safety precautions refer exclusively to the control as an individual component but not to the specific complete product, i.e. the machine tool.



Refer to your machine manual.

Before using the machine including the control, take the OEM documentation to inform yourself about the safety-related aspects, the necessary safety equipment as well as the requirements on the qualified personnel.

The following overview contains exclusively the generally valid safety precautions. Pay attention to additional safety precautions that may vary with the configuration and are given in the following chapters.



For ensuring maximum safety, all safety precautions are repeated at the relevant places within the chapters.

DANGER

Caution: hazard to the user!

Unsecured connections, defective cables, and improper use are always sources of electrical dangers. The hazard starts when the machine is powered up!

- ▶ Devices should be connected or removed only by authorized service technicians
- ▶ Only switch on the machine via a connected handwheel or a secured connection

DANGER

Caution: hazard to the user!

Machines and machine components always pose mechanical hazards. Electric, magnetic, or electromagnetic fields are particularly hazardous for persons with cardiac pacemakers or implants. The hazard starts when the machine is powered up!

- ▶ Read and follow the machine manual
- ▶ Read and follow the safety precautions and safety symbols
- ▶ Use the safety devices

WARNING

Caution: hazard to the user!

Manipulated data records or software can lead to an unexpected behavior of the machine. Malicious software (viruses, Trojans, malware, or worms) can cause changes to data records and software.

- ▶ Check any removable memory media for malicious software before using them
- ▶ Start the internal web browser only from within the sandbox

NOTICE**Danger of collision!**

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect pre-positioning or insufficient spacing between components can lead to a risk of collision when referencing the axes.

- ▶ Pay attention to the information on the screen
- ▶ If necessary, move to a safe position before referencing the axes
- ▶ Watch out for possible collisions

NOTICE**Danger of collision!**

The control uses the defined tool lengths for tool length compensation. Incorrect tool lengths will result in an incorrect tool length compensation. The control does not perform a length compensation and a collision check for tools with a length of **0** and after **TOOL CALL 0**. Danger of collision during subsequent tool positioning movements!

- ▶ Always define the actual tool length of a tool (not just the difference)
- ▶ Use **TOOL CALL 0** only to empty the spindle

NOTICE**Danger of collision!**

NC programs that were created on older controls can lead to unexpected axis movements or error messages on current control models. Danger of collision during machining!

- ▶ Check the NC program or program section using the graphic simulation
- ▶ Carefully test the NC program or program section in the **Program run, single block** operating mode
- ▶ Pay attention to the following known differences (the list below might not be complete!)

NOTICE**Caution: Data may be lost!**

The **DELETE** function permanently deletes the file. The control does not perform an automatic backup of the file prior to deletion (e.g., there is no recycle bin). Files are thereby irreversibly deleted.

- ▶ Regularly back up important data to external drives

NOTICE**Caution: Data may be lost!**

Never remove a connected USB device during data transfer—data can be damaged or deleted!

- ▶ Use the USB port for data transfer and backup only; do not use it for editing and executing NC programs
- ▶ Use the soft key to remove a USB device when data the transfer is complete

NOTICE**Caution: Data may be lost!**

The control must be shut down so that running processes can be concluded and data can be saved. Immediate switch-off of the control by turning off the main switch can lead to data loss no matter what state the control was in!

- ▶ Always shut down the control
- ▶ Only operate the main switch after being prompted on the screen

2.5 Software

This User's Manual describes the functions for setting up the machine as well as for programming and running your NC programs. These functions are available for a control featuring the full range of functions.



The actual range of functions depends, among other things, on the enabled software options.

Further information: "Software options", Page 31

The table shows the NC software numbers described in this User's Manual.



HEIDENHAIN has simplified the version schema, starting with NC software version 16:

- The publication period determines the version number.
- All control models of a publication period have the same version number.
- The version number of the programming stations corresponds to the version number of the NC software.

NC software number	Product
817620-16	TNC7
817621-16	TNC7 E
817625-16	TNC7 programming station



Refer to your machine manual.

This User's Manual describes the basic functions of the control. The machine manufacturer can adapt, enhance or restrict the control functions to the machine.

Check, on the basis of the machine tool manual, whether the machine manufacturer has adapted the functions of the control.

Definition

Abbreviation	Definition
E	The suffix E indicates the export version of the control. In this version, the software option 9 Advanced Function Set 2 is restricted to 4-axis interpolation.

2.5.1 Software options

Software options define the range of functions of the control. The optional functions are either machine- or application-specific. The software options give you the possibility of adapting the control to your individual needs.

You can check which software options are enabled on your machine.

Further information: User's Manual for Setup and Program Run

Overview and definitions

The **TNC7** features various software options, each of which can be enabled separately and even subsequently by the machine manufacturer. The following overview includes only those software options that are relevant for you as user.



The option numbers indicated in the User's Manual show you that a function is not included in the standard range of available functions. The Technical Manual informs about additional software options that are relevant for the machine manufacturer.



Keep in mind that particular software options also require hardware extensions.

Further information: User's Manual for Setup and Program Run

Software option	Definition and application
Additional Axis (options 0 to 7)	Additional control loop A control loop is required for each axis or spindle moved to a programmed nominal value by the control. Additional control loops are required e.g. for detachable and motor-driven tilting tables.
Advanced Function Set 1 (option 8)	Advanced functions (set 1) On machines with rotary axes this software option enables the machining of multiple workpiece sides in a single setup. The software option includes the following functions: <ul style="list-style-type: none"> ■ Tilting the working plane, e.g. with PLANE SPATIAL Further information: User's Manual for Programming and Testing ■ Programming of contours on the unrolled surface of a cylinder, e.g. by using Cycle 27 CYLINDER SURFACE Further information: User's Manual for Machining Cycles ■ Programming the rotary axis feed rate in mm/min with M116 Further information: User's Manual for Programming and Testing ■ 3-axis circular interpolation with a tilted working plane The advanced functions (set 1) reduce the setup effort and increase the workpiece accuracy.

Software option	Definition and application
Advanced Function Set 2 (option 9)	<p>Advanced functions (set 2)</p> <p>On machines with rotary axes this software option enables the simultaneous 5-axis machining of workpieces.</p> <p>The software option includes the following functions:</p> <ul style="list-style-type: none"> ■ TCPM (tool center point management): Automatic tracking of linear axes during the rotary axis positioning <p>Further information: User's Manual for Programming and Testing</p> <ul style="list-style-type: none"> ■ Running of NC programs with vectors, including optional 3D tool compensation <p>Further information: User's Manual for Programming and Testing</p> <ul style="list-style-type: none"> ■ Manual moving of axes in the active tool coordinate system T-CS ■ Linear interpolation in more than 4 axes (max. 4 axes in case of an export version) <p>The advanced functions (set 2) can be used to produce free-form surfaces.</p>
HEIDENHAIN DNC (option 18)	<p>HEIDENHAIN DNC</p> <p>This software option enables external Windows applications to access data of the control via the TCP/IP protocol.</p> <p>Potential fields of application are e.g.:</p> <ul style="list-style-type: none"> ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data <p>HEIDENHAIN DNC is required in conjunction with external Windows applications.</p>
Dynamic Collision Monitoring (option 40)	<p>Dynamic Collision Monitoring (DCM)</p> <p>The machine manufacturer can use this software option to define machine components as collision objects. The control monitors the defined collision objects during all machine movements.</p> <p>The software option includes the following functions:</p> <ul style="list-style-type: none"> ■ Automatic interruption of the program run whenever a collision is imminent. ■ Warnings in case of manual axis movements ■ Collision monitoring in Test Run mode <p>With DCM you can prevent collisions and thus avoid additional costs resulting from material damage or a machine downtime.</p> <p>Further information: User's Manual for Setup and Program Run</p>
CAD Import (option 42)	<p>CAD Import</p> <p>This software option is used to select positions and contours from CAD files and to transfer them into an NC program.</p> <p>With the CAD Import option you reduce the programming effort and prevent typical errors such as the incorrect entry of values. In addition, CAD Import contributes to paperless manufacturing.</p> <p>Further information: User's Manual for Setup and Program Run</p>
Global PGM Settings (option 44)	<p>Global Program Settings GPS</p> <p>This software option can be used for superimposed coordinate transformations and handwheel movements during program run without changing the NC program.</p> <p>With GPS you can adapt externally created NC programs to the machine and increase flexibility during program run.</p> <p>Further information: User's Manual for Setup and Program Run</p>

Software option	Definition and application
Adaptive Feed Control (option 45)	<p>Adaptive Feed Control AFC</p> <p>This software option enables an automatic feed control that depends on the current spindle load. The control increases the feed rate as the load decreases and reduces the feed rate as the load increases.</p> <p>With AFC you can shorten the machining time without adapting the NC program, while preventing machine damage from overload at the same time.</p> <p>Further information: User's Manual for Setup and Program Run</p>
KinematicsOpt (option 48)	<p>KinematicsOpt</p> <p>This software option uses automatic probing processes to check and optimize the active kinematics.</p> <p>With KinematicsOpt the control can correct position errors on rotary axes and thus increase the accuracy of machining operations in the tilted working plane and of simultaneous machining operations. In part, the control can compensate temperature-induced deviations through repeated measurements and corrections.</p> <p>Further information: "Touch Probe Cycles: Automatic Kinematics Measurement", Page 297</p>
Turning (option 50)	<p>Mill-turning</p> <p>This software option offers a comprehensive milling-specific function package for milling machines with rotary tables.</p> <p>The software option includes the following functions:</p> <ul style="list-style-type: none"> ■ Turning-specific tools ■ Turning-specific cycles and contour elements such as undercuts ■ Automatic tool radius compensation <p>Mill-turning enables mill-turning machining operations on only one machine, thus reducing e.g. the setup work effort considerably.</p> <p>Further information: User's Manual for Programming and Testing</p>
KinematicsComp (option 52)	<p>KinematicsComp</p> <p>This software option uses automatic probing processes to check and optimize the active kinematics.</p> <p>With KinematicsComp the control can correct position and component errors in three dimensions. This means it can compensate the errors of rotary and linear axes in three dimensions. Compared to KinematicsOpt (option 48), the corrections are even far more comprehensive.</p> <p>Further information: "Cycle 453 KINEMATICS GRID (option 48), (option 52)", Page 331</p>
OPC UA NC Server 1 to 6 (options 56 to 61)	<p>OPC UA NC Server</p> <p>The software options offer the OPC UA standardized interface for the external access to data and functions of the control.</p> <p>Potential fields of application are e.g.:</p> <ul style="list-style-type: none"> ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data <p>Each software option enables one client connection each. Several parallel connections require the use of multiple OPC UA NC servers.</p> <p>Further information: User's Manual for Setup and Program Run</p>

Software option	Definition and application
4 Additional Axes (option 77)	4 additional control loops see " Additional Axis (options 0 to 7)"
8 Additional Axes (option 78)	8 additional control loops see " Additional Axis (options 0 to 7)"
3D-ToolComp (option 92)	3D-ToolComp only in connection with Advanced Function Set 2 (option 9) With this software option, shape deviations on ball cutters and workpiece probes can be automatically compensated using a correction value table. 3D-ToolComp enables increasing the workpiece accuracy in conjunction with free-form surfaces, for example. Further information: User's Manual for Programming and Testing
Extended Tool Management (option 93)	Extended tool management This software option extends tool management by the two tables Tooling list and T usage order . The tables show the following contents: <ul style="list-style-type: none"> ■ The Tooling list shows the tool requirements of the NC program or pallet to be run ■ The T usage order shows the tool order of the NC program or pallet to be run Further information: User's Manual for Setup and Program Run The extended tool management enables you to detect the tool requirements in time and thus prevent interruptions during program run.
Advanced Spindle Interpolation (option 96)	Interpolating Spindle This software option enables interpolation turning, as the control couples the tool spindle with the linear axes. The software option includes the following cycles: <ul style="list-style-type: none"> ■ Cycle 291 COUPLG.TURNG.INTERP. for simple turning machining operations without contour subprograms ■ Cycle 292 CONTOUR.TURNG.INTRP. for finishing rotationally symmetrical contours The interpolating spindle enables you to execute a turning operation also on machines without rotary table. Further information: User's Manual for Machining Cycles
Spindle Synchronism (option 131)	Spindle synchronism This software option synchronizes two or more spindles and thus enables e.g. the manufacture gears by hobbing. The software option includes the following functions: <ul style="list-style-type: none"> ■ Spindle synchronism for special machining operation, e.g. polygonal turning ■ Cycle 880 GEAR HOBGING only in conjunction with mill-turning (option 50) Further information: User's Manual for Machining Cycles
Remote Desktop Manager (option 133)	Remote Desktop Manager This software option is used to display and operate externally linked computer units. With Remote Desktop Manager you reduce the distances covered between several workplaces and as a result increase the efficiency. Further information: User's Manual for Setup and Program Run

Software option	Definition and application
Dynamic Collision Monitoring v2 (option 140)	<p>Dynamic Collision Monitoring (DCM) version 2</p> <p>This software option includes the functions of software option 40 (Dynamic Collision Monitoring, DCM).</p> <p>In addition, this software option can be used for the collision monitoring of workpiece fixtures.</p> <p>Further information: User's Manual for Setup and Program Run</p>
Cross Talk Compensation (option 141)	<p>Compensation of axis couplings CTC</p> <p>Using this software option, the machine manufacturer can e.g. compensate acceleration-induced deviations at the tool and thus increase accuracy and dynamic performance.</p>
Position Adaptive Control (option 142)	<p>Position adaptive control PAC</p> <p>Using this software option, the machine manufacturer can e.g. compensate position-induced deviations at the tool and thus increase accuracy and dynamic performance.</p>
Load Adaptive Control (option 143)	<p>Load adaptive control LAC</p> <p>Using this software option, the machine manufacturer can e.g. compensate load-induced deviations at the tool and thus increase accuracy and dynamic performance.</p>
Motion Adaptive Control (option 144)	<p>Motion adaptive control MAC</p> <p>Using this software option, the machine manufacturer can e.g. change speed-dependent machine settings and thus increase the dynamic performance.</p>
Active Chatter Control (option 145)	<p>Active chatter control ACC</p> <p>With this software option the chatter tendency of a machine used for heavy machining can be reduced.</p> <p>The control can use ACC to improve the surface quality of the workpiece, increase the tool life and reduce the machine load. Depending on the type of machine, the metal-removal rate can be increased by more than 25 %.</p> <p>Further information: User's Manual for Setup and Program Run</p>
Machine Vibration Control (option 146)	<p>Vibration damping for machines MVC</p> <p>Damping of machine oscillations for improving the workpiece surface quality through the following functions:</p> <ul style="list-style-type: none"> ■ AVD Active Vibration Damping ■ FSC Frequency Shaping Control
CAD Model Optimizer (option 152)	<p>Optimization of CAD models</p> <p>This software option can be used, for example, to repair faulty files of fixtures and tool holders or to position STL files generated from the simulation for a different machining operation.</p> <p>Further information: User's Manual for Setup and Program Run</p>

Software option	Definition and application
Batch Process Manager (option 154)	<p>Batch Process Manager BPM</p> <p>This software option makes it easy to plan and execute multiple production jobs.</p> <p>If pallet management and extended tool management (option 93) are extended or combined, BPM offers the following additional data, for example:</p> <ul style="list-style-type: none"> ■ Machining time ■ Availability of necessary tools ■ Manual interventions to be made ■ Program test results of assigned NC programs <p>Further information: User's Manual for Programming and Testing</p>
Component Monitoring (option 155)	<p>Component monitoring</p> <p>This software option enables the automatic monitoring of machine components configured by the machine manufacturer.</p> <p>Component monitoring assists the control in preventing machine damage due to overload by way of hazard warnings and error messages.</p>
Grinding (option 156)	<p>Jig grinding</p> <p>This software option offers a comprehensive grinding-specific function package for milling machines.</p> <p>The software option includes the following functions:</p> <ul style="list-style-type: none"> ■ Grinding-specific tools including dressing tools ■ Cycles for reciprocating stroke and dressing <p>Jig-turning enables complete machining operations on just one machine, thus reducing e.g. the setup work effort considerably.</p> <p>Further information: User's Manual for Programming and Testing</p>
Gear Cutting (option 157)	<p>Gear manufacturing</p> <p>This software option enables the manufacturing of cylindrical gears or helical gears of any angle.</p> <p>The software option includes the following cycles:</p> <ul style="list-style-type: none"> ■ Cycle 285 DEFINE GEAR to define the gear geometry ■ Cycle 286 GEAR HOBGING ■ Cycle 287 GEAR SKIVING <p>Gear manufacturing expands the scope of functionality of milling machines with rotary tables even without mill-turning (option 50).</p> <p>Further information: User's Manual for Machining Cycles</p>
Turning v2 (option 158)	<p>Mill-turning version 2</p> <p>This software option includes all functions of Mill-Turning (software option 50). In addition, this software option offers the following advanced turning functions:</p> <ul style="list-style-type: none"> ■ Cycle 882 SIMULTANEOUS ROUGHING FOR TURNING ■ Cycle 883 TURNING SIMULTANEOUS FINISHING <p>The advanced turning functions not only enable you to manufacture undercut workpieces but also to use a larger area of the indexable insert during the machining operation.</p> <p>Further information: User's Manual for Machining Cycles</p>

Software option	Definition and application
Optimized Contour Milling (option 167)	<p>Optimized contour machining (OCM)</p> <p>This software option enables the trochoidal milling of closed or open pockets and islands of any shape. During trochoidal milling, the full cutting edge is used under constant cutting conditions.</p> <p>The software option includes the following cycles:</p> <ul style="list-style-type: none"> ■ Cycle 271 OCM CONTOUR DATA ■ Cycle 272 OCM ROUGHING ■ Cycle 273 OCM FINISHING FLOOR and cycle 274 OCM FINISHING SIDE ■ Cycle 277 OCM CHAMFERING ■ In addition, the control offers OCM FIGURES for frequently needed contours <p>With OCM you can shorten the machining time while reducing tool wear at the same time.</p> <p>Further information: User's Manual for Machining Cycles</p>
Process Monitoring (option 168)	<p>Process monitoring</p> <p>Reference-based monitoring of the machining process</p> <p>The control uses this software option to monitor defined machining sections during program run. The control compares changes in conjunction with the tool spindle or the tool with the values of a reference machining operation.</p> <p>Further information: User's Manual for Setup and Program Run</p>

2.5.2 Feature Content Level

New functions or functional enhancements of the control software may be protected by software options or by means of the Feature Content Levels.

On purchasing a new control you will receive the highest level of the **FCL** possible with the installed software. A subsequent software update e.g. within the frame of a service requirement, does not automatically increase the **FCL** version.



As yet, no functions are protected by Feature Content Level. If functions are protected in future, the User's Manual will indicate the marking **FCL n**. The **n** shows the required number of the **FCL** version.

2.5.3 Information on licensing and use

Open-source software

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

To get to the licensing terms on the control:



► Select the **Home** operating mode

► Select the **Settings** application

► Select the **Operating system** tab



► Double-tap or double-click **About HeROS**

► The control opens the **HEROS Licence Viewer** window.

OPC UA

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

OPC UA NC Server (options 56 to 61) and HEIDENHAIN DNC (option 18) can be used to influence the behavior of the control. Before using these interfaces for productive purposes, system tests must be performed to exclude the occurrence of any malfunctions or performance failures of the control. The manufacturer of the software product that uses these communication interfaces is responsible for performing these tests.

Further information: User's Manual for Setup and Program Run

2.6 Comparison between TNC 640 and TNC7

The following tables contain the main differences between the TNC 640 and the TNC7.

Operating modes

Mode of operation	TNC 640	TNC7
Manual operation	<ul style="list-style-type: none"> ■ Separate Manual Operation operating mode ■ Executing manual probing cycles ■ Opening preset table and tool table ■ Shutting down the control 	<ul style="list-style-type: none"> ■ Manual operation application in the Manual operating mode ■ Executing manual probing cycles in the Setup application ■ Opening tables in the Tables operating mode ■ Shutting down the control in the Home operating mode ■ Tool call in the Manual operation application is possible
Electronic handwheel	Separate Electronic handwheel operating mode	Handwheel switch in the Manual operation application
Positioning w/ Manual Data Input	Separate Positioning w/ Manual Data Input operating mode	MDI application in the Manual operating mode
Program run, single block	Separate Program run, single block operating mode	Single Block switch in the Program Run operating mode
Program run, full sequence	Separate Program run, full sequence operating mode	Program Run operating mode
Programming	<ul style="list-style-type: none"> ■ Programming operating mode ■ Programming graphics with split screen layout in PROGRAM GRAPHICS 	<ul style="list-style-type: none"> ■ Editor operating mode ■ Contour graphics workspace for the import, drawing and export of contours
Test Run	Test Run operating mode	Simulation workspace in the Editor , Manual and Program Run operating modes



On the TNC7 the operating modes of the control are allocated differently than on the TNC 640. For reasons of compatibility and to facilitate ease of operation, the keys on the keyboard unit remain the same. Keep in mind that particular keys no longer activate a change of operating modes but e.g. Instead activate a switch.

Further information: User's Manual for Programming and Testing

Further information: User's Manual for Setup and Program Run

Functions

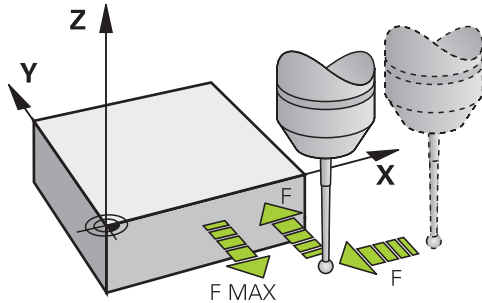
Function	TNC 640	TNC7
Programming and program run	<ul style="list-style-type: none"> ■ Programming and executing Klartext, ISO and FK ■ Inserting positioning blocks with the keyboard ■ Using soft keys to insert NC functions and cycles ■ Programming syntax in the text editor 	<ul style="list-style-type: none"> ■ Programming and executing Klartext ■ Programming and executing ISO and FK ■ Editing NC functions in the form ■ Importing and drawing contours, including FK ■ Exporting contours ■ Inserting positioning blocks using the keyboard, virtual keyboard or the Keyboard workspace ■ Inserting NC functions and cycles with the Insert NC function button ■ Programming syntax in the text editor
File management	Press the PGM MGT key to open it from the operating modes	Files operating mode and the Open File workspace
Tables	Opening the individual tables at specified places in the control	Separate Tables operating mode in which the tables of the control are opened and edited, if necessary
MOD functions	Adjusting settings in the MOD menu	Adjusting the settings in the Settings application of the Home operating mode
Calculator	<ul style="list-style-type: none"> ■ Loading the value from or into the dialog box via soft key ■ Applying axis values 	<ul style="list-style-type: none"> ■ Copying the value to the clipboard or pasting it from the clipboard ■ Restoring calculations from the history
Status display	<ul style="list-style-type: none"> ■ General status display and position display are always visible in the machine operating modes ■ Additional status display with split screen layout STATUS 	<ul style="list-style-type: none"> ■ General status display and position display in the Positions workspace ■ Additional status display in the Status workspace ■ Status overview and optional position display in the control bar

3

**Working with Touch
Probe Cycles**

3.1 General information about touch probe cycles

3.1.1 Method of function



The touch probe functions allow you to set presets on the workpiece, measure the workpiece, and determine and compensate for workpiece misalignment.

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 50

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).

Related topics

- Manual touch probe cycles
- Preset table
- Datum table
- Reference systems
- Preassigned variables

Further information: User's Manual for Setup and Program Run

Requirements

- Calibrated workpiece touch probe

Further information: "Touch Probe Cycles: Calibration", Page 279

Further information: "Touch Probe Cycles: Calibration", Page 279

If you are using a HEIDENHAIN touch probe, the software option 17, Touch Probe Functions, is automatically enabled.

3.1.2 Notes



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.

3.1.3 Touch probe cycles in the Manual Operation and Electronic Handwheel modes

In the **Setup** application under **Manual** mode, the control provides touch probe cycles that allow you to:

- Set presets
- Probe the angle
- Probe position
- Calibrate the touch probe
- Measure the tool

Further information: User's Manual for Setup and Program Run

3.1.4 Touch probe cycles for automatic operation

Besides the manual touch probe cycles, several cycles are available for a wide variety of applications in automatic operation:

- Automatic measurement of workpiece misalignment
- Automatic determination of the preset
- Automatic workpiece inspection
- Special functions
- Touch probe calibration
- Automatic kinematics measurement
- Automatic tool measurement

Defining touch probe cycles

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.

There are various ways to define the touch probe cycles. Touch probe cycles are programmed in the **Programming** mode of operation.

Inserting via NC function:







- ▶ Select **Insert NC function**
- The control opens the **Insert NC function** window.
- ▶ Select the desired cycle
- The control initiates a dialog and prompts you for all required input values.

Inserting via the TOUCH PROBE key:



- ▶ Select the **TOUCH PROBE** soft key
- The control opens the **Insert NC function** window.
- ▶ Select the desired cycle
- The control initiates a dialog and prompts you for all required input values.

Navigation in the cycle

Key	Function
	Navigation within the cycle: Jump to next parameter
	Navigation within the cycle: Jump to previous parameter
	Jump to the same parameter in the next cycle
	Jump to the same parameter in the previous cycle



The control provides selection possibilities for different cycle parameters via the action bar or the form.

Cycle input form

The control offers a **FORM** for various functions and cycles. This **FORM** can be used to enter various syntax elements or also cycle parameters on form-basis.

The screenshot shows a 'Cycle input form' with two main sections: 'Geometry' and 'Default'. The 'Geometry' section includes fields for 'First side length?' (60), 'Second side length?' (20), 'Corner radius?' (0), 'Depth?' (-20), and 'Workpiece surface coord...' (0). The 'Default' section includes 'Machining operation (0/1...)' (0), 'Plunging depth?' (5), 'Infeed for finishing?' (0), 'Feed rate for milling?' (F, 500), and 'Finishing feed rate?' (F, 500). At the bottom are 'Confirm', 'Discard', and 'Delete line' buttons.

The control allocates the cycle parameters in the **FORM** to groups based on their functions, e.g. geometry, standard, advanced, safety. The control provides selection possibilities for different cycle parameters via switches, for example. The control displays the currently edited cycle parameter in color.

After you have defined all required cycle parameters, you can confirm your input and conclude the cycle.

Opening the form:

- ▶ Open the **Editor** operating mode
- ▶ Open the **Program** workspace
- ▶ Select **FORM** via the title bar



If an input is invalid, the control displays an information symbol ahead of the syntax element. When you select the information symbol, the control displays information on the error.

Further information: User's Manual for Setup and Program Run

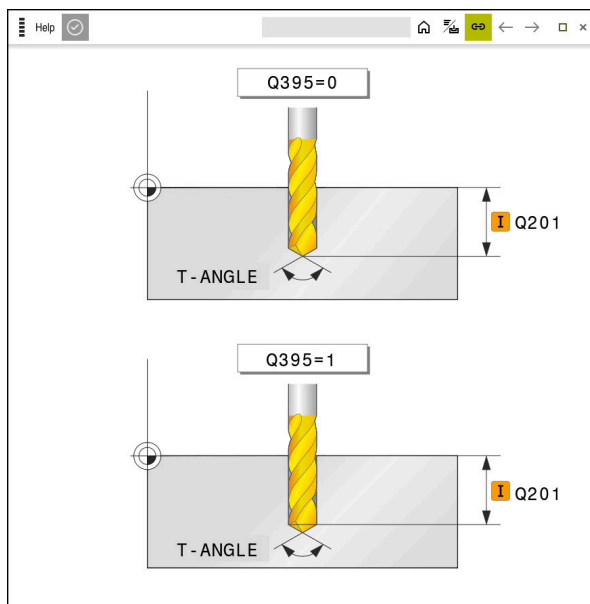
Help graphics

When you are editing a cycle, the control shows a help graphic for the current Q parameters. The size of the help graphic depends on the size of the **Program** workspace area.

The control shows the help graphic at the right edge of the workspace, or at the top or bottom edge. The help graphic is positioned in the half that does not contain the cursor.

When you tap or click on the help graphic, the control maximizes the help graphic.

If the **Help** workspace is active, the control will display the help graphic in this area instead of showing it in the **Program** workspace.



Help workspace with a help graphic for a cycle parameter

3.1.5 Available cycle groups

Machining cycles

Cycle group	Further information
Drilling/Thread <ul style="list-style-type: none"> ■ Drilling, reaming ■ Boring ■ Counterboring, centering ■ Tapping or thread milling 	Further information: User's Manual for Machining Cycles
Pockets/studs/slots <ul style="list-style-type: none"> ■ Pocket milling ■ Stud milling ■ Slot milling ■ Face milling 	Further information: User's Manual for Machining Cycles
Coordinate transformations <ul style="list-style-type: none"> ■ Mirroring ■ Rotating ■ Magnifying / Reducing 	Further information: User's Manual for Machining Cycles
SL cycles <ul style="list-style-type: none"> ■ SL (Subcontour List) cycles for the machining of contours that possibly consist of several subcontours ■ Cylinder surface machining ■ OCM (Optimized Contour Milling) cycles for combining subcontours to form complex contours 	Further information: User's Manual for Machining Cycles
Point patterns <ul style="list-style-type: none"> ■ Bolt hole circle ■ Linear hole pattern ■ Data Matrix code 	Further information: User's Manual for Machining Cycles
Turning cycles <ul style="list-style-type: none"> ■ Area clearance cycles, longitudinal and transverse ■ Recess turning cycles, radial and axial ■ Recessing cycles, radial and axial ■ Thread cutting cycles ■ Simultaneous turning cycles ■ Special cycles 	Further information: User's Manual for Machining Cycles

Cycle group	Further information
Special cycles <ul style="list-style-type: none"> ■ Dwell time ■ Program call ■ Tolerance ■ Oriented spindle stop ■ Engraving ■ Gear cycles ■ Interpolation turning 	Further information: User's Manual for Machining Cycles
Grinding cycles <ul style="list-style-type: none"> ■ Reciprocating stroke ■ Dressing ■ Correction cycles 	Further information: User's Manual for Machining Cycles

Measuring cycles

Cycle group	Further information
Rotation	
<ul style="list-style-type: none"> ■ Probing of plane, edge, two circles, beveled edge ■ Basic rotation ■ Two holes or studs ■ Via rotary axis ■ Via C-axis 	Page 55
Preset/Position	
<ul style="list-style-type: none"> ■ Rectangle, inside or outside ■ Circle, inside or outside ■ Corner, inside or outside ■ Center of bolt circle, slot or ridge ■ Touch probe axis or single axis ■ Four holes 	Page 119
Measuring	
<ul style="list-style-type: none"> ■ Angle ■ Circle, inside or outside ■ Rectangle, inside or outside ■ Slot or ridge ■ Bolt hole circle ■ Plane or coordinate 	Page 203
Special cycles	
<ul style="list-style-type: none"> ■ Measuring or measuring in 3D ■ Probing in 3D ■ Fast probing 	Page 261
Calibrating the touch probe	
<ul style="list-style-type: none"> ■ Calibrating the length ■ Calibration in a ring ■ Calibration on a stud ■ Calibration on a sphere 	Page 279
Measuring kinematics	
<ul style="list-style-type: none"> ■ Saving the kinematics ■ Measure kinematics ■ Preset compensation ■ Kinematics grid 	Page 297
Measuring the tool (TT)	
<ul style="list-style-type: none"> ■ Calibrating the TT ■ Tool length, radius or measuring completely ■ Calibrating the IR-TT ■ Lathe tool measurement 	Page 339

3.2 Before you start working with touch probe cycles!

3.2.1 General information

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

The following can be defined in the touch probe table:

- Type of tool
- Touch probe center offset
- Spindle angle during calibration
- Probing feed rate
- Rapid traverse in probing cycle
- Maximum measuring range
- Safety clearance
- Feed rate for pre-positioning
- Touch probe orientation
- Serial number
- Reaction in case of collision

Further information: User's Manual for Setup and Program Run

3.2.2 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.

Positioning logic

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 in the working plane to the first touch point, and then in the touch-probe axis directly to the set-up clearance.

Notes

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 **444** and **14xx**, no coordinate transformations must be active (e.g., Cycles **8 MIRRORING**, **11 SCALING FACTOR**, **26 AXIS-SPECIFIC SCALING**, **TRANS MIRROR**).

- ▶ Reset any coordinate transformations before the cycle call.

- Please note that the units of measure in the measuring log and the return parameters depend on the main program.
- The touch probe cycles **40x** to **43x** will reset an active basic rotation at the beginning of the cycle.
- The control interprets a basic transformation as a basic rotation, and an offset as a table rotation.
- You can compensate for the workpiece misalignment by rotating the table only if the machine is designed with a rotary table axis that is oriented perpendicularly with respect to the workpiece coordinate system **W-CS**.

Note about machine parameters

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

3.3 Program defaults for cycles

3.3.1 Entering GLOBAL DEF definitions

Insert
NC function

- ▶ Select **Insert NC function**
- The control opens the **Insert NC function** window.
- ▶ Select **GLOBAL DEF**
- ▶ Select the desired **GLOBAL DEF** function, e.g. **100 GENERAL**
- ▶ Enter the required definitions

3.3.2 Using GLOBAL DEF information

If you entered the corresponding **GLOBAL DEF** functions at program start, you can reference these globally valid values for the definition of any cycle.

Proceed as follows:

Insert
NC function

- ▶ Select **Insert NC function**
- The control opens the **Insert NC function** window.
- ▶ Select and define **GLOBAL DEF**
- ▶ Select **Insert NC function** again
- ▶ Select the desired cycle, e.g. **200 DRILLING**
- If the cycle includes global cycle parameters, the control superimposes the selection possibility **PREDEF** in the action bar or in the form as a selection menu.

PREDEF

- ▶ Select **PREDEF**
- The control then 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. Simulate your program before executing it
- ▶ If you enter fixed values in the cycles, they will not be changed by **GLOBAL DEF**.

3.3.3 Global data valid everywhere

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

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

Example

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

3.3.4 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, 1021, 1022, 1025, 1271, 1272, 1273, 1278**

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

Example

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

4

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

4.1 Overview



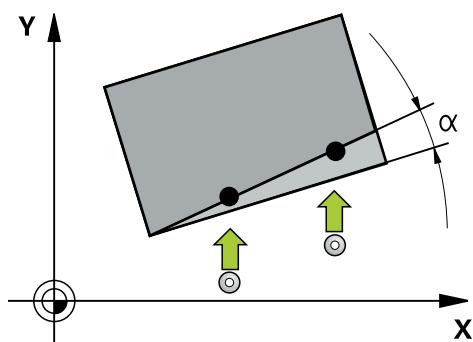
The control must be specifically prepared by the machine tool builder for the use of a 3-D touch probe.

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

Cycle	Call	Further information
1420 PROBING IN PLANE <ul style="list-style-type: none"> Automatic measurement using three points Compensation via basic rotation or rotary table rotation 	DEF- active	Page 67
1410 PROBING ON EDGE <ul style="list-style-type: none"> Automatic measurement using two points Compensation via basic rotation or rotary table rotation 	DEF- active	Page 73
1411 PROBING TWO CIRCLES <ul style="list-style-type: none"> Automatic measurement using two holes or studs Compensation via basic rotation or rotary table rotation 	DEF- active	Page 80
1412 INCLINED EDGE PROBING <ul style="list-style-type: none"> Automatic measurement using two points on an inclined edge Compensation via basic rotation or rotary table rotation 	DEF- active	Page 88
400 BASIC ROTATION <ul style="list-style-type: none"> Automatic measurement using two points Compensation via basic rotation 	DEF- active	Page 95
401 ROT OF 2 HOLES <ul style="list-style-type: none"> Automatic measurement using two holes Compensation via basic rotation 	DEF- active	Page 97
402 ROT OF 2 STUDS <ul style="list-style-type: none"> Automatic measurement using two studs Compensation via basic rotation 	DEF- active	Page 101
403 ROT IN ROTARY AXIS <ul style="list-style-type: none"> Automatic measurement using two points Compensation via rotary table rotation 	DEF- active	Page 106
405 ROT IN C AXIS <ul style="list-style-type: none"> Automatic alignment of an angular offset between a hole center and the positive Y axis Compensation via rotary table rotation 	DEF- active	Page 112
404 SET BASIC ROTATION <ul style="list-style-type: none"> Setting any basic rotation 	DEF- active	Page 117

4.2 Fundamentals of touch probe cycles 14xx

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



The cycles are capable of determining rotation and include the following:

- 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
Surface-normal vectors	

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

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

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

4.2.2 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.

For this purpose, precede the value for the required nominal position with **"?"**. You can do this by selecting **Name** in the action bar. Depending on the object, you need to define the nominal positions that determine the probing direction, see "Examples."



Depending on the object, you need to define the nominal positions that determine the probing direction,

Examples:

- see "Alignment using two holes", Page 60
- see "Alignment through an edge", Page 61
- see "Alignment via the plane", Page 62

Cycle sequence

Proceed as follows:



- ▶ Run the cycle
- The control interrupts the NC program.
- A window opens.
- ▶ Use the axis-direction keys to position the touch probe to the desired touch point

or



- ▶ Position the touch probe to the desired point using the electronic handwheel
- ▶ Change the probing direction in the window, if necessary
- ▶ Select the **NC start** key
- The control closes the window and performs the first probing operation.
- If **CLEAR. HEIGHT MODE Q1125 = 1** or **2**, the control opens a message in the **FN 16** tab, **Status** workspace, explaining that the Traverse to clearance height mode cannot be used here.



- ▶ Move the tool to a safe position
- ▶ Select the **NC start** key
- Cycle or program execution is resumed. You may then need to repeat the entire process for further touch points.

NOTICE

Danger of collision!

The control will ignore the programmed values 1 and 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.



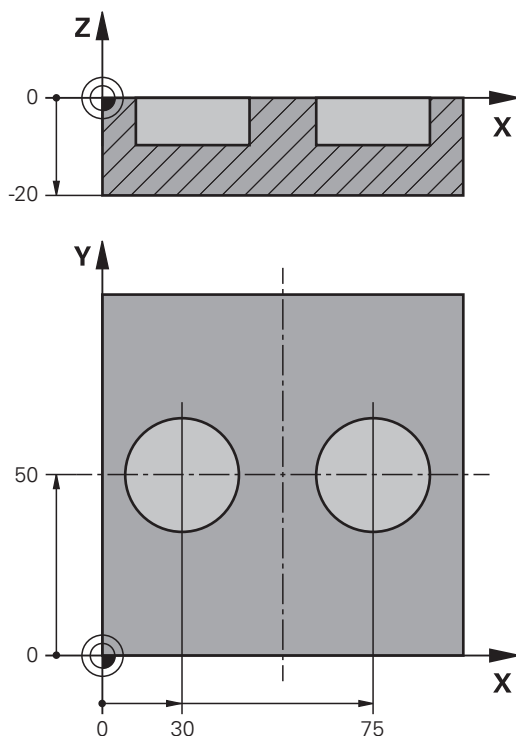
Programming and operating notes:

- See the drawing for these nominal positions.
- Semi-automatic mode is only executed in the machine operating modes, not in the simulation.
- 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.

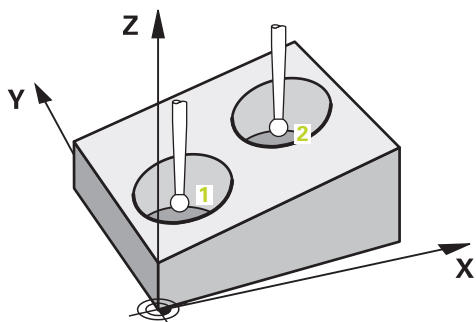
Examples

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

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



Alignment using two holes



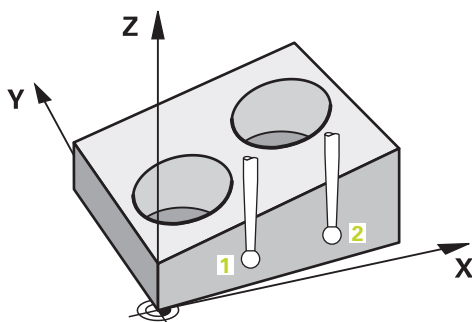
In this example, you will align two holes. Probing is done in the X axis (main axis) and in the Y axis (secondary axis). This means that it is mandatory to define the nominal position from the drawing for these axes! A nominal position for the Z axis (tool axis) is not necessary as you will not measure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

- **QS1100** = nominal position 1 in main axis is defined, but workpiece position is unknown
- **QS1101** = nominal position 1 in secondary axis is defined, but workpiece position is unknown
- **QS1102** = nominal position 1 in tool axis is unknown
- **QS1103** = nominal position 2 in main axis is defined, but workpiece position is unknown

- **QS1104** = nominal position 2 in secondary axis is defined, but workpiece position is unknown
- **QS1105** = nominal position 2 in tool axis is unknown

11 TCH PROBE 1411 PROBING TWO CIRCLES ~	
QS1100= "?30"	;1ST POINT REF AXIS ~
QS1101= "?50"	;1ST POINT MINOR AXIS ~
QS1102= "?"	;1ST POINT TOOL AXIS ~
Q1116=+10	;DIAMETER 1 ~
QS1103= "?75"	;2ND POINT REF AXIS ~
QS1104= "?50"	;2ND POINT MINOR AXIS ~
QS1105= "?"	;2ND POINT TOOL AXIS ~
Q1117=+10	;DIAMETER 2 ~
Q1115=+0	;GEOMETRY TYPE ~
Q423=+4	;NO. OF PROBE POINTS ~
Q325=+0	;STARTING ANGLE ~
Q1119=+360	;ANGULAR LENGTH ~
Q320=+2	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q1125=+2	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1126=+0	;ALIGN ROTARY AXIS ~
Q1120=+0	;TRANSER POSITION ~
Q1121=+0	;CONFIRM ROTATION

Alignment through an edge



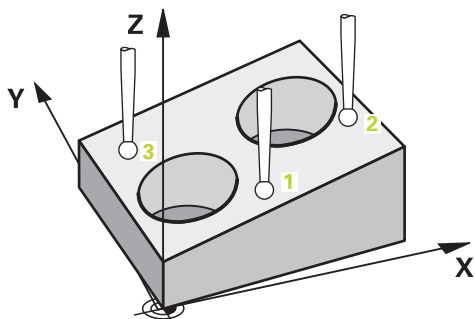
In this example, you will align an edge. Probing is done in the Y axis (secondary axis). This means that it is mandatory to define the nominal position from the drawing for these axes! Nominal positions for the X axis (main axis) and for the Z axis (tool axis) are not required because you will not measure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

- **QS1100** = nominal position 1 in main axis is unknown
- **QS1101** = nominal position 1 in secondary axis is defined, but workpiece position is unknown
- **QS1102** = nominal position 1 in tool axis is unknown
- **QS1103** = nominal position 2 in main axis is unknown

- **QS1104** = nominal position 2 in secondary axis is defined, but workpiece position is unknown
- **QS1105** = nominal position 2 in tool axis is unknown

11 TCH PROBE 1410 PROBING ON EDGE ~	
QS1100= "?"	;1ST POINT REF AXIS ~
QS1101= "?0"	;1ST POINT MINOR AXIS ~
QS1102= "?"	;1ST POINT TOOL AXIS ~
QS1103= "?"	;2ND POINT REF AXIS ~
QS1104= "?0"	;2ND POINT MINOR AXIS ~
QS1105= "?"	;2ND POINT TOOL AXIS ~
Q372=+2	;PROBING DIRECTION ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q1125=+2	;CLEAR. HEIGHT MODE ~
Q309=+0	;ERROR REACTION ~
Q1126=+0	;ALIGN ROTARY AXIS ~
Q1120=+0	;TRANSER POSITION ~
Q1121=+0	;CONFIRM ROTATION

Alignment via the plane



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

- **QS1100** = nominal position 1 in main axis is defined, but workpiece position is unknown
- **QS1101** = nominal position 1 in secondary axis is defined, but workpiece position is unknown
- **QS1102** = nominal position 1 in tool axis is defined, but workpiece position is unknown
- **QS1103** = nominal position 2 in main axis is defined, but workpiece position is unknown
- **QS1104** = nominal position 2 in secondary axis is defined, but workpiece position is unknown
- **QS1105** = nominal position 2 in tool axis is defined, but workpiece position is unknown
- **QS1106** = nominal position 3 in main axis is defined, but workpiece position is unknown

- **QS1107** = nominal position 3 in secondary axis is defined, but workpiece position is unknown
- **QS1108** = nominal position 3 in tool axis is defined, but workpiece position is unknown

11 TCH PROBE 1420 PROBING IN PLANE ~	; Define the cycle
QS1100= "?50" ;1ST POINT REF AXIS ~	; Nominal position 1 in main axis is defined, but workpiece position is unknown
QS1101= "?10" ;1ST POINT MINOR AXIS ~	; Nominal position 1 in secondary axis is defined, but workpiece position is unknown
QS1102= "?0" ;1ST POINT TOOL AXIS ~	; Nominal position 1 in tool axis is defined, but workpiece position is unknown
QS1103= "?80" ;2ND POINT REF AXIS ~	; Nominal position 2 in main axis is defined, but workpiece position is unknown
QS1104= "?50" ;2ND POINT MINOR AXIS ~	; Nominal position 2 in secondary axis is defined, but workpiece position is unknown
QS1105= "?0" ;2ND POINT TOOL AXIS ~	; Nominal position 2 in tool axis is defined, but workpiece position is unknown
QS1106= "?20" ;3RD POINT REF AXIS ~	; Nominal position 3 in main axis is defined, but workpiece position is unknown
QS1107= "?80" ;3RD POINT MINOR AXIS ~	; Nominal position 3 in secondary axis is defined, but workpiece position is unknown
QS1108= "?0" ;3RD POINT TOOL AXIS ~	; Nominal position 3 in tool axis is defined, but workpiece position is unknown
Q372=-3 ;PROBING DIRECTION ~	; Probing direction Z-
Q320=+2 ;SET-UP CLEARANCE ~	
Q260=+100 ;CLEARANCE HEIGHT ~	
Q1125=+2 ;CLEAR. HEIGHT MODE ~	
Q309=+0 ;ERROR REACTION ~	
Q1126=+0 ;ALIGN ROTARY AXIS ~	
Q1120=+0 ;TRANSER POSITION ~	
Q1121=+0 ;CONFIRM ROTATION	

4.2.3 Evaluation of tolerances

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

The following input values with tolerances are possible:

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



Pay attention to capitalization when entering tolerances.

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

The following cycle parameters allow input values with tolerances:

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

Program this as follows:

- ▶ Start the cycle definition
- ▶ Enable the Name selection option in the action bar
- ▶ Program nominal position/dimension incl. tolerance
- ▶ In the cycle, **QS1116="+8-2-1"** is defined, for example.



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

Cycle sequence

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

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

If Q309 = 1 or 2, proceed as follows:

- A window appears. The control displays all of the nominal and actual dimensions of the object.
 - Press the **CANCEL** button to interrupt NC program run
- or
- Press **NC start** to resume NC program run



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

Example

- QS1116 = diameter 1, tolerance specified
- QS1117 = diameter 2, tolerance specified

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

4.2.4 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 to the object. Based on the difference, the cycle calculates the required compensation values and applies tolerance monitoring.

Program this as follows:

- ▶ Define the cycle
- ▶ Enable the Name selection option in the action bar
- ▶ Program nominal position incl. tolerance, if necessary
- ▶ Program "@"
- ▶ Program actual position
- ▶ In the cycle, **QS1100="10+0.02@10.0123"** is defined, for example.



Programming and operating notes:

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

The nominal positions are defined here with tolerance monitoring and actual position.

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

4.3 Cycle 1420 PROBING IN PLANE

Application

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

If you program Cycle **1493 EXTRUSION PROBING** before this cycle, you can repeat probing points in a given direction over a specified distance.

Further information: "Cycle 1493 EXTRUSION PROBING", Page 276

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 58

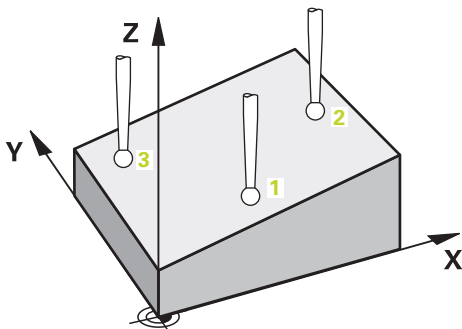
- Optionally, the cycle can monitor the tolerances. That way you can monitor the position and size of an object.

Further information: "Evaluation of tolerances", Page 64

- If you determine the actual position in advance, you can transfer it to the cycle as the actual position.

Further information: "Transferring the actual position", Page 66

Cycle sequence



- 1 Following the positioning logic, the control positions the touch probe to the programmed touch point **1** at rapid traverse **FMAX_PROBE**.
Further information: "Positioning logic", Page 50
- 2 The control then moves the touch probe to set-up clearance at rapid traverse **FMAX_PROBE**. The sum of **Q320, SET_UP** and the ball-tip radius is taken into account when probing in any probing direction.
- 3 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate **F** from the touch probe table.
- 4 The control offsets the touch probe by the safety clearance in the direction opposite to the direction of probing.
- 5 If you programmed retraction to clearance height **Q1125**, the touch probe returns to clearance height.
- 6 It then moves in the working plane to touch point **2** to measure the actual value of the second touch point in the plane.
- 7 The touch probe returns to the clearance height (depending on **Q1125**), then moves in the working plane to touch point **3** and measures the actual position of the third point of the plane.
- 8 Finally, the control retracts the touch probe to the clearance height (depending on **Q1125**) and saves the measured values in the following Q parameters:

Q parameter number	Meaning
Q950 to Q952	Measured position 1 in the main axis, secondary axis, and tool axis
Q953 to Q955	Measured position 2 in the main axis, secondary axis, and tool axis
Q956 to Q958	Measured position 3 in the main axis, secondary axis, and tool axis
Q961 to Q963	Measured spatial angle SPA, SPB, and SPC in the W-CS
Q980 to Q982	Measured deviations of touch point 1
Q983 to Q985	Measured deviations of touch point 2
Q986 to Q988	Deviation 3 of the positions
Q183	Workpiece status <ul style="list-style-type: none"> ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap
Q970	If you have programmed Cycle 1493 EXTRUSION PROBING before: Mean value of all deviations from the ideal line of the first touch point
Q971	If you have programmed Cycle 1493 EXTRUSION PROBING before: Mean value of all deviations from the ideal line of the second touch point
Q972	If you have programmed Cycle 1493 EXTRUSION PROBING before: Mean value of all deviations from the ideal line of the third touch point

Notes

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

NOTICE

Danger of collision!

When running touch probe cycles **444** and **14xx**, no coordinate transformations must be active (e.g., Cycles **8 MIRRORING**, **11 SCALING FACTOR**, **26 AXIS-SPECIFIC SCALING**, **TRANS MIRROR**).

- ▶ Reset any coordinate transformations before the cycle call.

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control can calculate the angle values only if the three touch points are not positioned on a straight line.
- The nominal spatial angle results from the defined nominal positions. The 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.



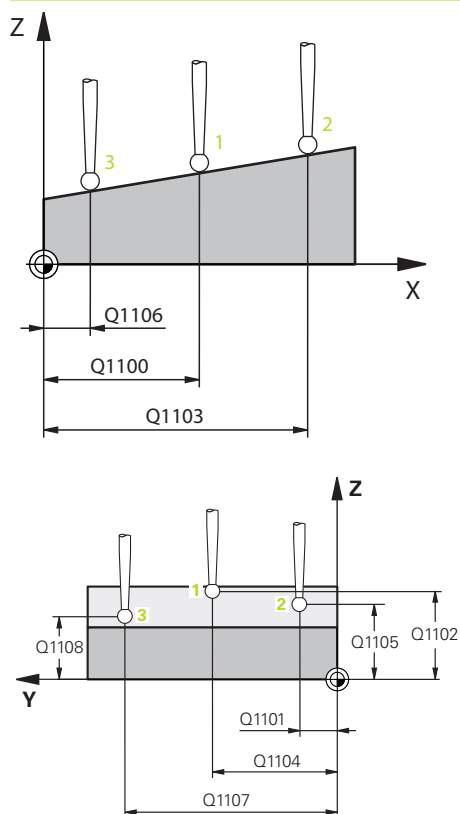
- HEIDENHAIN recommends avoiding the use of axis angles in this cycle!

Aligning the rotary table axes:

- Alignment with rotary table axes is possible only 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 used (**Q1121** not equal to 0). Otherwise, an error message will be displayed.

4.3.1 Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

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

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

- **?**: Semiautomatic mode, see Page 58
- **-, +**: Evaluation of the tolerance, see Page 64
- **@**: Transferring the actual position, see Page 66

Q1101 1st noml. position of minor axis?

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

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

Q1102 1st nominal position tool axis?

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

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

Q1103 2nd noml. position of ref axis?

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

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

Q1104 2nd noml. position of minor axis?

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

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

Q1105 2nd nominal pos. of tool axis?

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

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

Q1106 3rd noml. position of ref axis?

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

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

Help graphic

Parameter

Q1107 3rd noml. position minor axis?

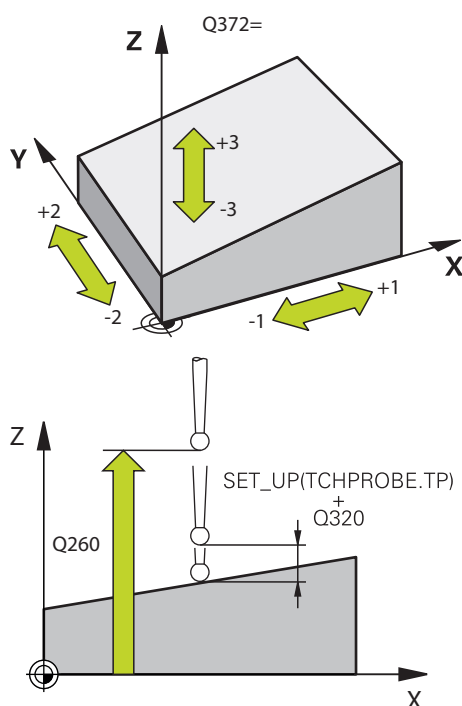
Absolute nominal position of the third touch point in the secondary axis of the working plane

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

Q1108 3rd nominal position tool axis?

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

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

**Q372 Probe direction (-3 to +3)?**

Axis defining the direction of probing. With the algebraic sign, you define the positive or negative direction of traverse of the probing axis.

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

-1: Do not move to clearance height.

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 touch point. Pre-positioning occurs at **FMAX_PROBE**.

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

Help graphic	Parameter
	<p>Q309 Reaction to tolerance error?</p> <p>Reaction when tolerance is exceeded:</p> <p>0: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results.</p> <p>1: Interrupt program run when tolerance is exceeded. The control opens a window with the results.</p> <p>2: The control opens a window with the results if the actual position is in the scrap range. Program run is interrupted. The control does not open a window with the results if rework is necessary.</p> <p>Input: 0, 1, 2</p>
	<p>Q1126 Align rotary axes?</p> <p>Position the rotary axes for inclined machining:</p> <p>0: Retain the current position of the rotary axis.</p> <p>1: Automatically position the rotary axis, and orient the tool tip (MOVE). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes.</p> <p>2: Automatically position the rotary axis without orienting the tool tip (TURN).</p> <p>Input: 0, 1, 2</p>
	<p>Q1120 Transfer position?</p> <p>Define which touch point will be used to correct the active preset:</p> <p>0: No correction</p> <p>1: Correction based on the 1st touch point</p> <p>2: Correction based on the 2nd touch point</p> <p>3: Correction based on the 3rd touch point</p> <p>4: Correction based on the averaged touch point position</p> <p>Input: 0, 1, 2, 3, 4</p>
	<p>Q1121 Confirm basic rotation?</p> <p>Define whether the control will use the determined misalignment as a basic rotation:</p> <p>0: No basic rotation</p> <p>1: Set basic rotation: The control will save the basic rotation</p> <p>Input: 0, 1</p>

Example

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

4.4 Cycle 1410 PROBING ON EDGE

Application

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

If you program Cycle **1493 EXTRUSION PROBING** before this cycle, you can repeat probing points in a given direction over a specified distance.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 276

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.

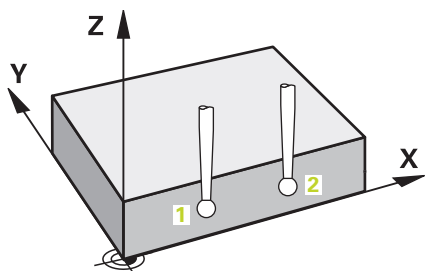
Further information: "Semi-automatic mode", Page 58

- 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 transfer it to the cycle as the actual position.

Further information: "Transferring the actual position", Page 66

Cycle sequence

- 1 Following the positioning logic, the control positions the touch probe to the programmed touch point **1** at rapid traverse **FMAX_PROBE**.
Further information: "Positioning logic", Page 50
- 2 The control then moves the touch probe to set-up clearance at rapid traverse **FMAX_PROBE**. The sum of **Q320, SET_UP** and the ball-tip radius is taken into account when probing in any probing direction.
- 3 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate **F** from the touch probe table.
- 4 The control offsets the touch probe by the safety clearance in the direction opposite to the direction of probing.
- 5 If you programmed retraction to clearance height **Q1125**, the touch probe returns to clearance height.
- 6 The touch probe then moves to the next touch point **2** and probes again.
- 7 Finally, the control retracts the touch probe to the clearance height (depending on **Q1125**) and saves the measured values in the following Q parameters:

Q parameter number	Meaning
Q950 to Q952	Measured position 1 in the main axis, secondary axis, and tool axis
Q953 to Q955	Measured position 2 in the main axis, secondary axis, and tool axis
Q964	Measured basic rotation
Q965	Measured table rotation
Q980 to Q982	Measured deviations of touch point 1
Q983 to Q985	Measured deviations of touch point 2
Q994	Measured angle deviation of basic rotation
Q995	Measured angle deviation of table rotation
Q183	Workpiece status <ul style="list-style-type: none"> ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap
Q970	If you have programmed Cycle 1493 EXTRUSION PROBING before: Mean value of all deviations from the ideal line of the first touch point
Q971	If you have programmed Cycle 1493 EXTRUSION PROBING before: Mean value of all deviations from the ideal line of the second touch point

Notes

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

NOTICE

Danger of collision!

When running touch probe cycles **444** and **14xx**, no coordinate transformations must be active (e.g., Cycles **8 MIRRORING**, **11 SCALING FACTOR**, **26 AXIS-SPECIFIC SCALING**, **TRANS MIRROR**).

- ▶ Reset any coordinate transformations before the cycle call.

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

Note about rotary axes:

When determining the basic rotation in a tilted working plane, keep the following in mind:

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

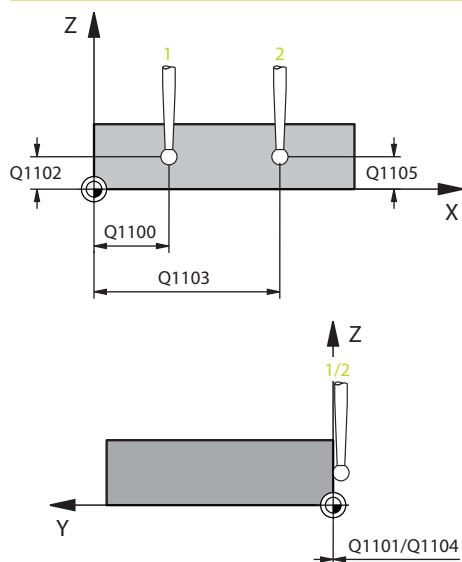
In the optional machine parameter **chkTiltingAxes** (no. 204601), the machine manufacturer defines a check verifying the tilting situation. If no checking is configured, the cycle always assumes that the working plane is consistent. The basic rotation is then calculated in the **I-CS**.

Aligning the rotary table axes:

- The control can align the rotary table only if the measured rotation can be compensated using a rotary table axis. This axis must be the first rotary table axis (as viewed from the workpiece).
- To align the rotary table axes (**Q1126** not equal to 0), you need to adopt the rotation (**Q1121** not equal to 0). Otherwise, the control will display an error message.

4.4.1 Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

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

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

- **?**: Semiautomatic mode, see Page 58
- **-, +**: Evaluation of the tolerance, see Page 64
- **@**: Transferring the actual position, see Page 66

Q1101 1st noml. position of minor axis?

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

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

Q1102 1st nominal position tool axis?

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

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

Q1103 2nd noml. position of ref axis?

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

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

Q1104 2nd noml. position of minor axis?

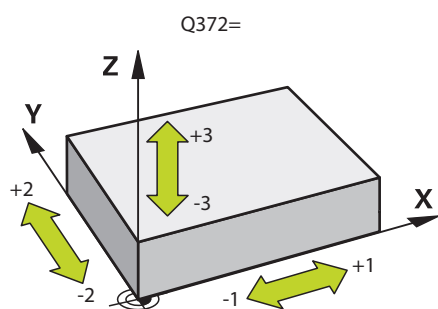
Absolute nominal position of the second touch point in the secondary axis of the working plane

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

Q1105 2nd nominal pos. of tool axis?

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

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

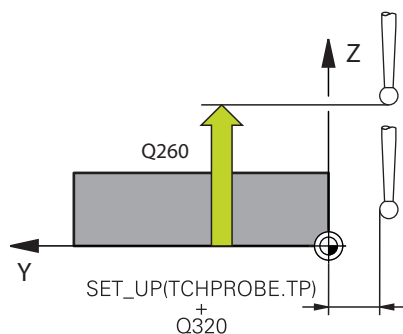


Q372 Probe direction (-3 to +3)?

Axis defining the direction of probing. With the algebraic sign, you define the positive or negative direction of traverse of the probing axis.

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

Help graphic



Parameter

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

-1: Do not move to clearance height.

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 touch point. Pre-positioning occurs at **FMAX_PROBE**.

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

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

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

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

2: The control opens a window with the results if the actual position is in the scrap range. Program run is interrupted. The control does not open a window with the results if rework is necessary.

Input: **0, 1, 2**

Help graphic	Parameter
	<p>Q1126 Align rotary axes?</p> <p>Position the rotary axes for inclined machining:</p> <p>0: Retain the current position of the rotary axis.</p> <p>1: Automatically position the rotary axis, and orient the tool tip (MOVE). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes.</p> <p>2: Automatically position the rotary axis without orienting the tool tip (TURN).</p> <p>Input: 0, 1, 2</p>
	<p>Q1120 Transfer position?</p> <p>Define which touch point will be used to correct the active preset:</p> <p>0: No correction</p> <p>1: Correction based on the 1st touch point</p> <p>2: Correction based on the 2nd touch point</p> <p>3: Correction based on the averaged touch point position</p> <p>Input: 0, 1, 2, 3</p>
	<p>Q1121 CONFIRM ROTATION?</p> <p>Define whether the control will use the determined misalignment as a basic rotation:</p> <p>0: No basic rotation</p> <p>1: Set basic rotation: The control transfers the misalignment to the preset table as basic transformations.</p> <p>2: Rotate the rotary table: The control transfers the misalignment to the preset table as offset.</p> <p>Input: 0, 1, 2</p>

Example

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

4.5 Cycle 1411 PROBING TWO CIRCLES

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.

If you program Cycle **1493 EXTRUSION PROBING** before this cycle, you can repeat probing points in a given direction over a specified distance.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 276

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.

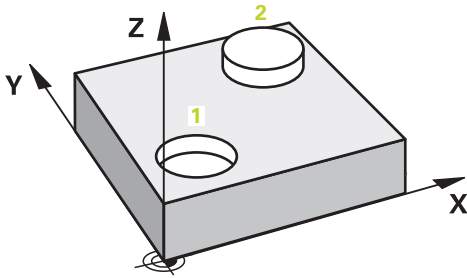
Further information: "Semi-automatic mode", Page 58

- 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 transfer it to the cycle as the actual position.

Further information: "Transferring the actual position", Page 66

Cycle sequence

- 1 Following the positioning logic, the control positions the touch probe to the programmed center point **1** at the feed rate (depending on **Q1125**).
Further information: "Positioning logic", Page 50
- 2 The control then moves the touch probe to set-up clearance at rapid traverse **FMAX_PROBE**. The sum of **Q320, SET_UP** and the ball-tip radius is taken into account when probing in any probing direction.
- 3 Then the touch probe moves to the entered measuring height at the probing feed rate **F** from the touch probe table and probes (the number of touch points to be probed depends on **Q423** Number of Probes) the first hole or stud center point.
- 4 The control offsets the touch probe by the safety clearance in the direction opposite to the direction of probing.
- 5 The touch probe returns to the clearance height and then to the position entered as center of the second hole or second stud **2**.
- 6 Then the control moves the touch probe to the entered measuring height and probes (the number of touch points to be probed depends on **Q423** Number of Probes) the second hole or stud center point.
- 7 Finally, the control retracts the touch probe to the clearance height (depending on **Q1125**) and saves the measured values in the following Q parameters:

Q parameter number	Meaning
Q950 to Q952	Measured circle center point 1 in the main axis, secondary axis, and tool axis
Q953 to Q955	Measured circle center point 2 in the main axis, secondary axis, and tool axis
Q964	Measured basic rotation
Q965	Measured table rotation
Q966 to Q967	Measured first and second diameters
Q980 to Q982	Measured deviations of circle center point 1
Q983 to Q985	Measured deviations of circle center point 2
Q994	Measured angle deviation of basic rotation
Q995	Measured angle deviation of table rotation
Q996 to Q997	Measured deviation of the diameters
Q183	Workpiece status <ul style="list-style-type: none"> ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap
Q970	If you have programmed Cycle 1493 EXTRUSION PROBING : Mean value of all deviations from the ideal line of the first circle center point
Q971	If you have programmed Cycle 1493 EXTRUSION PROBING : Mean value of all deviations from the ideal line of the second circle center point
Q973	If you have programmed Cycle 1493 EXTRUSION PROBING : Mean value of all deviations of the diameters of circle 1
Q974	If you have programmed Cycle 1493 EXTRUSION PROBING : Mean value of all deviations of the diameters of circle 2



Operating note:

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

You have the following options:

- If there is no danger of collision, you can press NC start to run the cycle with the values from the dialog. The effective set-up clearance will be reduced to the displayed value for this object only.
- You can cancel the cycle by pressing Cancel.

Notes

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

NOTICE

Danger of collision!

When running touch probe cycles **444** and **14xx**, no coordinate transformations must be active (e.g., Cycles **8 MIRRORING**, **11 SCALING FACTOR**, **26 AXIS-SPECIFIC SCALING**, **TRANS MIRROR**).

- ▶ Reset any coordinate transformations before the cycle call.

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

Note about rotary axes:

When determining the basic rotation in a tilted working plane, keep the following in mind:

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

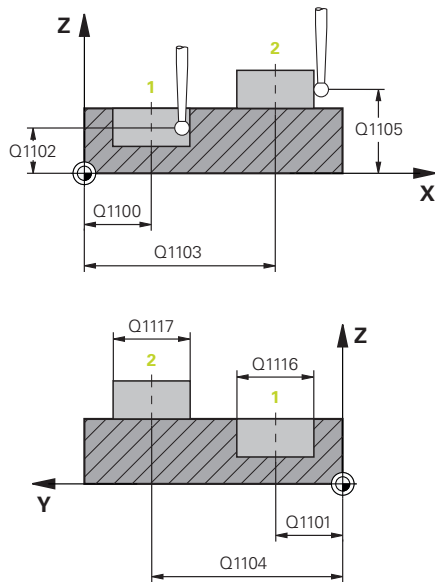
In the optional machine parameter **chkTiltingAxes** (no. 204601), the machine manufacturer defines a check verifying the tilting situation. If no checking is configured, the cycle always assumes that the working plane is consistent. The basic rotation is then calculated in the **I-CS**.

Aligning the rotary table axes:

- The control can align the rotary table only if the measured rotation can be compensated using a rotary table axis. This axis must be the first rotary table axis (as viewed from the workpiece).
- To align the rotary table axes (**Q1126** not equal to 0), you need to adopt the rotation (**Q1121** not equal to 0). Otherwise, the control will display an error message.

4.5.1 Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

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

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

- **?**: Semiautomatic mode, see Page 58
- **-, +**: Evaluation of the tolerance, see Page 64
- **@**: Transferring the actual position, see Page 66

Q1101 1st noml. position of minor axis?

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

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

Q1102 1st nominal position tool axis?

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

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

Q1116 Diameter of 1st position?

Diameter of the first hole or the first stud

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

"...-...+...": Evaluation of the tolerance, see Page 64

Q1103 2nd noml. position of ref axis?

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

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

Q1104 2nd noml. position of minor axis?

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

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

Q1105 2nd nominal pos. of tool axis?

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

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

Help graphic

Parameter

Q1117 Diameter of 2nd position?

Diameter of the second hole or the second stud

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

"...-...+...": Evaluation of the tolerance, see Page 64

Q1115 Geometry type (0-3)?

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

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

Q423 Number of probes?

Number of touch points on the diameter

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

Q325 Starting angle?

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

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

Q1119 Arc angular length?

Angular range in which the touch points are distributed.

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

Q320 Set-up clearance?

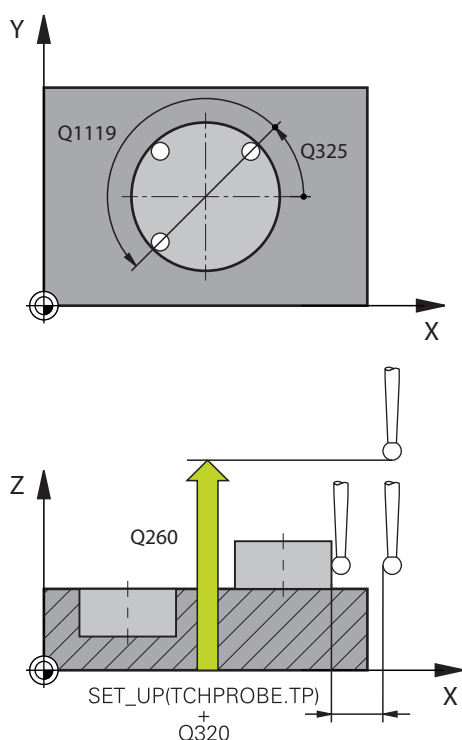
Additional distance between touch point and ball tip. **Q320** is added to **SET_UP** (touch probe table), and is only effective when the preset is probed in the touch probe axis. This value has an incremental effect.

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

Q260 Clearance height?

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

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



Help graphic

Parameter

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

-1: Do not move to clearance height.

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 touch point. Pre-positioning occurs at **FMAX_PROBE**.

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

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

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

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

2: The control opens a window with the results if the actual position is in the scrap range. Program run is interrupted. The control does not open a window with the results if rework is necessary.

Input: **0, 1, 2**

Q1126 Align rotary axes?

Position the rotary axes for inclined machining:

0: Retain the current position of the rotary axis.

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

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

Input: **0, 1, 2**

Q1120 Transfer position?

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

0: No correction

1: Correction based on the 1st touch point

2: Correction based on the 2nd touch point

3: Correction based on the averaged touch point position

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

Help graphic**Parameter****Q1121 CONFIRM ROTATION?**

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

0: No basic rotation

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

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

Input: **0, 1, 2**

Example

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

4.6 Cycle 1412 INCLINED EDGE PROBING

Application

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

If you program Cycle **1493 EXTRUSION PROBING** before this cycle, you can repeat probing points in a given direction over a specified distance.

Further information: "Cycle 1493 EXTRUSION PROBING", Page 276

Cycle **1412** additionally offers the following functions:

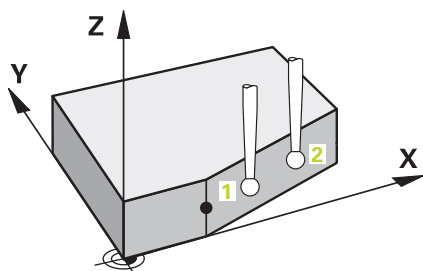
- If the probing position relative to the current datum is unknown, you can run the cycle in semi-automatic mode.

Further information: "Semi-automatic mode", Page 58

- If you determine the actual position in advance, you can then transfer it to the cycle as the actual position.

Further information: "Transferring the actual position", Page 66

Cycle sequence



- 1 Following the positioning logic, the control positions the touch probe to the touch point **1** at rapid traverse **FMAX_PROBE**.

Further information: "Positioning logic", Page 50

- 2 The control then moves the touch probe to set-up clearance **Q320** at rapid traverse **FMAX_PROBE**. The sum of **Q320**, **SET_UP**, and the ball-tip radius is taken into account when probing in any probing direction.
- 3 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate **F** from the touch probe table.
- 4 The control retracts the touch probe by the safety clearance in the direction opposite to the direction of probing.
- 5 If you programmed retraction to clearance height **Q1125**, the touch probe returns to clearance height.
- 6 The touch probe then moves to the touch point **2** and probes again.
- 7 Finally, the control retracts the touch probe to the clearance height (depending on **Q1125**) and saves the measured values in the following Q parameters:

Q parameter number	Meaning
Q950 to Q952	Measured position 1 in the main axis, secondary axis, and tool axis
Q953 to Q955	Measured position 2 in the main axis, secondary axis, and tool axis
Q964	Measured basic rotation
Q965	Measured table rotation
Q980 to Q982	Measured deviations of touch point 1
Q983 to Q985	Measured deviations of touch point 2
Q994	Measured angle deviation of basic rotation
Q995	Measured angle deviation of table rotation
Q183	Workpiece status <ul style="list-style-type: none"> ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap
Q970	If you have programmed Cycle 1493 EXTRUSION PROBING before: Mean value of all deviations from the ideal line of the first touch point
Q971	If you have programmed Cycle 1493 EXTRUSION PROBING before: Mean value of all deviations from the ideal line of the second touch point

Notes

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

NOTICE

Danger of collision!

When running touch probe cycles **444** and **14xx**, no coordinate transformations must be active (e.g., Cycles **8 MIRRORING**, **11 SCALING FACTOR**, **26 AXIS-SPECIFIC SCALING**, **TRANS MIRROR**).

- ▶ Reset any coordinate transformations before the cycle call.

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

Note about rotary axes:

When determining the basic rotation in a tilted working plane, keep the following in mind:

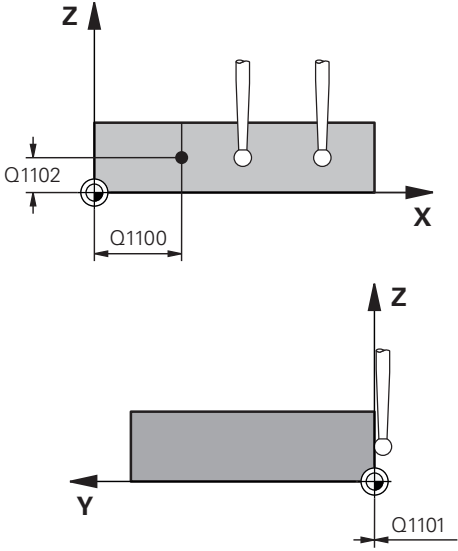
- If the current coordinates of the rotary axes and the defined tilt angles (3-D rotation window) match, the working plane is consistent. The control calculates the basic rotation in the input coordinate system **I-CS**.
- If the current coordinates of the rotary axes and the defined tilt angles (3-D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

In the optional machine parameter **chkTiltingAxes** (no. 204601), the machine manufacturer defines a check verifying the tilting situation. If no checking is configured, the cycle always assumes that the working plane is consistent. The basic rotation is then calculated in the **I-CS**.

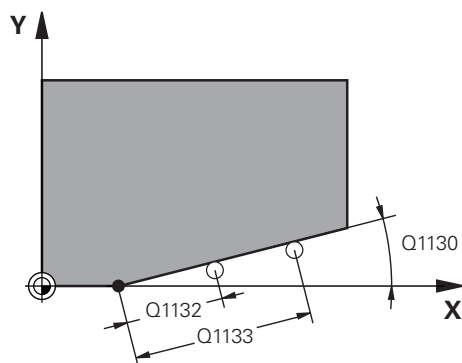
Aligning the rotary table axes:

- The control can align the rotary table only if the measured rotation can be compensated using a rotary table axis. This axis must be the first rotary table axis (as viewed from the workpiece).
- To align the rotary table axes (**Q1126** not equal to 0), you need to adopt the rotation (**Q1121** not equal to 0). Otherwise, the control will display an error message.

4.6.1 Cycle parameters

Help graphic	Parameter
	<p>Q1100 1st noml. position of ref. axis? Absolute nominal position at which the inclined edge begins in the main axis. Input: -99999.9999...+99999.9999 or optionally ?, +, -, @</p> <ul style="list-style-type: none"> ■ ?: Semiautomatic mode, see Page 58 ■ -, +: Evaluation of the tolerance, see Page 64 ■ @: Transferring the actual position, see Page 66
	<p>Q1101 1st noml. position of minor axis? Absolute nominal position at which the inclined edge begins in the secondary axis. Input: -99999.9999...+99999.9999 or optional input (see Q1100)</p>
	<p>Q1102 1st nominal position tool axis? Absolute nominal position of the first touch point in the tool axis Input: -99999.9999...+9999.9999 or optional input (see Q1100)</p>
	<p>QS400 Tolerance value? Tolerance band monitored by the cycle. The tolerance defines the deviation permitted for the surface normals along the inclined edge. This deviation is determined between the nominal coordinate and the actual coordinate of the workpiece. Examples:</p> <ul style="list-style-type: none"> ■ QS400 ="0.4-0.1": Upper dimension = nominal coordinate +0.4; lower dimension = nominal coordinate -0.1. The following tolerance band thus results for the cycle: "nominal coordinate +0.4" to "nominal coordinate - 0.1" ■ QS400 =" ": No tolerance band. ■ QS400 ="0": No tolerance band. ■ QS400 ="0.1+0.1": No tolerance band. <p>Input: Max. 255 characters</p>

Help graphic



Parameter

Q1130 Nominal angle for 1st line?

Nominal angle of the first straight line

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

Q1131 Probing direction for 1st line?

Probing direction for the first straight line:

+1: The control rotates the probing direction by $+90^\circ$ with respect to the nominal angle **Q1130**

-1: The control rotates the probing direction by -90° with respect to the nominal angle **Q1130**

Input: **-1, +1**

Q1132 First distance on 1st line?

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

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

Q1133 Second distance on 1st line?

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

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

Q1139 Plane for object (1-3)?

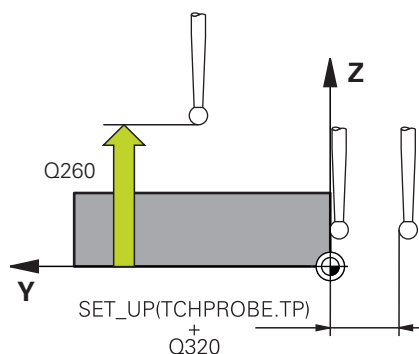
Plane in which the control construes the nominal angle **Q1130** and the probing direction **Q1131**.

1: The nominal angle is in the YZ plane.

2: The nominal angle is in the ZX plane.

3: The nominal angle is in the XY plane.

Input: **1, 2, 3**

**Q320 Set-up clearance?**

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

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

Q260 Clearance height?

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

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

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

-1: Do not move to clearance height.

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 touch point. Pre-positioning occurs at **FMAX_PROBE**.

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

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

Reaction when tolerance is exceeded:

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

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

2: The control opens a window with the results if the actual position is in the scrap range. Program run is interrupted. The control does not open a window with the results if rework is necessary.

Input: **0, 1, 2**

Q1126 Align rotary axes?

Position the rotary axes for inclined machining:

0: Retain the current position of the rotary axis.

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

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

Input: **0, 1, 2**

Q1120 Transfer position?

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

0: No correction

1: Correction based on the 1st touch point

2: Correction based on the 2nd touch point

3: Correction based on the averaged touch point position

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

Q1121 CONFIRM ROTATION?

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

0: No basic rotation

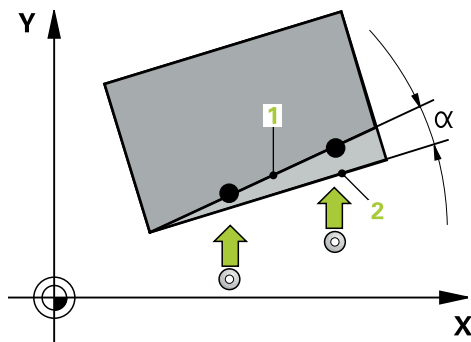
1: Set basic rotation: The control transfers the misalignment to the preset table as basic transformations.

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

Input: **0, 1, 2**

Example

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

4.7 Touch probe cycles 4xx: fundamentals**4.7.1 Characteristics common to all touch probe cycles for measuring workpiece misalignment**

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



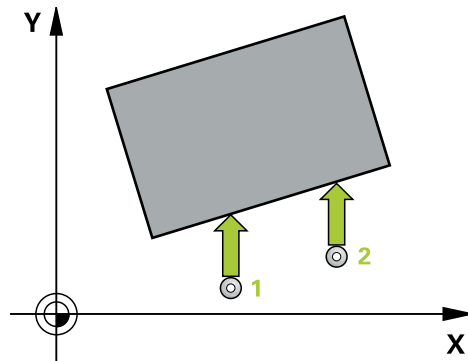
These cycles do not work with 3-D ROT! In such a case, use Cycles **14xx**. **Further information:** "Fundamentals of touch probe cycles 14xx", Page 57

4.8 Cycle 400 BASIC ROTATION

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 sequence



- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) to the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the defined traverse direction
Further information: "Positioning logic", Page 50
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 The touch probe then moves to the next touch point **2** and probes again.
- 4 The control returns the touch probe to the clearance height and performs the basic rotation it determined.

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, 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.

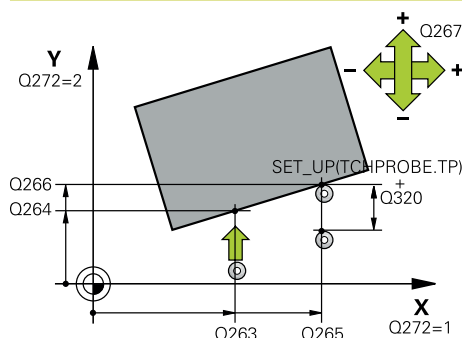
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

4.8.1 Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

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

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

Q264 1st measuring point in 2nd axis?

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

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

Q265 2nd measuring point in 1st axis?

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

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

Q266 2nd measuring point in 2nd axis?

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

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

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

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

1: Main axis = measuring axis

2: Secondary axis = measuring axis

Input: **1, 2**

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

Direction in which the touch probe will approach the workpiece:

-1: Negative traverse direction

+1: Positive traverse direction

Input: **-1, +1**

Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

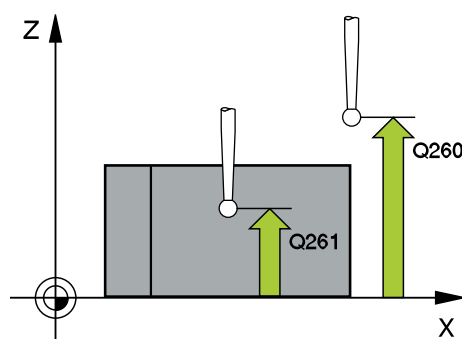
Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

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

Q260 Clearance height?

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

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



Help graphic	Parameter
	<p>Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: 0, 1</p>
	<p>Q307 Preset value for rotation angle If the misalignment will be measured against a straight line other than the main axis, enter the angle of this reference line. For the basic rotation, the control will then calculate the difference between the value measured and the angle of the reference line. The value has an absolute effect. Input: -360.000...+360.000</p>
	<p>Q305 Preset number in table? Specify the number of the row in the preset table in which the control will save the calculated basic rotation. If you enter Q305 = 0, the control automatically stores the calculated basic rotation in the ROT menu of the Manual Operation mode. Input: 0...99999</p>

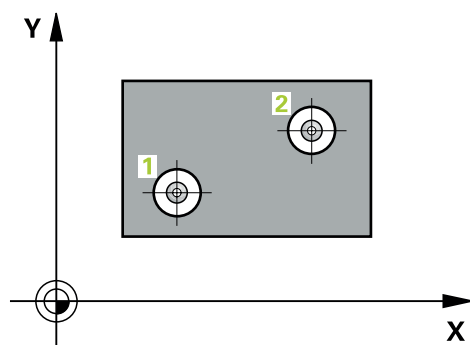
Example

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

4.9 Cycle 401 ROT OF 2 HOLES

Application

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

Cycle sequence

- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) to the programmed center point of the first hole **1**.
- Further information:** "Positioning logic", Page 50
- 2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
- 3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
- 4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
- 5 Then the control returns the touch probe to the clearance height and performs the basic rotation it determined.

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

When running touch probe cycles **400** to **499**, 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.

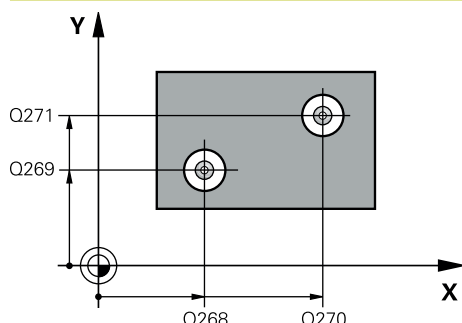
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.
- If you want to compensate the misalignment by rotating the rotary table, the control will automatically use the following rotary axes:
 -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

Note on programming

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

4.9.1 Cycle parameters

Help graphic



Parameter

Q268 1st hole: center in 1st axis?

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

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

Q269 1st hole: center in 2nd axis?

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

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

Q270 2nd hole: center in 1st axis?

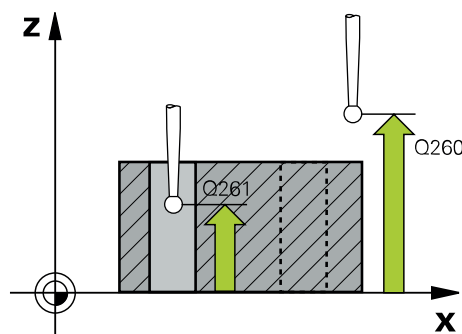
Center of the second hole in the main axis of the working plane. The value has an absolute effect.

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

Q271 2nd hole: center in 2nd axis?

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

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



Q261 Measuring height in probe axis?

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

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

Q260 Clearance height?

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

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

Q307 Preset value for rotation angle

If the misalignment will be measured against a straight line other than the main axis, enter the angle of this reference line. For the basic rotation, the control will then calculate the difference between the value measured and the angle of the reference line. The value has an absolute effect.

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

Help graphic

Parameter

Q305 Number in table?

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

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

Q305 depends on the following parameters:

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

Input: **0...99999**

Q402 Basic rotation/alignment (0/1)

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

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

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

Input: **0, 1**

Q337 Set to zero after alignment?

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

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

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

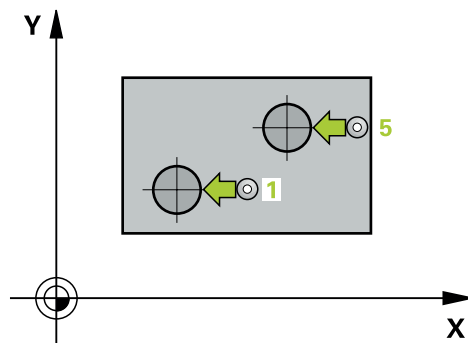
Input: **0, 1**

Example

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

4.10 Cycle 402 ROT OF 2 STUDS**Application**

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

Cycle sequence

- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from FMAX column) to touch point **1** of the first stud.
Further information: "Positioning logic", Page 50
- 2 Then the touch probe moves to the entered **measuring height 1** and probes four points to find the center of the first stud. The touch probe moves along a circular arc between the touch points, each of which is offset by 90°.
- 3 The touch probe returns to the clearance height and then moves to the touch point **5** of the second stud.
- 4 The control moves the touch probe to the entered **measuring height 2** and probes four points to determine the center of the second stud.
- 5 Then the control returns the touch probe to the clearance height and performs the calculated basic rotation.

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, 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.
- The control will reset an active basic rotation at the beginning of the cycle.
- If you want to compensate the misalignment by rotating the rotary table, the control will automatically use the following rotary axes:
 - C for tool axis Z
 - B for tool axis Y
 - A for tool axis X

Note on programming

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

4.10.1 Cycle parameters

Help graphic

Parameter

Q268 1st stud: center in 1st axis?

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

Input: -99999.9999...+99999.9999

Q269 1st stud: center in 2nd axis?

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

Input: -99999.9999...+99999.9999

Q313 Diameter of stud 1?

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

Input: 0...99999.9999

Q261 Meas. height stud 1 in TS axis?

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

Input: -99999.9999...+99999.9999

Q270 2nd stud: center in 1st axis?

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

Input: -99999.9999...+99999.9999

Q271 2nd stud: center in 2nd axis?

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

Input: -99999.9999...+99999.9999

Q314 Diameter of stud 2?

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

Input: 0...99999.9999

Q315 Meas. height stud 2 in TS axis?

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

Input: -99999.9999...+99999.9999

Q320 Set-up clearance?

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

Input: 0...99999.9999 or PREDEF

Q260 Clearance height?

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

Input: -99999.9999...+99999.9999 or PREDEF

Help graphic

Parameter

Q301 Move to clearance height (0/1)?

Specify how the touch probe moves between measuring points:

0: Move at measuring height between measuring points

1: Move at clearance height between measuring points

Input: **0, 1**

Q307 Preset value for rotation angle

If the misalignment will be measured against a straight line other than the main axis, enter the angle of this reference line. For the basic rotation, the control will then calculate the difference between the value measured and the angle of the reference line. The value has an absolute effect.

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

Q305 Number in table?

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

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

Q305 depends on the following parameters:

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

Input: **0...99999**

Help graphic

Parameter

Q402 Basic rotation/alignment (0/1)

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

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

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

Input: **0, 1**

Q337 Set to zero after alignment?

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

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

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

Input: **0, 1**

Example

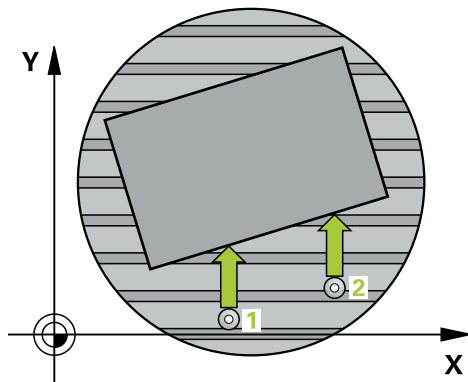
11 TCH PROBE 402 ROT OF 2 STUDS ~	
Q268=-37	;1ST CENTER 1ST AXIS ~
Q269=+12	;1ST CENTER 2ND AXIS ~
Q313=+60	;DIAMETER OF STUD 1 ~
Q261=-5	;MEAS. HEIGHT STUD 1 ~
Q270=+75	;2ND CENTER 1ST AXIS ~
Q271=+20	;2ND CENTER 2ND AXIS ~
Q314=+60	;DIAMETER OF STUD 2 ~
Q315=-5	;MEAS. HEIGHT STUD 2 ~
Q320=+0	;SET-UP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

4.11 Cycle 403 ROT IN ROTARY AXIS

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 sequence



- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the defined traverse direction

Further information: "Positioning logic", Page 50

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

Notes

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

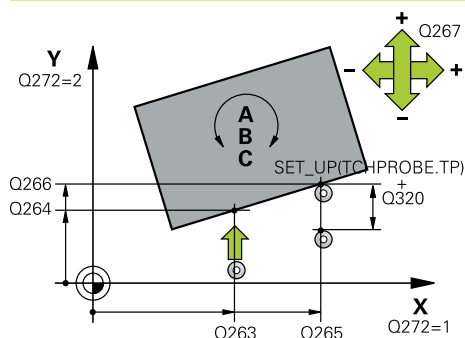
When running touch probe cycles **400** to **499**, no cycles for coordinate transformation must be active.

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

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

4.11.1 Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

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

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

Q264 1st measuring point in 2nd axis?

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

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

Q265 2nd measuring point in 1st axis?

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

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

Q266 2nd measuring point in 2nd axis?

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

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

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

Axis in which the measurement will be made:

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

Input: **1, 2, 3**

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

Direction in which the touch probe will approach the workpiece:

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

Input: **-1, +1**

Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

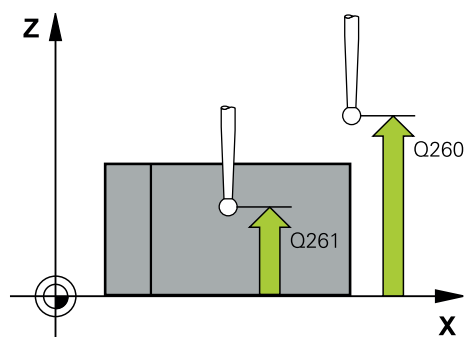
Additional distance between touch point and ball tip. **Q320** is in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

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

Q260 Clearance height?

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

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



Help graphic	Parameter
	<p>Q301 Move to clearance height (0/1)?</p> <p>Specify how the touch probe moves between measuring points:</p> <p>0: Move at measuring height between measuring points</p> <p>1: Move at clearance height between measuring points</p> <p>Input: 0, 1</p>
	<p>Q312 Axis for compensating movement?</p> <p>Define the rotary axis in which the control will compensate the measured misalignment:</p> <p>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!</p> <p>4: Compensate misalignment with rotary axis A</p> <p>5: Compensate misalignment with rotary axis B</p> <p>6: Compensate misalignment with rotary axis C</p> <p>Input: 0, 4, 5, 6</p>
	<p>Q337 Set to zero after alignment?</p> <p>Define whether the control will set the angle of the aligned rotary axis to 0 in the preset table or in the datum table after the alignment.</p> <p>0: Do not set the angle of the rotary axis to 0 in the table after the alignment</p> <p>1: Set the angle of the rotary axis to 0 in the table after the alignment</p> <p>Input: 0, 1</p>

Help graphic	Parameter
	<p>Q305 Number in table?</p> <p>Specify the number of the row in the preset table in which the control will enter the basic rotation.</p> <p>Q305 = 0: The rotary axis is zeroed in row number 0 of the preset table. The control will make an entry in the OFFSET column. In addition, all other values (X, Y, Z, etc.) of the currently active preset will be transferred to row 0 of the preset table. In addition, the control activates the preset from row 0.</p> <p>Q305 > 0: Specify the number of the row in the preset table in which the control will zero the rotary axis. The control will make an entry in the OFFSET column of the preset table.</p> <p>Q305 depends on the following parameters:</p> <ul style="list-style-type: none"> ■ Q337 = 0: Parameter Q305 is not effective ■ Q337 = 1: Parameter Q305 has the effect described above ■ Q312 = 0: Parameter Q305 has the effect described above ■ Q305 Number in table? 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. <p>Input: 0...99999</p>

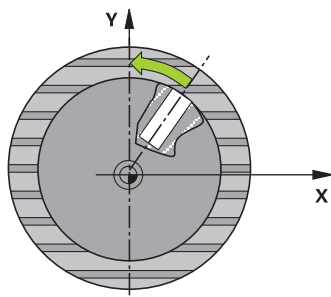
Help graphic	Parameter
	<p>Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: 0: Write the calculated preset to the active datum table as a datum shift. The reference system is the active workpiece coordinate system. 1: Write the calculated preset to the preset table. Input: 0, 1</p>
	<p>Q380 Ref. angle in ref. axis? Angle to which the control will align the probed straight line. Only effective if the rotary axis is in automatic mode or if C is selected (Q312 = 0 or 6). Input: 0...360</p>

Example

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

4.12 Cycle 405 ROT IN C AXIS

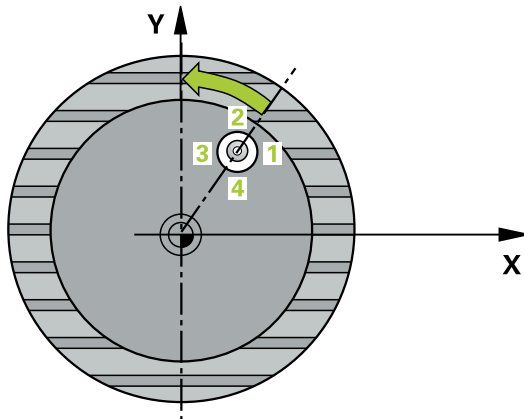
Application



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 sequence

- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
Further information: "Positioning logic", Page 50
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
- 3 Then, the touch probe moves along a circular arc, either at measuring height or at clearance height, to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times and then positions the touch probe on the calculated hole center.
- 5 Finally, the control returns the touch probe to the clearance height and aligns the workpiece by rotating the rotary table. The control rotates the rotary table in such a way that the hole center, after compensation, lies in the direction of the positive Y axis or at the nominal position of the hole center point—both with a vertical and a horizontal touch probe axis. The measured angular offset is also available in the parameter **Q150**.

Notes

NOTICE

Danger of collision!

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

- ▶ 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.

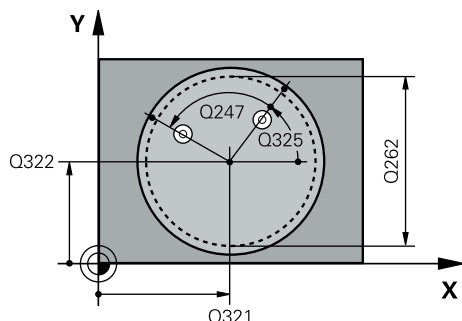
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

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

4.12.1 Cycle parameters

Help graphic



Parameter

Q321 Center in 1st axis?

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

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

Q322 Center in 2nd axis?

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

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

Q262 Nominal diameter?

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

Input: **0...99999.9999**

Q325 Starting angle?

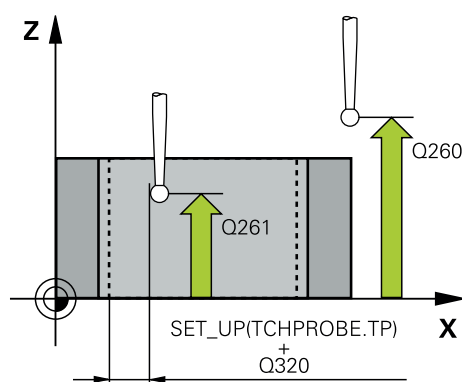
Angle between the main axis of the working plane and the first touch point. The value has an absolute effect.

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

Q247 Intermediate stepping angle?

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

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



Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Help graphic

Parameter

Q301 Move to clearance height (0/1)?

Specify how the touch probe moves between measuring points:

0: Move at measuring height between measuring points

1: Move at clearance height between measuring points

Input: **0, 1**

Q337 Set to zero after alignment?

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

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

Input: **0...2999**

Example

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

4.13 Cycle 404 SET BASIC ROTATION

Application

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

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, 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.

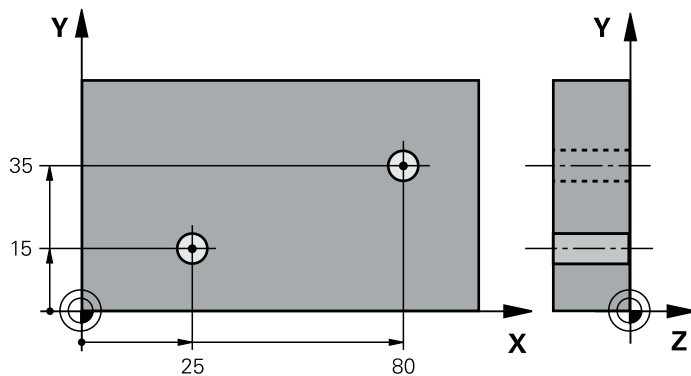
4.13.1 Cycle parameters

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

Example

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

4.14 Example: Determining a basic rotation from two holes



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

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

5

**Touch Probe Cycles:
Automatic Preset
Measurement**

5.1 Overview

The control offers cycles for automatic preset measurement.



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.

Cycle	Call	Further information
1400 POSITION PROBING <ul style="list-style-type: none"> ■ Measurement of single position ■ Definition of preset, if necessary 	DEF- active	Page 121
1401 CIRCLE PROBING <ul style="list-style-type: none"> ■ Measurement of points on the inside or outside of a circle ■ Definition of circle center as preset, if necessary 	DEF- active	Page 125
1402 SPHERE PROBING <ul style="list-style-type: none"> ■ Measurement of points on a sphere ■ Definition of sphere center as preset, if necessary 	DEF- active	Page 129
410 PRESET INSIDE RECTAN <ul style="list-style-type: none"> ■ Measurement of inside length and width of a rectangle ■ Definition of rectangle center as preset 	DEF- active	Page 136
411 PRESET OUTS. RECTAN <ul style="list-style-type: none"> ■ Measurement of outside length and width of a rectangle ■ Definition of rectangle center as preset 	DEF- active	Page 141
412 PRESET INSIDE CIRCLE <ul style="list-style-type: none"> ■ Measurement of any four points on the inside of a circle ■ Definition of circle center as preset 	DEF- active	Page 147
413 PRESET OUTS. CIRCLE <ul style="list-style-type: none"> ■ Measurement of any four points on the outside of a circle ■ Definition of circle center as preset 	DEF- active	Page 153
414 PRESET OUTS. CORNER <ul style="list-style-type: none"> ■ Measurement of two straight lines on the outside ■ Definition of intersection of the lines as preset 	DEF- active	Page 159
415 PRESET INSIDE CORNER <ul style="list-style-type: none"> ■ Measurement of two straight lines on the inside ■ Definition of intersection of the lines as preset 	DEF- active	Page 164
416 PRESET CIRCLE CENTER <ul style="list-style-type: none"> ■ Measurement of any three holes on a circular hole pattern ■ Definition of center of circular hole pattern as preset 	DEF- active	Page 170

Cycle	Call	Further information
417 PRESET IN TS AXIS <ul style="list-style-type: none"> ■ Measurement of any position in the tool axis ■ Definition of any position as preset 	DEF- active	Page 177
418 PRESET FROM 4 HOLES <ul style="list-style-type: none"> ■ Measurement of two holes on each line crosswise ■ Definition of the intersection of the lines as preset 	DEF- active	Page 181
419 PRESET IN ONE AXIS <ul style="list-style-type: none"> ■ Measurement of any position in a selectable axis ■ Definition of any position in a selectable axis as preset 	DEF- active	Page 186
408 SLOT CENTER PRESET <ul style="list-style-type: none"> ■ Measurement of width of an inside slot ■ Definition of slot center as preset 	DEF- active	Page 189
409 RIDGE CENTER PRESET <ul style="list-style-type: none"> ■ Measurement of width of an outside ridge ■ Definition of ridge center as preset 	DEF- active	Page 194

5.2 Fundamentals of touch probe cycles 14xx for presetting

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

Preset and tool axis

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

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

Measurement results in Q parameters

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

5.3 Cycle 1400 POSITION PROBING

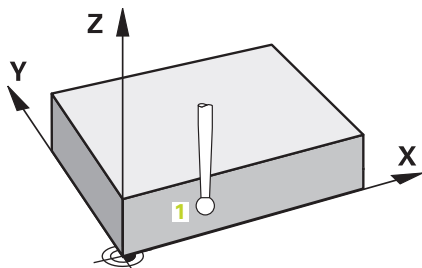
Application

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

If you program Cycle **1493 EXTRUSION PROBING** before this cycle, you can repeat probing points in a given direction over a specified distance.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 276

Cycle sequence



- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the programmed touch point **1**. The control takes into account the set-up clearance **Q320** during pre-positioning.
Further information: "Positioning logic", Page 50
- 2 Then the touch probe moves to the entered measuring height and measures the actual position with a single probing movement.
- 3 The control returns the touch probe to the clearance height.
- 4 The control saves the measured position in the following Q parameters. If **Q1120 = 1**, then the control writes the measured position to the active row of the preset table.

Further information: "Fundamentals of touch probe cycles 14xx for presetting", Page 121

Q parameter number	Meaning
Q950 to Q952	Measured position 1 in the main axis, secondary axis, and tool axis
Q980 to Q982	Measured deviations of touch point 1
Q183	Workpiece status <ul style="list-style-type: none"> ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap
Q970	If you have programmed Cycle 1493 EXTRUSION PROBING : Mean value of all deviations from the ideal line of the second touch point

Notes

NOTICE

Danger of collision!

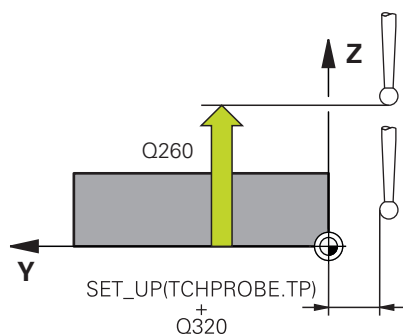
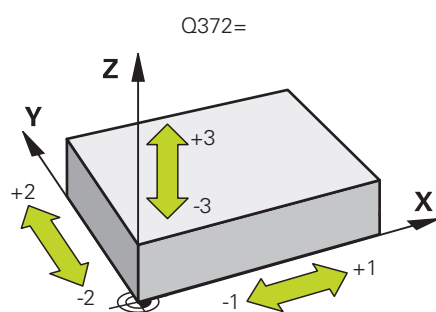
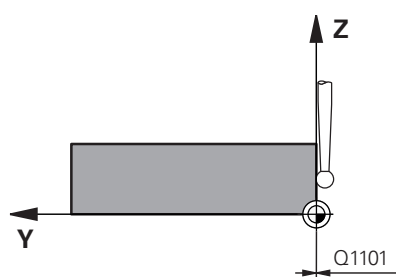
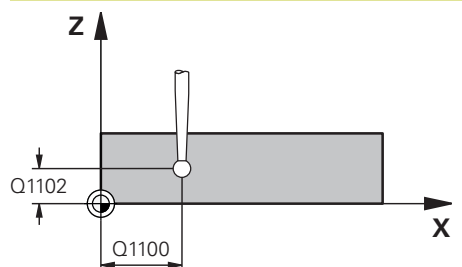
When running touch probe cycles **444** and **14xx**, no coordinate transformations must be active (e.g., Cycles **8 MIRRORING**, **11 SCALING FACTOR**, **26 AXIS-SPECIFIC SCALING**, **TRANS MIRROR**).

- Reset any coordinate transformations before the cycle call.

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

5.3.1 Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

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

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

?: Semiautomatic mode, see Page 58

-, +: Evaluation of the tolerance, see Page 64

@: Transferring the actual position, see Page 66

Q1101 1st noml. position of minor axis?

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

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

Q1102 1st nominal position tool axis?

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

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

Q372 Probe direction (-3 to +3)?

Axis defining the direction of probing. With the algebraic sign, you define the positive or negative direction of traverse of the probing axis.

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Help graphic

Parameter

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

-1: Do not move to clearance height.

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

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

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

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

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

2: The control opens a window with the results if the actual position is in the scrap range. Program run is interrupted. The control does not open a window with the results if rework is necessary.

Input: **0, 1, 2**

Q1120 Transfer position?

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

0: No correction

1: Correction based on the 1st touch point

Input: **0, 1**

Example

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

5.4 Cycle 1401 CIRCLE PROBING

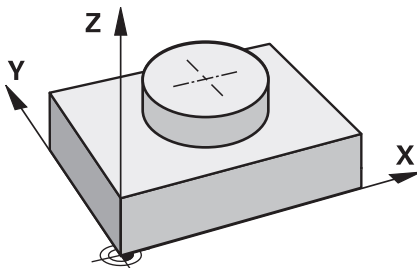
Application

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

If you program Cycle **1493 EXTRUSION PROBING** before this cycle, you can repeat probing points in a given direction over a specified distance.

Further information: "Cycle 1493 EXTRUSION PROBING ", Page 276

Cycle sequence



- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) to the programmed touch point. The control takes the set-up clearance **Q320** into account during pre-positioning.
Further information: "Positioning logic", Page 50
- 2 Then the touch probe moves to the entered measuring height **Q1102** and measures the actual position of the first touch point.
- 3 The control returns the touch probe to the clearance height **Q260** at **FMAX_PROBE** and then moves it to the next touch point.
- 4 The control moves the touch probe to the entered measuring height **Q1102** and measures the next touch point.
- 5 Depending on the definition of **Q423 NO. OF PROBE POINTS**, the control repeats the steps 3 to 4.
- 6 The control returns the touch probe to the clearance height **Q260**.
- 7 The control saves the measured position in the following Q parameters. If **Q1120 = 1**, then the control writes the measured position to the active row of the preset table.

Further information: "Fundamentals of touch probe cycles 14xx for presetting", Page 121

Q parameter number	Meaning
Q950 to Q952	Measured circle center point in the main axis, secondary axis, and tool axis
Q966	Measured diameter
Q980 to Q982	Measured deviations of the circle center point
Q996	Measured deviation of the diameter
Q183	Workpiece status <ul style="list-style-type: none"> ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap
Q970	If you have programmed Cycle 1493 EXTRUSION PROBING : Mean value of all deviations from the ideal line of the first circle center point
Q973	If you have programmed Cycle 1493 EXTRUSION PROBING : Mean value of all deviations of the diameters of circle 1

Notes

NOTICE

Danger of collision!

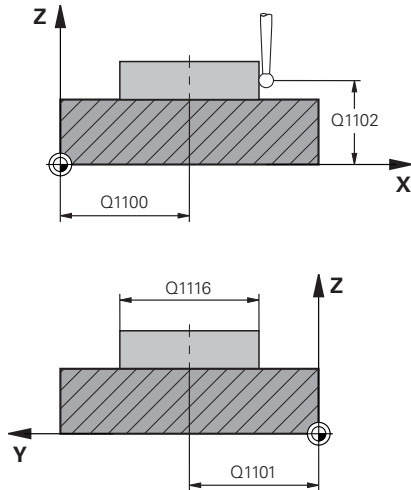
When running touch probe cycles **444** and **14xx**, no coordinate transformations must be active (e.g., Cycles **8 MIRRORING**, **11 SCALING FACTOR**, **26 AXIS-SPECIFIC SCALING**, **TRANS MIRROR**).

- ▶ Reset any coordinate transformations before the cycle call.

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

5.4.1 Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

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

Input: **-99999.9999...+99999.9999** or optional input:

"?...": Semiautomatic mode, see Page 58

"...-...+...": Evaluation of the tolerance, see Page 64

"...@...": Transferring the actual position, see Page 66

Q1101 1st noml. position of minor axis?

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

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

Q1102 1st nominal position tool axis?

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

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

Q1116 Diameter of 1st position?

Diameter of the first hole or the first stud

"...-...+...": Evaluation of the tolerance, see Page 64

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

Q1115 Geometry type (0/1)?

Geometry of the object:

0: Hole

1: Stud

Input: **0, 1**

Q423 Number of probes?

Number of touch points on the diameter

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

Q325 Starting angle?

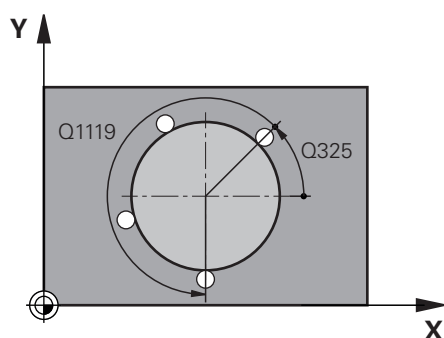
Angle between the main axis of the working plane and the first touch point. The value has an absolute effect.

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

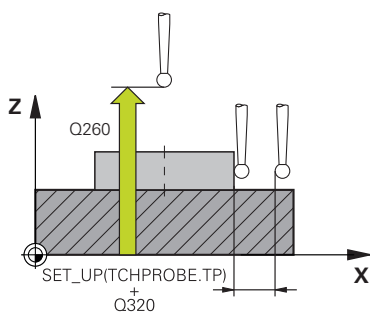
Q1119 Arc angular length?

Angular range in which the touch points are distributed.

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



Help graphic



Parameter

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q1125 Traverse to clearance height?

Positioning behavior between the touch points

-1: Do not move to clearance height.

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

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

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

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

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

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

2: The control opens a window with the results if the actual position is in the scrap range. Program run is interrupted. The control does not open a window with the results if rework is necessary.

Input: **0, 1, 2**

Q1120 Transfer position?

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

0: No correction

1: Correction based on the 1st touch point

Input: **0, 1**

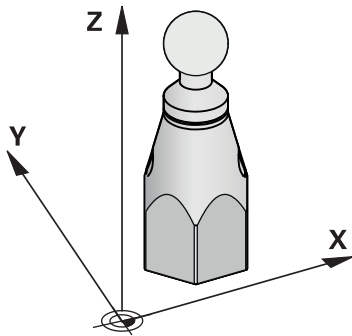
Example

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

5.5 Cycle 1402 SPHERE PROBING

Application

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

Cycle sequence

- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) to the programmed touch point. The control takes the set-up clearance **Q320** into account during pre-positioning.
- Further information:** "Positioning logic", Page 50
- 2 Then the touch probe moves to the entered measuring height **Q1102** and measures the actual position of the first touch point with a single probing movement.
- 3 The control returns the touch probe to the clearance height **Q260** at **FMAX_PROBE** and then moves it to the next touch point.
- 4 The control moves the touch probe to the entered measuring height **Q1102** and measures the next touch point.
- 5 Depending on the definition of **Q423** Number of Probes, the control repeats the steps 3 to 4.
- 6 The control moves the touch probe in the tool axis by the set-up clearance to a position above the sphere.
- 7 The touch probe moves to the center of the sphere and probes another touch point.
- 8 The touch probe returns to the clearance height **Q260**.
- 9 The control saves the measured position in the following Q parameters. If **Q1120 = 1**, then the control writes the measured position to the active row of the preset table.

Further information: "Fundamentals of touch probe cycles 14xx for presetting", Page 121

Q parameter number	Meaning
Q950 to Q952	Measured circle center point in the main axis, secondary axis, and tool axis
Q966	Measured diameter
Q980 to Q982	Measured deviations of the circle center point
Q996	Measured deviations of the diameter
Q183	Workpiece status <ul style="list-style-type: none"> ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap

Notes

NOTICE

Danger of collision!

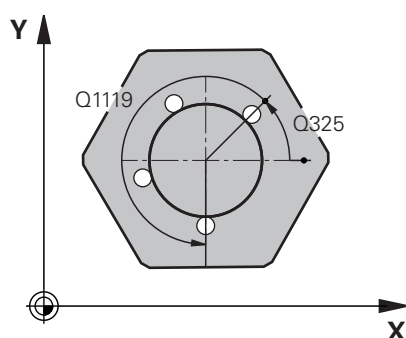
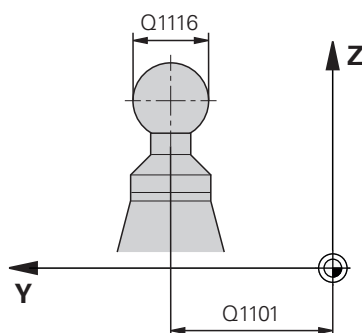
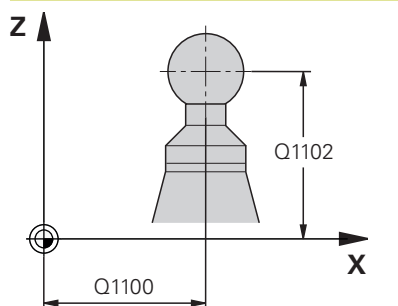
When running touch probe cycles **444** and **14xx**, no coordinate transformations must be active (e.g., Cycles **8 MIRRORING**, **11 SCALING FACTOR**, **26 AXIS-SPECIFIC SCALING**, **TRANS MIRROR**).

- ▶ Reset any coordinate transformations before the cycle call.

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

5.5.1 Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

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

Input: **-99999.9999...+99999.9999** or optional input:

"?...": Semiautomatic mode, see Page 58

"...-...+...": Evaluation of the tolerance, see Page 64

"...@...": Transferring the actual position, see Page 66

Q1101 1st noml. position of minor axis?

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

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

Q1102 1st nominal position tool axis?

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

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

Q1116 Diameter of 1st position?

Diameter of the sphere

"...-...+...": Evaluation of the tolerance, see Page 64

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

Q423 Number of probes?

Number of touch points on the diameter

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

Q325 Starting angle?

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

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

Q1119 Arc angular length?

Angular range in which the touch points are distributed.

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

Q320 Set-up clearance?

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

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

Help graphic	Parameter
	<p>Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: -99999.9999...+99999.9999 or PREDEF</p>
	<p>Q1125 Traverse to clearance height? Positioning behavior between the touch points -1: Do not move to clearance height. 0, 1: Move to clearance height before and after the cycle. Pre-positioning occurs at FMAX_PROBE. 2: Move to clearance height before and after each touch point. Pre-positioning occurs at FMAX_PROBE. Input: -1, 0, +1, +2</p>
	<p>Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: 0: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. 1: Interrupt program run when tolerance is exceeded. The control opens a window with the results. 2: The control opens a window with the results if the actual position is in the scrap range. Program run is interrupted. The control does not open a window with the results if rework is necessary. Input: 0, 1, 2</p>
	<p>Q1120 Transfer position? Define which touch point will be used to correct the active preset: 0: No correction 1: Correction based on center of the sphere Input: 0, 1</p>

Example

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

5.6 Fundamentals of touch probe cycles 4xx for preset setting

5.6.1 Characteristics common to all touch probe cycles 4xx for preset setting



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.

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

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

Preset and touch probe axis

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

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

Saving the calculated preset

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

- **Q305 = 0, Q303 = 1:**
The control copies the active preset to row 0, changes it and activates row 0, deleting simple transformations.
- **Q305 not equal to 0, Q303 = 0:**
The result is written to the datum table, row **Q305**; **activate the datum with TRANS DATUM in the NC program**
Further information: User's Manual for Programming and Testing
- **Q305 not equal to 0, Q303 = 1:**
The result is written to the preset table, row **Q305**; **use Cycle 247 to activate the preset in the NC program**
- **Q305 not equal to 0, Q303 = -1**



This combination can only occur if you

- read in NC programs (containing Cycles **410** to **418**) that were created on a TNC 4xx
- read in NC programs (containing Cycles **410** to **418**) that were created with an older software version of an iTNC 530
- did not specifically define the measured-value transfer with parameter **Q303** when defining the cycle

In these cases, the control outputs an error message, since the complete handling of REF-referenced datum tables has changed. You must define a measured-value transfer yourself with parameter **Q303**.

Measurement results in Q parameters

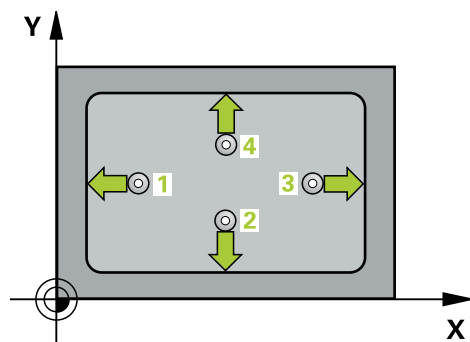
The control saves the measurement results of the respective probing cycle in the globally effective Q parameters **Q150** to **Q160**. You can use these parameters in your NC program. Note the table of result parameters listed with every cycle description.

5.7 Cycle 410 PRESET INSIDE RECTAN

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 sequence



- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
Further information: "Positioning logic", Page 50
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 The control returns the touch probe to the clearance height.
- 6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, see "Fundamentals of touch probe cycles 4xx for preset setting", Page 134
- 7 Then the control saves the actual values in the Q parameters listed below.
- 8 If desired, the control subsequently determines the preset in the touch probe axis in a separate probing operation.

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

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, 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

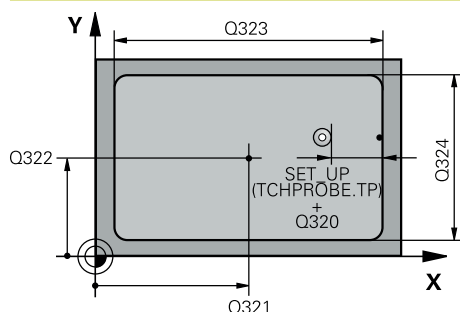
Danger of collision!

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

5.7.1 Cycle parameters

Help graphic



Parameter

Q321 Center in 1st axis?

Center of the pocket in the main axis of the working plane. The value has an absolute effect.

Input: -99999.9999...+99999.9999

Q322 Center in 2nd axis?

Center of the pocket in the secondary axis of the working plane. The value has an absolute effect.

Input: -99999.9999...+99999.9999

Q323 First side length?

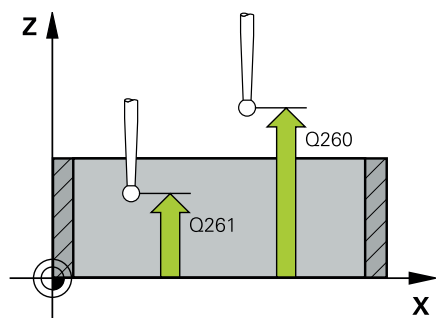
Pocket length, parallel to the main axis of the working plane. This value has an incremental effect.

Input: 0...99999.9999

Q324 Second side length?

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

Input: 0...99999.9999



Q261 Measuring height in probe axis?

The value has an absolute effect. Input: -99999.9999...+99999.9999

Input: -99999.9999...+99999.9999

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q301 Move to clearance height (0/1)?

Specify how the touch probe moves between measuring points:

0: Move at measuring height between measuring points

1: Move at clearance height between measuring points

Input: 0, 1

Help graphic	Parameter
	<p>Q305 Number in table?</p> <p>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.</p> <p>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.</p> <p>If Q303 = 0, the control will write the data to the datum table. The datum is not automatically activated.</p> <p>Further information: "Saving the calculated preset", Page 135</p> <p>Input: 0...99999</p>
	<p>Q331 New preset in reference axis?</p> <p>Coordinate in the main axis at which the control will set the calculated pocket center. Default setting = 0. The value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q332 New preset in minor axis?</p> <p>Coordinate in the secondary axis at which the control will set the calculated pocket center. Default setting = 0. The value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q303 Meas. value transfer (0,1)?</p> <p>Define whether the calculated preset will be saved in the datum table or in the preset table:</p> <p>-1: Do not use. Is entered by the control when old NC programs are read in, see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 134</p> <p>0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system.</p> <p>1: Write the calculated preset to the preset table.</p> <p>Input: -1, 0, +1</p>
	<p>Q381 Probe in TS axis? (0/1)</p> <p>Define whether the control will also set the preset in the touch probe axis:</p> <p>0: Do not set the preset in the touch probe axis</p> <p>1: Set the preset in the touch probe axis</p> <p>Input: 0, 1</p>

Help graphic	Parameter
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect. Input: -99999.9999...+99999.9999

Example

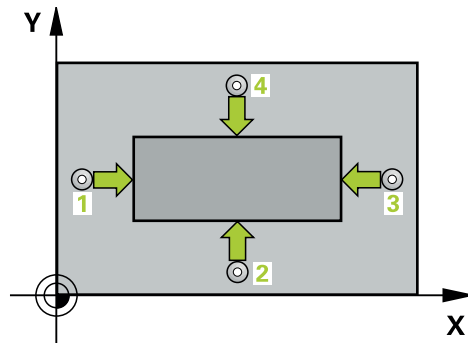
11 CYCL DEF 410 PRESET INSIDE RECTAN ~	
Q321=+50	;CENTER IN 1ST AXIS ~
Q322=+50	;CENTER IN 2ND AXIS ~
Q323=+60	;FIRST SIDE LENGTH ~
Q324=+20	;2ND SIDE LENGTH ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q305=+10	;NUMBER IN TABLE ~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

5.8 Cycle 411 PRESET OUTS. RECTAN

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 sequence



- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
Further information: "Positioning logic", Page 50
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 The control returns the touch probe to the clearance height.
- 6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, see "Fundamentals of touch probe cycles 4xx for preset setting", Page 134
- 7 Then the control saves the actual values in the Q parameters listed below.
- 8 If desired, the control subsequently determines the preset in the touch probe axis in a separate probing operation.

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

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, 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!

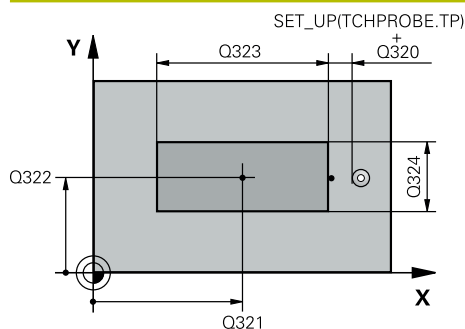
To prevent a collision between touch probe and workpiece, enter **high** estimates for the lengths of the 1st and 2nd sides.

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

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

5.8.1 Cycle parameters

Help graphic



Parameter

Q321 Center in 1st axis?

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

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

Q322 Center in 2nd axis?

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

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

Q323 First side length?

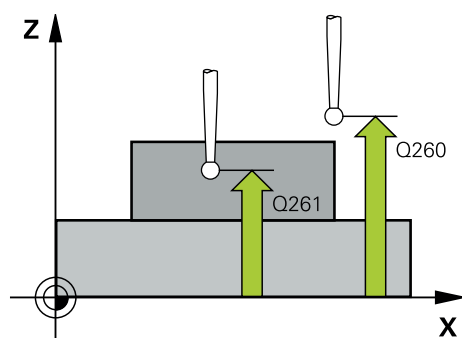
Length of stud parallel to the main axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q324 Second side length?

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

Input: **0...99999.9999**



Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q301 Move to clearance height (0/1)?

Specify how the touch probe moves between measuring points:

0: Move at measuring height between measuring points

1: Move at clearance height between measuring points

Input: **0, 1**

Help graphic	Parameter
	<p>Q305 Number in table?</p> <p>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.</p> <p>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.</p> <p>If Q303 = 0, the control will write the data to the datum table. The datum is not automatically activated.</p> <p>Further information: "Saving the calculated preset", Page 135</p> <p>Input: 0...99999</p>
	<p>Q331 New preset in reference axis?</p> <p>Coordinate in the main axis at which the control will set the calculated stud center. Default setting = 0. The value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q332 New preset in minor axis?</p> <p>Coordinate in the secondary axis at which the control will set the calculated stud center. Default setting = 0. The value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q303 Meas. value transfer (0,1)?</p> <p>Define whether the calculated preset will be saved in the datum table or in the preset table:</p> <p>-1: Do not use. Is entered by the control when old NC programs are read in, see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 134</p> <p>0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system.</p> <p>1: Write the calculated preset to the preset table.</p> <p>Input: -1, 0, +1</p>

Help graphic	Parameter
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: 0: Do not set the preset in the touch probe axis 1: Set the preset in the touch probe axis Input: 0, 1
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect. Input: -99999.9999...+99999.9999

Example

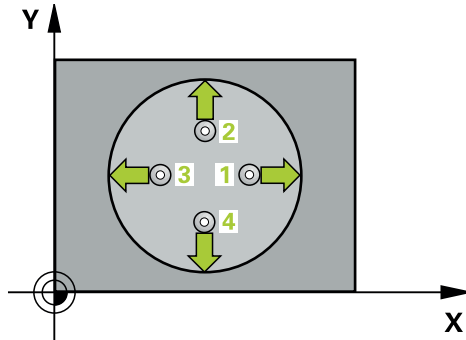
11 TCH PROBE 411 PRESET OUTS. RECTAN ~	
Q321=+50	;CENTER IN 1ST AXIS ~
Q322=+50	;CENTER IN 2ND AXIS ~
Q323=+60	;FIRST SIDE LENGTH ~
Q324=+20	;2ND SIDE LENGTH ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q305=+0	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+85	;1ST CO. FOR TS AXIS ~
Q383=+50	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+1	;PRESET

5.9 Cycle 412 PRESET INSIDE CIRCLE

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 sequence



- Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
Further information: "Positioning logic", Page 50
- 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.
- Then, the touch probe moves in a circular arc either at measuring height or linearly at clearance height to the next touch point **2** and probes again.
- The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- The control returns the touch probe to the clearance height.
- Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, see "Fundamentals of touch probe cycles 4xx for preset setting", Page 134
- Then the control saves the actual values in the Q parameters listed below.
- If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

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

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, 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.

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

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

Notes on programming

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



Program the stepping angle to be less than 90°

5.9.1 Cycle parameters

Help graphic

Parameter

Q321 Center in 1st axis?

Center of the pocket in the main axis of the working plane. The value has an absolute effect.

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

Q322 Center in 2nd axis?

Center of the pocket in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point to the positive Y axis. If you program **Q322** not equal to 0, then the control aligns the hole center point to the nominal position. The value has an absolute effect.

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

Q262 Nominal diameter?

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

Input: **0...99999.9999**

Q325 Starting angle?

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

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

Q247 Intermediate stepping angle?

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

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

Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Help graphic	Parameter
	<p>Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: 0, 1</p>
	<p>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, 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 will write the data to the datum table. The datum is not automatically activated. Further information: "Saving the calculated preset", Page 135 Input: 0...99999</p>
	<p>Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated pocket center. Default setting = 0. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated pocket center. Default setting = 0. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: -1: Do not use. Is entered by the control when old NC programs are read in, see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 134 0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. 1: Write the calculated preset to the preset table. Input: -1, 0, +1</p>

Help graphic	Parameter
	<p>Q381 Probe in TS axis? (0/1)</p> <p>Define whether the control will also set the preset in the touch probe axis:</p> <p>0: Do not set the preset in the touch probe axis</p> <p>1: Set the preset in the touch probe axis</p> <p>Input: 0, 1</p>
	<p>Q382 Probe TS axis: Coord. 1st axis?</p> <p>Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q383 Probe TS axis: Coord. 2nd axis?</p> <p>Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q384 Probe TS axis: Coord. 3rd axis?</p> <p>Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q333 New preset in TS axis?</p> <p>Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q423 No. probe points in plane (4/3)?</p> <p>Define whether the control will use three or four touch points to measure the circle:</p> <p>3: Use three measuring points</p> <p>4: Use four measuring points (default setting)</p> <p>Input: 3, 4</p>
	<p>Q365 Type of traverse? Line=0/arc=1</p> <p>Specify the path function to be used by the tool for moving between the measuring points if "traverse to clearance height" (Q301 = 1) is active.</p> <p>0: Move in a straight line between machining operations</p> <p>1: Move along a circular arc on the pitch circle diameter between machining operations</p> <p>Input: 0, 1</p>

Example

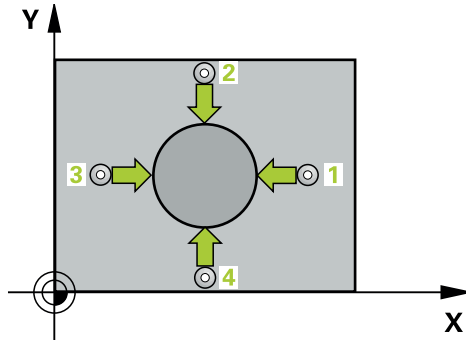
11 TCH PROBE 412 PRESET INSIDE CIRCLE ~	
Q321=+50	;CENTER IN 1ST AXIS ~
Q322=+50	;CENTER IN 2ND AXIS ~
Q262=+75	;NOMINAL DIAMETER ~
Q325=+0	;STARTING ANGLE ~
Q247=+60	;STEPPING ANGLE ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q305=+12	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+85	;1ST CO. FOR TS AXIS ~
Q383=+50	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+1	;PRESET ~
Q423=+4	;NO. OF PROBE POINTS ~
Q365=+1	;TYPE OF TRAVERSE

5.10 Cycle 413 PRESET OUTS. CIRCLE

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 sequence



- Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
Further information: "Positioning logic", Page 50
- 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.
- 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.
- The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- The control returns the touch probe to the clearance height.
- Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, see "Fundamentals of touch probe cycles 4xx for preset setting", Page 134
- Then the control saves the actual values in the Q parameters listed below.
- If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

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

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, 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.

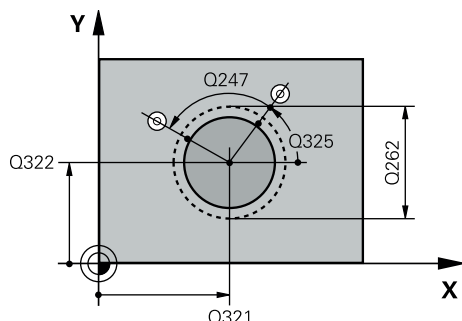
- The control will reset an active basic rotation at the beginning of the cycle.
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The smaller the stepping angle **Q247**, the less accurately the control can calculate the preset. Minimum input value: 5°



Program the stepping angle to be less than 90°

5.10.1 Cycle parameters

Help graphic



Parameter

Q321 Center in 1st axis?

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

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

Q322 Center in 2nd axis?

Center of the stud in the secondary axis of the working plane. If you program **Q322** = 0, the control aligns the hole center point to the positive Y axis. If you program **Q322** not equal to 0, then the control aligns the hole center point to the nominal position. The value has an absolute effect.

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

Q262 Nominal diameter?

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

Input: **0...99999.9999**

Q325 Starting angle?

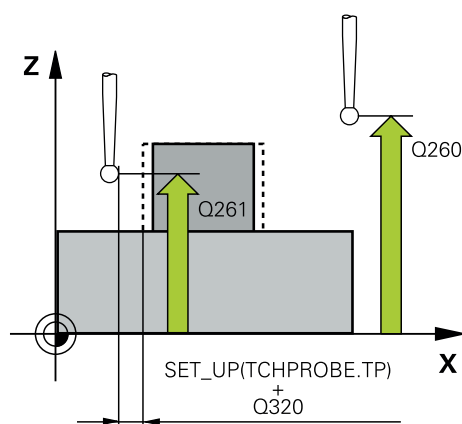
Angle between the main axis of the working plane and the first touch point. The value has an absolute effect.

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

Q247 Intermediate stepping angle?

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

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



Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Help graphic	Parameter
	<p>Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: 0, 1</p>
	<p>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, 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 will write the data to the datum table. The datum is not automatically activated. Further information: "Saving the calculated preset", Page 135 Input: 0...99999</p>
	<p>Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated stud center. Default setting = 0. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated stud center. Default setting = 0. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: -1: Do not use. Is entered by the control when old NC programs are read in, see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 134 0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. 1: Write the calculated preset to the preset table. Input: -1, 0, +1</p>

Help graphic	Parameter
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: 0: Do not set the preset in the touch probe axis 1: Set the preset in the touch probe axis Input: 0, 1
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1 . The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1 . The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1 . The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q423 No. probe points in plane (4/3)? Define whether the control will use three or four touch points to measure the circle: 3: Use three measuring points 4: Use four measuring points (default setting) Input: 3, 4
	Q365 Type of traverse? Line=0/arc=1 Specify the path function to be used by the tool for moving between the measuring points if "traverse to clearance height" (Q301 = 1) is active. 0: Move in a straight line between machining operations 1: Move along a circular arc on the pitch circle diameter between machining operations Input: 0, 1

Example

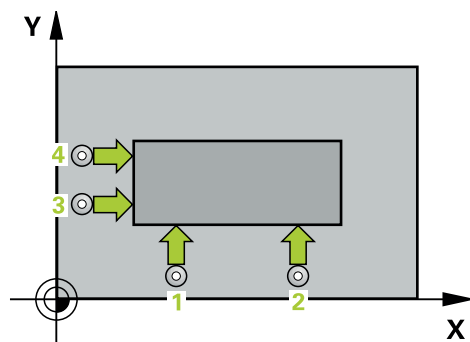
11 TCH PROBE 413 PRESET OUTS. CIRCLE ~	
Q321=+50	;CENTER IN 1ST AXIS ~
Q322=+50	;CENTER IN 2ND AXIS ~
Q262=+75	;NOMINAL DIAMETER ~
Q325=+0	;STARTING ANGLE ~
Q247=+60	;STEPPING ANGLE ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q305=+15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

5.11 Cycle 414 PRESET OUTS. CORNER

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 sequence



- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) at touch point **1** (see figure). The control offsets the touch probe by the set-up clearance in the direction opposite the respective traverse direction.

Further information: "Positioning logic", Page 50

- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the 3rd measuring point.
- 3 The touch probe then moves to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 The control returns the touch probe to the clearance height.
- 6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, see "Fundamentals of touch probe cycles 4xx for preset setting", Page 134
- 7 Then the control saves the coordinates of the calculated corner 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.

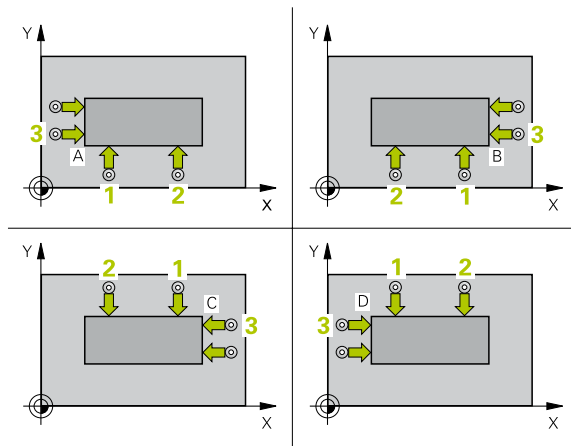


The control always measures the first line in the direction of the minor axis of the working plane.

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

Definition of the corner

By defining the positions of the measuring points **1** and **3**, you also determine the corner at which the control sets the preset (see the following figure and table below).



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

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

When running touch probe cycles **400** to **499**, 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.

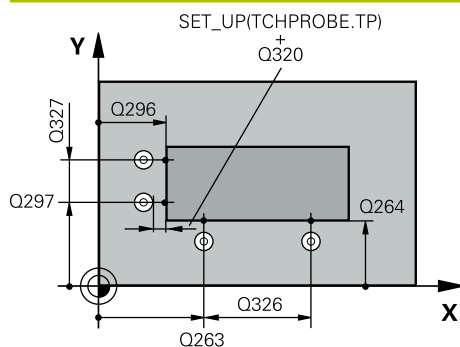
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

5.11.1 Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

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

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

Q264 1st measuring point in 2nd axis?

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

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

Q326 Spacing in 1st axis?

Distance between the first and second measuring points in the main axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q296 3rd measuring point in 1st axis?

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

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

Q297 3rd measuring point in 2nd axis?

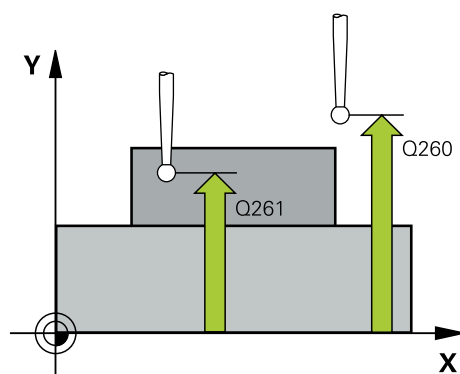
Coordinate of the third touch point in the secondary axis of the working plane. The value has an absolute effect.

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

Q327 Spacing in 2nd axis?

Distance between third and fourth measuring points in the secondary axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**



Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Help graphic	Parameter
	<p>Q260 Clearance height?</p> <p>Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999 or PREDEF</p>
	<p>Q301 Move to clearance height (0/1)?</p> <p>Specify how the touch probe moves between measuring points:</p> <p>0: Move at measuring height between measuring points 1: Move at clearance height between measuring points</p> <p>Input: 0, 1</p>
	<p>Q304 Execute basic rotation (0/1)?</p> <p>Define whether the control will compensate workpiece misalignment with a basic rotation:</p> <p>0: No basic rotation 1: Basic rotation</p> <p>Input: 0, 1</p>
	<p>Q305 Number in table?</p> <p>Indicate the number of the row of the preset table or datum table, in which the control saves the corner coordinates. Depending on Q303, the control writes the entry to the preset table or datum table:</p> <p>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.</p> <p>If Q303 = 0, the control will write the data to the datum table. The datum is not automatically activated.</p> <p>Further information: "Saving the calculated preset", Page 135</p> <p>Input: 0...99999</p>
	<p>Q331 New preset in reference axis?</p> <p>Coordinate in the main axis at which the control will set the calculated corner. Default setting = 0. The value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q332 New preset in minor axis?</p> <p>Coordinate in the secondary axis at which the control will set the calculated corner. Default setting = 0. The value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>

Help graphic	Parameter
	<p>Q303 Meas. value transfer (0,1)?</p> <p>Define whether the calculated preset will be saved in the datum table or in the preset table:</p> <p>-1: Do not use. Is entered by the control when old NC programs are read in, see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 134</p> <p>0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system.</p> <p>1: Write the calculated preset to the preset table.</p> <p>Input: -1, 0, +1</p>
	<p>Q381 Probe in TS axis? (0/1)</p> <p>Define whether the control will also set the preset in the touch probe axis:</p> <p>0: Do not set the preset in the touch probe axis</p> <p>1: Set the preset in the touch probe axis</p> <p>Input: 0, 1</p>
	<p>Q382 Probe TS axis: Coord. 1st axis?</p> <p>Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q383 Probe TS axis: Coord. 2nd axis?</p> <p>Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q384 Probe TS axis: Coord. 3rd axis?</p> <p>Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q333 New preset in TS axis?</p> <p>Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>

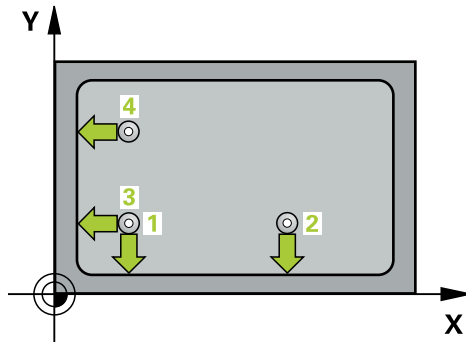
Example

11 TCH PROBE 414 PRESET OUTS. CORNER ~	
Q263=+37	;1ST POINT 1ST AXIS ~
Q264=+7	;1ST POINT 2ND AXIS ~
Q326=+50	;SPACING IN 1ST AXIS ~
Q296=+95	;3RD PNT IN 1ST AXIS ~
Q297=+25	;3RD PNT IN 2ND AXIS ~
Q327=+45	;SPACING IN 2ND AXIS ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q304=+0	;BASIC ROTATION ~
Q305=+7	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+85	;1ST CO. FOR TS AXIS ~
Q383=+50	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+1	;PRESET

5.12 Cycle 415 PRESET INSIDE CORNER

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 sequence

- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) at touch point **1** (see figure). The control offsets the touch probe in the main axis and the secondary axis by the set-up clearance **Q320 + SET_UP** + ball-tip radius (in the direction opposite the respective traverse direction)
- Further information:** "Positioning logic", Page 50
- 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 main axis by the set-up clearance **Q320 + SET_UP** + ball-tip radius and then performs the fourth probing operation
- 6 The control returns the touch probe to the clearance height.
- 7 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, see "Fundamentals of touch probe cycles 4xx for preset setting", Page 134
- 8 Then the control saves the coordinates of the calculated corner in the Q parameters listed below.
- 9 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.



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

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

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, 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.

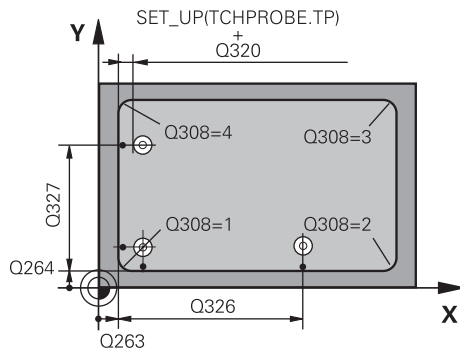
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

5.12.1 Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

Coordinate of the corner in the main axis of the working plane. The value has an absolute effect.

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

Q264 1st measuring point in 2nd axis?

Coordinate of the corner in the secondary axis of the working plane. The value has an absolute effect.

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

Q326 Spacing in 1st axis?

Distance between the first corner and the second measuring point in the main axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q327 Spacing in 2nd axis?

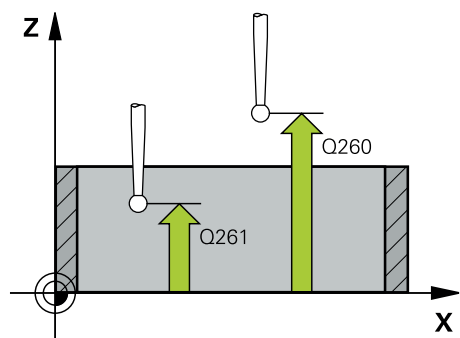
Distance between the corner and the fourth measuring point in the secondary axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q308 Corner? (1/2/3/4)

Number identifying the corner at which the control will set the preset.

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



Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q301 Move to clearance height (0/1)?

Specify how the touch probe moves between measuring points:

0: Move at measuring height between measuring points

1: Move at clearance height between measuring points

Input: **0, 1**

Help graphic	Parameter
	<p>Q304 Execute basic rotation (0/1)? Define whether the control will compensate workpiece misalignment with a basic rotation: 0: No basic rotation 1: Basic rotation Input: 0, 1</p>
	<p>Q305 Number in table? Indicate the number of the row of the preset table or datum table, in which the control saves the corner coordinates. Depending on Q303, the control writes the entry to the preset table or datum table: If Q303 = 1, the control will write the data to the preset table. If 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 will write the data to the datum table. The datum is not automatically activated. Further information: "Saving the calculated preset", Page 135 Input: 0...99999</p>
	<p>Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated corner. Default setting = 0. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated corner. Default setting = 0. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: -1: Do not use. Is entered by the control when old NC programs are read in, see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 134 0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. 1: Write the calculated preset to the preset table. Input: -1, 0, +1</p>

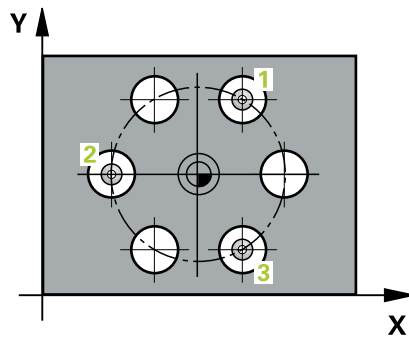
Help graphic	Parameter
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: 0: Do not set the preset in the touch probe axis 1: Set the preset in the touch probe axis Input: 0, 1
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1 . The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1 . The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1 . The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect. Input: -99999.9999...+99999.9999

Example

11 TCH PROBE 415 PRESET INSIDE CORNER ~	
Q263=+37	;1ST POINT 1ST AXIS ~
Q264=+7	;1ST POINT 2ND AXIS ~
Q326=+50	;SPACING IN 1ST AXIS ~
Q327=+45	;SPACING IN 2ND AXIS ~
Q308=+1	;CORNER ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q304=+0	;BASIC ROTATION ~
Q305=+7	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+85	;1ST CO. FOR TS AXIS ~
Q383=+50	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+1	;PRESET

5.13 Cycle 416 PRESET CIRCLE CENTER**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 sequence

- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) to the programmed center point of the first hole **1**.
- Further information:** "Positioning logic", Page 50
- 2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
- 3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
- 4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
- 5 The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
- 6 The control moves the touch probe to the entered measuring height and probes four points to determine the third hole center point.
- 7 The control returns the touch probe to the clearance height.
- 8 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, see "Fundamentals of touch probe cycles 4xx for preset setting", Page 134
- 9 Then the control saves the actual values in the Q parameters listed below.
- 10 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

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

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, 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.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

5.13.1 Cycle parameters

Help graphic	Parameter
	<p>Q273 Center in 1st axis (nom. value)? Bolt hole circle center (nominal value) in the main axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q274 Center in 2nd axis (nom. value)? Bolt hole circle center (nominal value) in the secondary axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q262 Nominal diameter? Enter the approximate bolt hole circle diameter. The smaller the hole diameter, the more exact the nominal diameter must be. Input: 0...99999.9999</p>
	<p>Q291 Polar coord. angle of 1st hole? Polar coordinate angle of the first hole center in the working plane. The value has an absolute effect. Input: -360.000...+360.000</p>
	<p>Q292 Polar coord. angle of 2nd hole? Polar coordinate angle of the second hole center in the working plane. The value has an absolute effect. Input: -360.000...+360.000</p>
	<p>Q293 Polar coord. angle of 3rd hole? Polar coordinate angle of the third hole center in the working plane. The value has an absolute effect. Input: -360.000...+360.000</p>
	<p>Q261 Measuring height in probe axis? Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: -99999.9999...+99999.9999 or PREDEF</p>

Help graphic

Parameter

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**, 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 will write the data to the datum table. The datum is not automatically activated.

Further information: "Saving the calculated preset", Page 135

Input: **0...99999**

Q331 New preset in reference axis?

Coordinate in the main axis at which the control will set the calculated bolt-hole center. Default setting = 0. The value has an absolute effect.

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

Q332 New preset in minor axis?

Coordinate in the secondary axis at which the control will set the calculated bolt-hole circle center. Default setting = 0. The value has an absolute effect.

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

Q303 Meas. value transfer (0,1)?

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

-1: Do not use. Is entered by the control when old NC programs are read in, see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 134

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

1: Write the calculated preset to the preset table.

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

Q381 Probe in TS axis? (0/1)

Define whether the control will also set the preset in the touch probe axis:

0: Do not set the preset in the touch probe axis

1: Set the preset in the touch probe axis

Input: **0, 1**

Help graphic	Parameter
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q320 Set-up clearance? Additional distance between touch point and ball tip. Q320 is added to SET_UP (touch probe table), and is only effective when the preset is probed in the touch probe axis. This value has an incremental effect. Input: 0...99999.9999 or PREDEF

Example

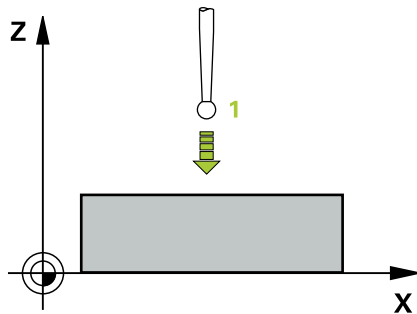
11 TCH PROBE 416 PRESET CIRCLE CENTER ~	
Q273=+50	;CENTER IN 1ST AXIS ~
Q274=+50	;CENTER IN 2ND AXIS ~
Q262=+90	;NOMINAL DIAMETER ~
Q291=+34	;ANGLE OF 1ST HOLE ~
Q292=+70	;ANGLE OF 2ND HOLE ~
Q293=+210	;ANGLE OF 3RD HOLE ~
Q261=-5	;MEASURING HEIGHT ~
Q260=+20	;CLEARANCE HEIGHT ~
Q305=+12	;NUMBER IN TABLE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

5.14 Cycle 417 PRESET IN TS AXIS

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 sequence



- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the direction of the positive touch probe axis.

Further information: "Positioning logic", Page 50

- 2 Then the touch probe moves in its own axis to the coordinate entered as touch point **1** and measures the actual position with a simple probing movement
- 3 The control returns the touch probe to the clearance height.
- 4 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, see "Fundamentals of touch probe cycles 4xx for preset setting", Page 134
- 5 Then the control saves the actual values in the Q parameters listed below.

Q parameter number	Meaning
Q160	Actual value of measured point

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, 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.
- The control sets the preset in this axis.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

5.14.1 Cycle parameters

Help graphic	Parameter
	Q263 1st measuring point in 1st axis? Coordinate of the first touch point in the main axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q294 1st measuring point in 3rd axis? Coordinate of the first touch point in the touch probe axis. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q320 Set-up clearance? Additional distance between touch point and ball tip. Q320 is in addition to the SET_UP column in the touch probe table. This value has an incremental effect. Input: 0...99999.9999 or PREDEF
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: -99999.9999...+99999.9999 or PREDEF
	Q305 Number in table? Indicate the number of the row of the preset table or datum table, in which the control saves the coordinates. Depending on Q303 , the control writes the entry to the preset table or datum table. If Q303 = 1 , the control will write the data to the preset table. If 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 will write the data to the datum table. The datum is not automatically activated. Further information: "Saving the calculated preset", Page 135
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect. Input: -99999.9999...+99999.9999

Help graphic	Parameter
	<p>Q303 Meas. value transfer (0,1)?</p> <p>Define whether the calculated preset will be saved in the datum table or in the preset table:</p> <p>-1: Do not use. Is entered by the control when old NC programs are read in, see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 134</p> <p>0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system.</p> <p>1: Write the calculated preset to the preset table.</p> <p>Input: -1, 0, +1</p>

Example

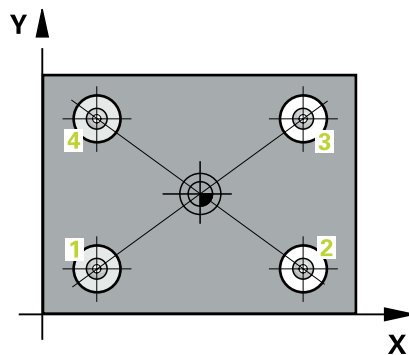
11 TCH PROBE 417 PRESET IN TS AXIS ~	
Q263=+25	;1ST POINT 1ST AXIS ~
Q264=+25	;1ST POINT 2ND AXIS ~
Q294=+25	;1ST POINT 3RD AXIS ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q305=+0	;NUMBER IN TABLE ~
Q333=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER

5.15 Cycle 418 PRESET FROM 4 HOLES

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 sequence



- Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) to the center point of the first hole **1**.
Further information: "Positioning logic", Page 50
- Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
- The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
- The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
- The control repeats this step for holes **3** and **4**.
- The control returns the touch probe to the clearance height.
- Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, see "Fundamentals of touch probe cycles 4xx for preset setting", Page 134
- 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.
- If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

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

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, 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.

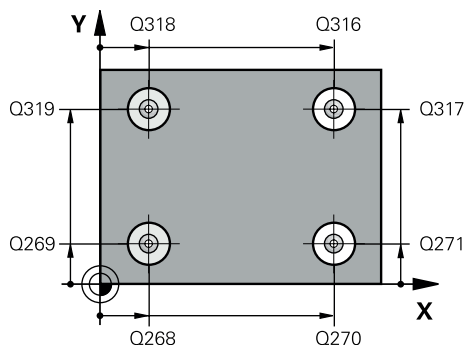
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

5.15.1 Cycle parameters

Help graphic



Parameter

Q268 1st hole: center in 1st axis?

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

Input: -99999.9999...+9999.9999

Q269 1st hole: center in 2nd axis?

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

Input: -99999.9999...+9999.9999

Q270 2nd hole: center in 1st axis?

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

Input: -99999.9999...+9999.9999

Q271 2nd hole: center in 2nd axis?

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

Input: -99999.9999...+9999.9999

Q316 3rd hole: Center in 1st axis?

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

Input: -99999.9999...+9999.9999

Q317 3rd hole: Center in 2nd axis?

Center of the third hole in the secondary axis of the working plane. The value has an absolute effect.

Input: -99999.9999...+9999.9999

Q318 4th hole: Center in 1st axis?

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

Input: -99999.9999...+9999.9999

Q319 4th hole: Center in 2nd axis?

Center of the fourth hole in the secondary axis of the working plane. The value has an absolute effect.

Input: -99999.9999...+9999.9999

Q261 Measuring height in probe axis?

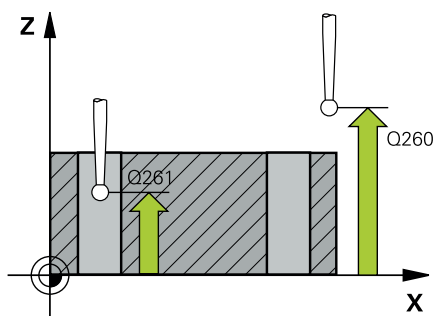
Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect.

Input: -99999.9999...+9999.9999

Q260 Clearance height?

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

Input: -99999.9999...+9999.9999 or PREDEF



Help graphic

Parameter

Q305 Number in table?

Indicate the number of the row in the preset table or datum table in which the control saves the coordinates of the point of intersection of the connecting lines. Depending on **Q303**, the control writes the entry to the preset table or datum table.

If **Q303 = 1**, the control will write the data to the preset table. If 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 will write the data to the datum table. The datum is not automatically activated.

Further information: "Saving the calculated preset", Page 135

Input: **0...99999**

Q331 New preset in reference axis?

Coordinate in the main axis at which the control will set the calculated intersection of the connecting lines. Default setting = 0. The value has an absolute effect.

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

Q332 New preset in minor axis?

Coordinate in the secondary axis at which the control will set the calculated intersection of the connecting lines. Default setting = 0. The value has an absolute effect.

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

Q303 Meas. value transfer (0,1)?

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

-1: Do not use. Is entered by the control when old NC programs are read in, see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 134

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

1: Write the calculated preset to the preset table.

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

Q381 Probe in TS axis? (0/1)

Define whether the control will also set the preset in the touch probe axis:

0: Do not set the preset in the touch probe axis

1: Set the preset in the touch probe axis

Input: **0, 1**

Help graphic	Parameter
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q333 New preset in TS axis? Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect. Input: -99999.9999...+99999.9999

Example

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

5.16 Cycle 419 PRESET IN ONE AXIS

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 sequence

- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the programmed touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the programmed probing direction.

Further information: "Positioning logic", Page 50

- 2 Then the touch probe moves to the programmed measuring height and measures the actual position with a simple probing movement.
- 3 The control returns the touch probe to the clearance height.
- 4 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, see "Fundamentals of touch probe cycles 4xx for preset setting", Page 134

Notes

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.

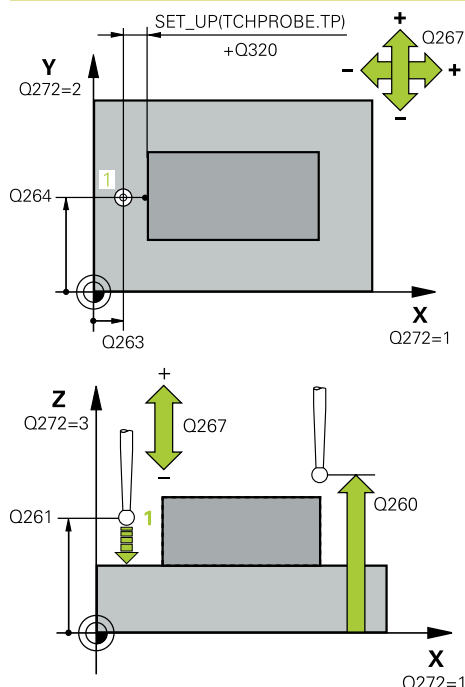
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If you want to save the preset in several axes in the preset table, you can use Cycle **419** several times in a row. However, you also have to reactivate the preset number after every run of Cycle **419**. If you work with preset 0 as active preset, this process is not required.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

5.16.1 Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

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

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

Q264 1st measuring point in 2nd axis?

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

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

Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

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

Axis in which the measurement will be made:

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

Axis assignment

Active touch probe axis: Q272 = 3	Corresponding main axis: Q272 = 1	Corresponding secondary axis: Q272 = 2
Z	X	Y
Y	Z	X
X	Y	Z

Input: **1, 2, 3**

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

Direction in which the touch probe will approach the workpiece:

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

Input: **-1, +1**

Help graphic

Parameter

Q305 Number in table?

Indicate the number of the row of the preset table or datum table, in which the control saves the coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table.

If **Q303 = 1**, the control will write the data to the preset table. If 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 will write the data to the datum table. The datum is not automatically activated.

Further information: "Saving the calculated preset", Page 135

Q333 New preset?

Coordinate at which the control will set the preset. Default setting = 0. The value has an absolute effect.

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

Q303 Meas. value transfer (0,1)?

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

-1: Do not use. Is entered by the control when old NC programs are read in, see "Characteristics common to all touch probe cycles 4xx for preset setting", Page 134

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

1: Write the calculated preset to the preset table.

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

Example

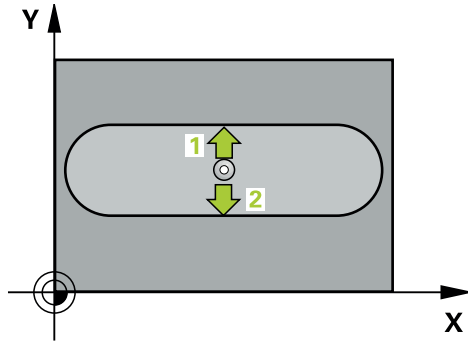
11 TCH PROBE 419 PRESET IN ONE AXIS ~	
Q263=+25	;1ST POINT 1ST AXIS ~
Q264=+25	;1ST POINT 2ND AXIS ~
Q261=+25	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q272=+1	;MEASURING AXIS ~
Q267=+1	;TRAVERSE DIRECTION ~
Q305=+0	;NUMBER IN TABLE ~
Q333=+0	;PRESET ~
Q303=+1	;MEAS.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

5.17 Cycle 408 SLOT CENTER PRESET

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 sequence



- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
Further information: "Positioning logic", Page 50
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control returns the touch probe to the clearance height.
- 5 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, see "Fundamentals of touch probe cycles 4xx for preset setting", Page 134
- 6 Then the control saves the actual values in the Q parameters listed below.
- 7 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

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

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, 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!

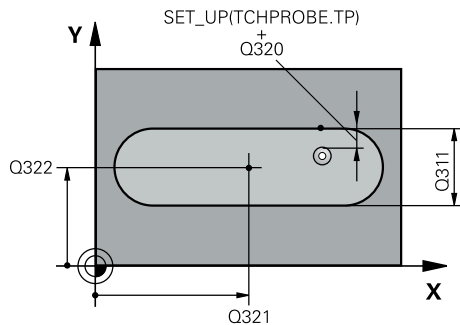
To prevent a collision between touch probe and workpiece, enter a **low** estimate for the slot width. If the slot width and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the slot. In this case, the touch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

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

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

5.17.1 Cycle parameters

Help graphic



Parameter

Q321 Center in 1st axis?

Center of the slot in the main axis of the working plane. The value has an absolute effect.

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

Q322 Center in 2nd axis?

Center of the slot in the secondary axis of the working plane. The value has an absolute effect.

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

Q311 Width of slot?

Width of the slot, regardless of its position in the working plane. This value has an incremental effect.

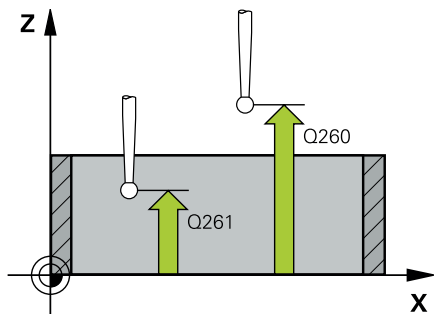
Input: **0...99999.9999**

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

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

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

Input: **1, 2**



Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q301 Move to clearance height (0/1)?

Specify how the touch probe moves between measuring points:

- 0: Move at measuring height between measuring points
- 1: Move at clearance height between measuring points

Input: **0, 1**

Help graphic	Parameter
	<p>Q305 Number in table?</p> <p>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.</p> <p>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.</p> <p>If Q303 = 0, the control will write the data to the datum table. The datum is not automatically activated.</p> <p>Further information: "Saving the calculated preset", Page 135</p> <p>Input: 0...99999</p>
	<p>Q405 New preset?</p> <p>Coordinate in the measuring axis at which the control will set the calculated slot center. Default setting = 0. The value has an absolute effect.</p> <p>Input: -99999.9999...+9999.9999</p>
	<p>Q303 Meas. value transfer (0,1)?</p> <p>Define whether the calculated preset will be saved in the datum table or in the preset table:</p> <p>0: Write the calculated preset to the active datum table as a datum shift. The reference system is the active workpiece coordinate system.</p> <p>1: Write the calculated preset to the preset table.</p> <p>Input: 0, 1</p>
	<p>Q381 Probe in TS axis? (0/1)</p> <p>Define whether the control will also set the preset in the touch probe axis:</p> <p>0: Do not set the preset in the touch probe axis</p> <p>1: Set the preset in the touch probe axis</p> <p>Input: 0, 1</p>
	<p>Q382 Probe TS axis: Coord. 1st axis?</p> <p>Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>

Help graphic

Parameter

Q383 Probe TS axis: Coord. 2nd axis?

Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect.

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

Q384 Probe TS axis: Coord. 3rd axis?

Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect.

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

Q333 New preset in TS axis?

Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect.

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

Example

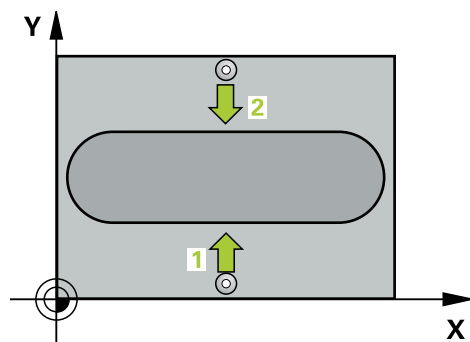
11 TCH PROBE 408 SLOT CENTER PRESET ~	
Q321=+50	;CENTER IN 1ST AXIS ~
Q322=+50	;CENTER IN 2ND AXIS ~
Q311=+25	;SLOT WIDTH ~
Q272=+1	;MEASURING AXIS ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q305=+10	;NUMBER IN TABLE ~
Q405=+0	;PRESET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

5.18 Cycle 409 RIDGE CENTER PRESET

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 sequence



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

Further information: "Positioning logic", Page 50

- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves at clearance height to the next touch point **2** and probes it.
- 4 The control returns the touch probe to the clearance height.
- 5 Depending on the cycle parameters **Q303** and **Q305**, the control processes the calculated preset, see "Fundamentals of touch probe cycles 4xx for preset setting", Page 134
- 6 Then the control saves the actual values in the Q parameters listed below.
- 7 If desired, the control subsequently measures the preset in the touch probe axis in a separate probing operation.

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

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, 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

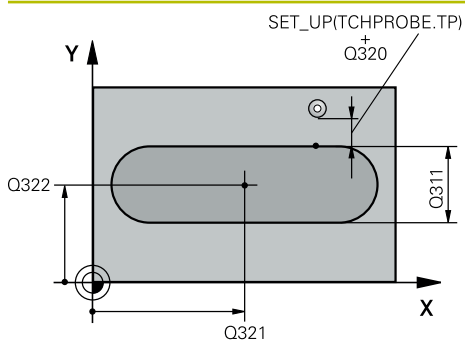
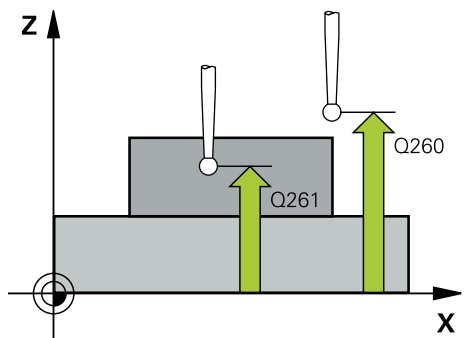
Danger of collision!

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.
- The control will reset an active basic rotation at the beginning of the cycle.

5.18.1 Cycle parameters

Help graphic	Parameter
	Q321 Center in 1st axis? Center of the ridge in the main axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q322 Center in 2nd axis? Center of the ridge in the secondary axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q311 Ridge width? Width of the ridge, regardless of its position in the working plane. This value has an incremental effect. Input: 0...99999.9999
	Q272 Measuring axis (1=1st / 2=2nd)? Axis in the working plane in which the measurement will be performed: 1: Main axis = measuring axis 2: Secondary axis = measuring axis Input: 1, 2
	Q261 Measuring height in probe axis? Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q320 Set-up clearance? Additional distance between touch point and ball tip. Q320 is in addition to the SET_UP column in the touch probe table. This value has an incremental effect. Input: 0...99999.9999 or PREDEF
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: -99999.9999...+99999.9999 or PREDEF

Help graphic	Parameter
	<p>Q305 Number in table?</p> <p>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.</p> <p>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.</p> <p>If Q303 = 0, the control will write the data to the datum table. The datum is not automatically activated.</p> <p>Further information: "Saving the calculated preset", Page 135</p> <p>Input: 0...99999</p>
	<p>Q405 New preset?</p> <p>Coordinate in the measuring axis at which the control will set the calculated ridge center. Default setting = 0. The value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q303 Meas. value transfer (0,1)?</p> <p>Define whether the calculated preset will be saved in the datum table or in the preset table:</p> <p>0: Write the calculated preset to the active datum table as a datum shift. The reference system is the active workpiece coordinate system.</p> <p>1: Write the calculated preset to the preset table.</p> <p>Input: 0, 1</p>
	<p>Q381 Probe in TS axis? (0/1)</p> <p>Define whether the control will also set the preset in the touch probe axis:</p> <p>0: Do not set the preset in the touch probe axis</p> <p>1: Set the preset in the touch probe axis</p> <p>Input: 0, 1</p>
	<p>Q382 Probe TS axis: Coord. 1st axis?</p> <p>Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. The value has an absolute effect.</p> <p>Input: -99999.9999...+99999.9999</p>

Help graphic

Parameter

Q383 Probe TS axis: Coord. 2nd axis?

Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect.

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

Q384 Probe TS axis: Coord. 3rd axis?

Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. The value has an absolute effect.

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

Q333 New preset in TS axis?

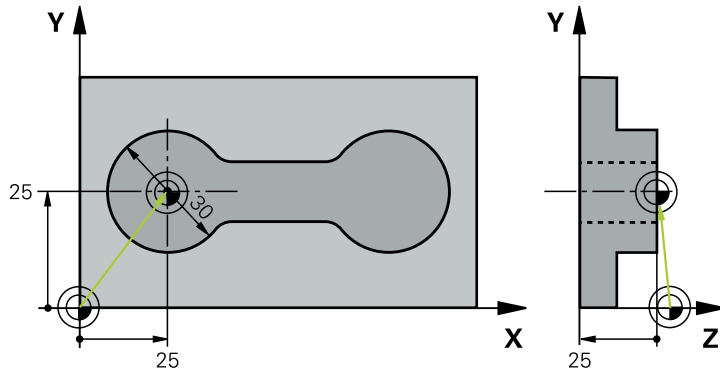
Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. The value has an absolute effect.

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

Example

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

5.19 Example: Presetting at center of a circular segment and on top surface of workpiece

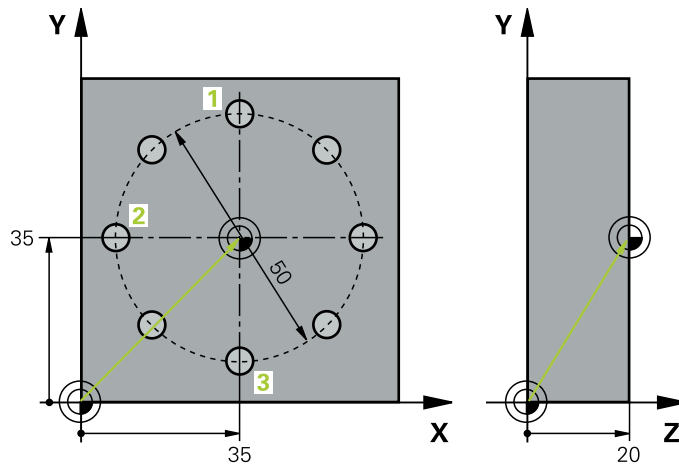


- **Q325** = Polar coordinate angle for touch point 1
- **Q247** = Stepping angle for calculating the touch points 2 to 4
- **Q305** = Write to row number 5 of the preset table
- **Q303** = Write the calculated preset to the preset table
- **Q381** = Also set the preset in the touch probe axis
- **Q365** = Move on circular path between measuring points

0 BEGIN PGM 413 MM	
1 TOOL CALL "TOUCH_PROBE" Z	
2 TCH PROBE 413 PRESET OUTS. CIRCLE ~	
Q321=+25	;CENTER IN 1ST AXIS ~
Q322=+25	;CENTER IN 2ND AXIS ~
Q262=+30	;NOMINAL DIAMETER ~
Q325=+90	;STARTING ANGLE ~
Q247=+45	;STEPPING ANGLE ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+2	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q305=+5	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+10	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+25	;1ST CO. FOR TS AXIS ~
Q383=+25	;2ND CO. FOR TS AXIS ~
Q384=+0	;3RD CO. FOR TS AXIS ~
Q333=+0	;PRESET ~
Q423=+4	;NO. OF PROBE POINTS ~
Q365=+0	;TYPE OF TRAVERSE
3 END PGM 413 MM	

5.20 Example: Presetting on top surface of workpiece and at center of a bolt hole circle

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



- **Q291** = Polar coordinate angle for first hole center **1**
- **Q292** = Polar coordinate angle for second hole center **2**
- **Q293** = Polar coordinate angle for third hole center **3**
- **Q305** = Write center of bolt hole circle (X and Y) to row 1
- **Q303** = In the preset table **PRESET.PR**, save the calculated preset referenced to the machine-based coordinate system (REF system)

0 BEGIN PGM 416 MM	
1 TOOL CALL "TOUCH_PROBE" Z	
2 TCH PROBE 416 PRESET CIRCLE CENTER ~	
Q273=+35	;CENTER IN 1ST AXIS ~
Q274=+35	;CENTER IN 2ND AXIS ~
Q262=+50	;NOMINAL DIAMETER ~
Q291=+90	;ANGLE OF 1ST HOLE ~
Q292=+180	;ANGLE OF 2ND HOLE ~
Q293=+270	;ANGLE OF 3RD HOLE ~
Q261=+15	;MEASURING HEIGHT ~
Q260=+10	;CLEARANCE HEIGHT ~
Q305=+1	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE IN TS AXIS ~
Q382=+7.5	;1ST CO. FOR TS AXIS ~
Q383=+7.5	;2ND CO. FOR TS AXIS ~
Q384=+20	;3RD CO. FOR TS AXIS ~
Q333=+0	;PRESET ~
Q320=+0	;SET-UP CLEARANCE.
3 CYCL DEF 247 PRESETTING ~	
Q339=+1	;PRESET NUMBER
4 END PGM 416 MM	

6

**Touch Probe
Cycles: Automatic
Workpiece
Inspection**

6.1 Fundamentals

6.1.1 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.

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 cycles for measuring workpieces automatically:

Cycle		Call	Further information
0	REF. PLANE <ul style="list-style-type: none"> ■ Measuring a coordinate in a selectable axis 	DEF- active	Page 209
1	POLAR PRESET <ul style="list-style-type: none"> ■ Measuring a point ■ Probing direction via angle 	DEF- active	Page 210
420	MEASURE ANGLE <ul style="list-style-type: none"> ■ Measuring an angle in the working plane 	DEF- active	Page 212
421	MEASURE HOLE <ul style="list-style-type: none"> ■ Measuring the position of a hole ■ Measuring the diameter of a hole ■ Nominal-to-actual value comparison, if applicable 	DEF- active	Page 215
422	MEAS. CIRCLE OUTSIDE <ul style="list-style-type: none"> ■ Measuring the position of a circular stud ■ Measuring the diameter of a circular stud ■ Nominal-to-actual value comparison, if applicable 	DEF- active	Page 221
423	MEAS. RECTAN. INSIDE <ul style="list-style-type: none"> ■ Measuring the position of a rectangular pocket ■ Measuring the length and width of a rectangular pocket ■ Nominal-to-actual value comparison, if applicable 	DEF- active	Page 227
424	MEAS. RECTAN. OUTS. <ul style="list-style-type: none"> ■ Measuring the position of a rectangular stud ■ Measuring the length and width of a rectangular stud ■ Nominal-to-actual value comparison, if applicable 	DEF- active	Page 231

Cycle	Call	Further information
425 MEASURE INSIDE WIDTH <ul style="list-style-type: none"> ■ Measuring the position of a slot ■ Measuring the width of a slot ■ Nominal-to-actual value comparison, if applicable 	DEF- active	Page 236
426 MEASURE RIDGE WIDTH <ul style="list-style-type: none"> ■ Measuring the position of a ridge ■ Measuring the width of a ridge ■ Nominal-to-actual value comparison, if applicable 	DEF- active	Page 240
427 MEASURE COORDINATE <ul style="list-style-type: none"> ■ Measuring any coordinate in a selectable axis ■ Nominal-to-actual value comparison, if applicable 	DEF- active	Page 243
430 MEAS. BOLT HOLE CIRC <ul style="list-style-type: none"> ■ Measuring the center point of a bolt hole circle ■ Measuring the diameter of a bolt hole circle ■ Nominal-to-actual value comparison, if applicable 	DEF- active	Page 248
431 MEASURE PLANE <ul style="list-style-type: none"> ■ Finding the angle of a plane by measuring three points 	DEF- active	Page 253

6.1.2 Recording the results of measurement

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

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

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

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



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

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

Measuring log for Probing Cycle 421 Hole Measuring

Date: 30-06-2005

Time: 6:55:04

Measuring program: TNC:\GEH35712\CHECK1.H

Type of dimension (0 = MM / 1 = INCH): 0

Nominal values:

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

Given limit values:

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

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

Actual values:

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

Deviations:

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

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

End of measuring log

6.1.3 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. The highlighted result parameter belongs to that input parameter.

6.1.4 Classification of results

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

Parameter value	Measuring status
Q180 = 1	Measurement results are within tolerance
Q181 = 1	Rework is required
Q182 = 1	Scrap

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

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



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

6.1.5 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.

6.1.6 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. Select the tool name input via **Name** in the action bar.



- 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 cutter

If you reference a milling cutter in parameter **Q330**, the appropriate values will be compensated as follows:

The control always compensates the tool radius in the **DR** column of the tool table, even if the measured deviation lies within the given tolerance.

You can inquire whether re-working is necessary via parameter **Q181** in the NC program (**Q181=1**: rework required).

Turning tool

Only applies to Cycles **421**, **422**, **427**.

If you 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).

Compensating an indexed tool

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

- **Q50** = "TOOL NAME"
- **FN18: SYSREAD Q0 = ID990 NR10 IDX0**; specify the number of the **QS** parameter in **IDX**
- **Q0** = **Q0** + 0.2; add the index of the basic tool number
- In the cycle: **Q330** = **Q0**; use the indexed tool number

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 and Program Run

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).

6.1.7 Reference system for measurement results

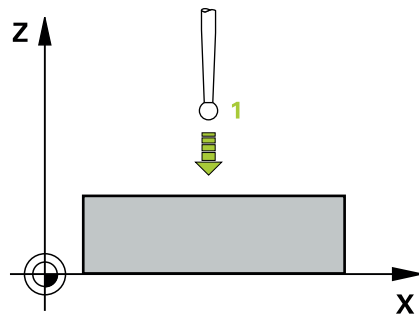
The control transfers all measurement results, which reference the active coordinate system, or as the case may be, the shifted or/and rotated/tilted coordinate system, to the result parameters and the log file.

6.2 Cycle 0 REF. PLANE

Application

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

Cycle sequence



- 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.

Notes

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 to a position where there is no danger of collision when the programmed pre-positioning point is approached

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

6.2.1 Cycle parameters

Help graphic	Parameter
	Parameter number for result? Enter the number of the Q parameter to which you want to assign the coordinate.. Input: 0...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. Input: -, +
	Position value? Use the axis keys or the alphabetic keyboard to enter all coordinates for pre-positioning of the touch probe. Input: -999999999...+999999999

Example

```
11 TCH PROBE 0.0 REF. PLANE Q9 Z+
```

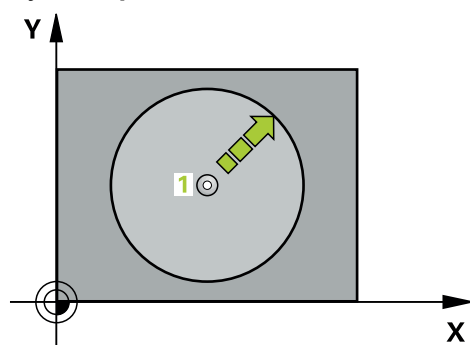
```
12 TCH PROBE 0.1 X+99 Y+22 Z+2
```

6.3 Cycle 1 POLAR PRESET

Application

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

Cycle sequence



- 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**

Notes

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 to a position where there is no danger of collision when the programmed pre-positioning point is approached

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The probing axis defined in the cycle specifies the probing plane:
 Probing axis X: X/Y plane
 Probing axis Y: Y/Z plane
 Probing axis Z: Z/X plane

6.3.1 Cycle parameters

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

Example

```

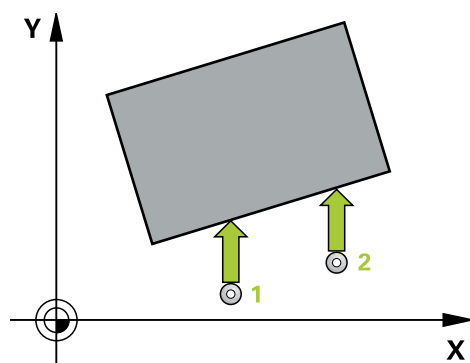
11 TCH PROBE 1.0 POLAR PRESET
12 TCH PROBE 1.1 X WINKEL:+30
13 TCH PROBE 1.2 X+0 Y+10 Z+3
  
```

6.4 Cycle 420 MEASURE ANGLE

Application

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

Cycle sequence



- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the programmed touch point **1**. The sum of **Q320**, **SET_UP** and the ball-tip radius is taken into account for probe movements in any probing direction. When the probe movement starts, the center of the ball tip will be offset by this sum in the direction opposite the probing direction.

Further information: "Positioning logic", Page 50

- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 The touch probe then moves to the next touch point **2** and probes again.
- 4 The control returns the touch probe to the clearance height and saves the measured angle in the following Q parameter:

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

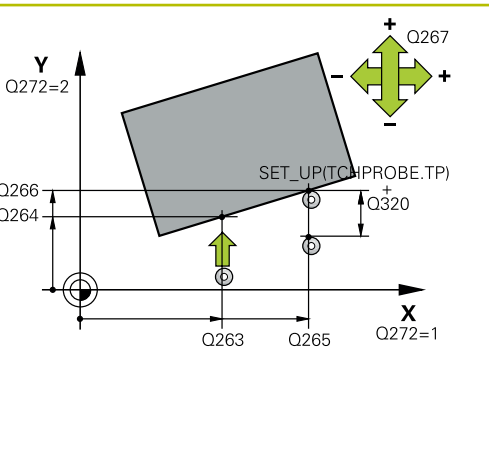
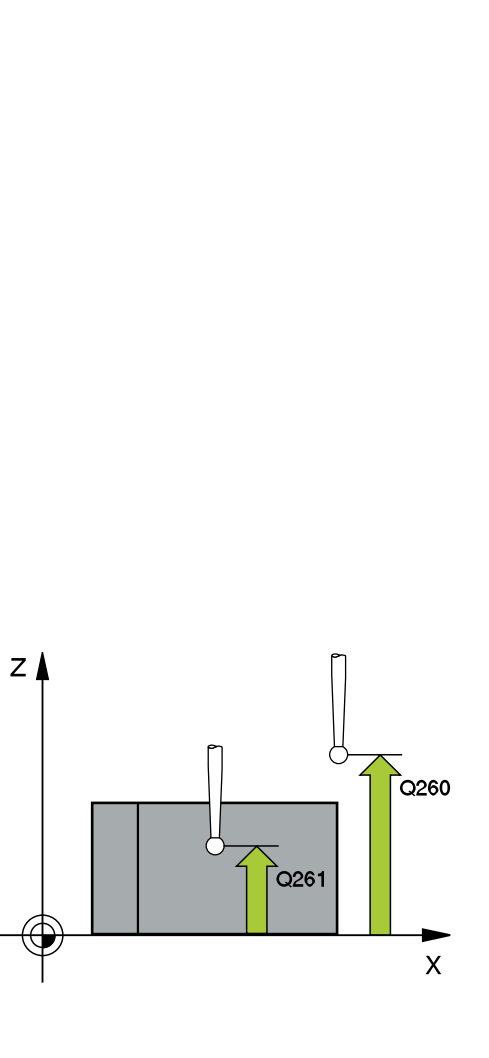
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If touch probe axis = measuring axis, you can measure the angle in the direction of the A axis or B axis:
 - If you want to measure the angle in the direction of the A axis, set **Q263** equal to **Q265** and **Q264** unequal to **Q266**.
 - If you want to measure the angle in the direction of the B axis, set **Q263** not equal to **Q265** and **Q264** equal to **Q266**.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

6.4.1 Cycle parameters

Help graphic	Parameter
	<p>Q263 1st measuring point in 1st axis? Coordinate of the first touch point in the main axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999</p> <hr/> <p>Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999</p> <hr/> <p>Q265 2nd measuring point in 1st axis? Coordinate of the second touch point in the main axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999</p> <hr/> <p>Q266 2nd measuring point in 2nd axis? Coordinate of the second touch point in the secondary axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999</p> <hr/> <p>Q272 Meas. axis (1/2/3, 1=ref. axis)? Axis in which the measurement will be made: 1: Main axis = measuring axis 2: Secondary axis = measuring axis 3: Touch probe axis = measuring axis Input: 1, 2, 3</p> <hr/> <p>Q267 Trav. direction (+1=+ / -1=-)? Direction in which the touch probe will approach the workpiece: -1: Negative traverse direction +1: Positive traverse direction Input: -1, +1</p> <hr/>
	<p>Q261 Measuring height in probe axis? Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect. Input: -99999.9999...+99999.9999</p> <hr/> <p>Q320 Set-up clearance? Additional distance between measuring point and ball tip. The touch probe movement will start with an offset of the sum of Q320, SET_UP, and the ball-tip radius, even when probing in the tool axis direction. This value has an incremental effect. Input: 0...99999.9999 or PREDEF</p>

Help graphic	Parameter
	<p>Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: -99999.9999...+99999.9999 or PREDEF</p>
	<p>Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: 0, 1</p>
	<p>Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: Define whether the control will create a measuring log: 1: Create a measuring log: The control will save the log file named TCHPR420.TXT in the folder that also contains the associated NC program. 2: Interrupt program run and display the measuring log on the control screen (you can later resume the NC program run with NC Start) Input: 0, 1, 2</p>

Example

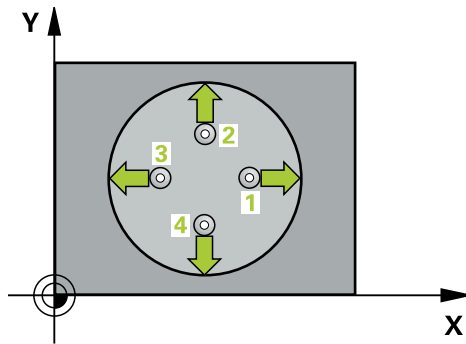
11 TCH PROBE 420 MEASURE ANGLE ~	
Q263=+10	;1ST POINT 1ST AXIS ~
Q264=+10	;1ST POINT 2ND AXIS ~
Q265=+15	;2ND PNT IN 1ST AXIS ~
Q266=+95	;2ND POINT 2ND AXIS ~
Q272=+1	;MEASURING AXIS ~
Q267=-1	;TRAVERSE DIRECTION ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+10	;CLEARANCE HEIGHT ~
Q301=+1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

6.5 Cycle 421 MEASURE HOLE

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 sequence



- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in 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:

Further information: "Positioning logic", Page 50

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

Notes

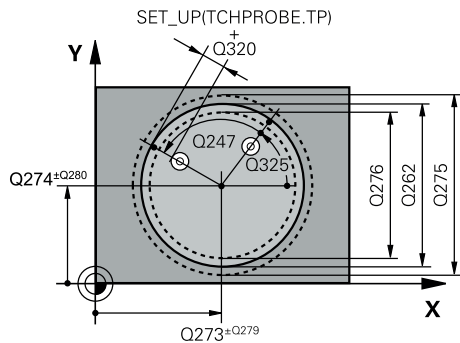
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- If parameter **Q330** references a milling tool, the information in parameters **Q498** and **Q531** has no effect
- If parameter **Q330** references a turning tool, the following applies:
 - Parameters **Q498** and **Q531** have to be defined
 - The information in parameters **Q498**, **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**.

6.5.1 Cycle parameters

Help graphic



Parameter

Q273 Center in 1st axis (nom. value)?

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

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

Q274 Center in 2nd axis (nom. value)?

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

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

Q262 Nominal diameter?

Enter the diameter of the hole.

Input: **0...99999.9999**

Q325 Starting angle?

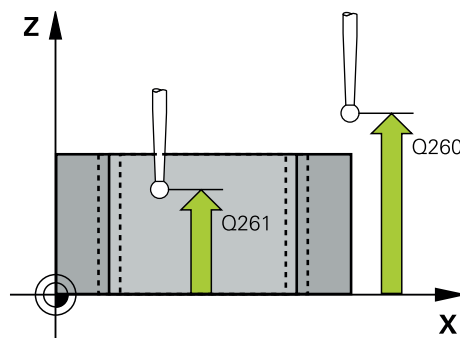
Angle between the main axis of the working plane and the first touch point. The value has an absolute effect.

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

Q247 Intermediate stepping angle?

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

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



Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q301 Move to clearance height (0/1)?

Specify how the touch probe moves between measuring points:

0: Move at measuring height between measuring points

1: Move at clearance height between measuring points

Input: **0, 1**

Help graphic	Parameter
	Q275 Maximum limit of size for hole? Maximum permissible diameter for the hole (circular pocket) Input: 0...99999.9999
	Q276 Minimum limit of size? Minimum permissible diameter for the hole (circular pocket) Input: 0...99999.9999
	Q279 Tolerance for center 1st axis? Permissible position deviation in the main axis of the working plane. Input: 0...99999.9999
	Q280 Tolerance for center 2nd axis? Permissible position deviation in the secondary axis of the working plane. Input: 0...99999.9999
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: 0: Do not create a measuring log 1: Create a measuring log: The control will save the log file named TCHPR421.TXT by default in the directory that also contains the associated NC program. 2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with NC Start . Input: 0, 1, 2
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: 0: Do not interrupt program run; no error message 1: Interrupt program run and output an error message Input: 0, 1
	Q330 Tool for monitoring? Define whether the control will monitor the tool (see "Tool monitoring", Page 207): 0: Monitoring not active > 0: Number or name of the tool used for machining. Via a selection in the action bar, you have the option of applying a tool directly from the tool table. Input: 0...99999.9 or max. 255 characters

Help graphic	Parameter
	<p>Q423 No. probe points in plane (4/3)? Define whether the control will use three or four touch points to measure the circle: 3: Use three measuring points 4: Use four measuring points (default setting) Input: 3, 4</p>
	<p>Q365 Type of traverse? Line=0/arc=1 Specify the path function to be used by the tool for moving between the measuring points if "traverse to clearance height" (Q301 = 1) is active. 0: Move in a straight line between machining operations 1: Move along a circular arc on the pitch circle diameter between machining operations Input: 0, 1</p>
	<p>Q498 Reverse tool (0=no/1=yes)? Only relevant if you have entered a turning tool in parameter Q330 before. For proper monitoring of the turning tool, the control requires the exact machining situation. Therefore, enter the following: 1: Turning tool is mirrored (rotated by 180°) by, for example, Cycle 800 and parameter Reverse the tool Q498 = 1 0: Turning tool corresponds to the description in the turning tool table (toolturn.trn); no modification by, for example, Cycle 800 and parameter Reverse the tool Q498 = 0 Input: 0, 1</p>
	<p>Q531 Angle of incidence? Only relevant if you have entered a turning tool in parameter Q330 before. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining (e.g., from Cycle 800, Angle of incidence? Q531). Input: -180...+180</p>

Example

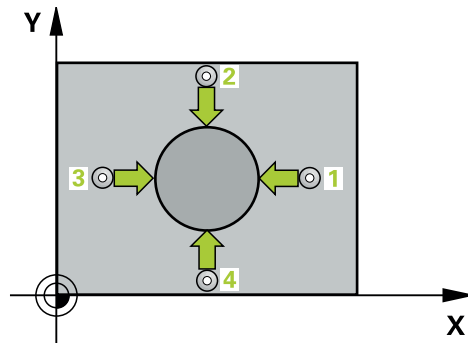
11 TCH PROBE 421 MEASURE HOLE ~	
Q273=+50	;CENTER IN 1ST AXIS ~
Q274=+50	;CENTER IN 2ND AXIS ~
Q262=+75	;NOMINAL DIAMETER ~
Q325=+0	;STARTING ANGLE ~
Q247=+60	;STEPPING ANGLE ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+1	;MOVE TO CLEARANCE ~
Q275=+75.12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

6.6 Cycle 422 MEAS. CIRCLE OUTSIDE

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 sequence



- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
Further information: "Positioning logic", Page 50
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
- 3 Then, the touch probe moves in a circular arc either at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

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

Notes

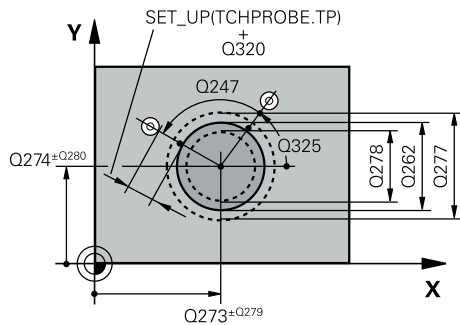
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- If parameter **Q330** references a milling tool, the information in parameters **Q498** and **Q531** has no effect
- If parameter **Q330** references a turning tool, the following applies:
 - Parameters **Q498** and **Q531** have to be defined
 - The information in parameters **Q498**, **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**.

6.6.1 Cycle parameters

Help graphic



Parameter

Q273 Center in 1st axis (nom. value)?

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

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

Q274 Center in 2nd axis (nom. value)?

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

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

Q262 Nominal diameter?

Enter the diameter of the stud.

Input: **0...99999.9999**

Q325 Starting angle?

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

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

Q247 Intermediate stepping angle?

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

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

Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

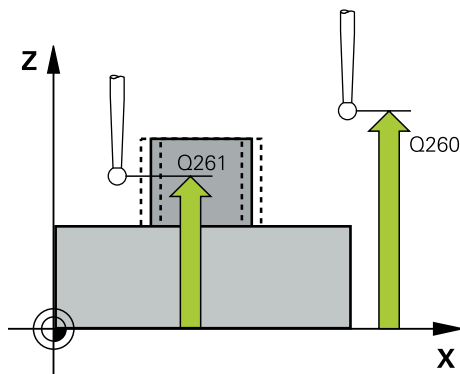
Input: **-99999.9999...+99999.9999** or **PREDEF**

Q301 Move to clearance height (0/1)?

Specify how the touch probe moves between measuring points:

- 0: Move at measuring height between measuring points
- 1: Move at clearance height between measuring points

Input: **0, 1**



Help graphic	Parameter
	Q277 Maximum limit of size for stud? Maximum permissible diameter for the stud. Input: 0...99999.9999
	Q278 Minimum limit of size for stud? Minimum permissible diameter for the stud. Input: 0...99999.9999
	Q279 Tolerance for center 1st axis? Permissible position deviation in the main axis of the working plane. Input: 0...99999.9999
	Q280 Tolerance for center 2nd axis? Permissible position deviation in the secondary axis of the working plane. Input: 0...99999.9999
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: 0: Do not create a measuring log 1: Create a measuring log: The control will save the log file named TCHPR422.TXT in the folder that also contains the associated NC program. 2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with NC Start . Input: 0, 1, 2
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: 0: Do not interrupt program run; no error message 1: Interrupt program run and output an error message Input: 0, 1
	Q330 Tool for monitoring? Define whether the control will monitor the tool Page 207. 0: Monitoring not active > 0: Tool number in tool table TOOL.T Input: 0...99999.9 or max. 255 characters
	Q423 No. probe points in plane (4/3)? Define whether the control will use three or four touch points to measure the circle: 3: Use three measuring points 4: Use four measuring points (default setting) Input: 3, 4

Help graphic	Parameter
	<p>Q365 Type of traverse? Line=0/arc=1</p> <p>Specify the path function to be used by the tool for moving between the measuring points if "traverse to clearance height" (Q301 = 1) is active.</p> <p>0: Move in a straight line between machining operations</p> <p>1: Move along a circular arc on the pitch circle diameter between machining operations</p> <p>Input: 0, 1</p>
	<p>Q498 Reverse tool (0=no/1=yes)?</p> <p>Only relevant if you have entered a turning tool in parameter Q330 before. For proper monitoring of the turning tool, the control requires the exact machining situation. Therefore, enter the following:</p> <p>1: Turning tool is mirrored (rotated by 180°) by, for example, Cycle 800 and parameter Reverse the tool Q498 = 1</p> <p>0: Turning tool corresponds to the description in the turning tool table (toolturn.trn); no modification by, for example, Cycle 800 and parameter Reverse the tool Q498 = 0</p> <p>Input: 0, 1</p>
	<p>Q531 Angle of incidence?</p> <p>Only relevant if you have entered a turning tool in parameter Q330 before. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining (e.g., from Cycle 800, Angle of incidence? Q531).</p> <p>Input: -180...+180</p>

Example

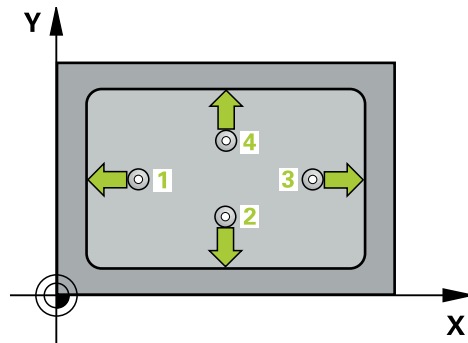
11 TCH PROBE 422 MEAS. CIRCLE OUTSIDE ~	
Q273=+50	;CENTER IN 1ST AXIS ~
Q274=+50	;CENTER IN 2ND AXIS ~
Q262=+75	;NOMINAL DIAMETER ~
Q325=+90	;STARTING ANGLE ~
Q247=+30	;STEPPING ANGLE ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+10	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q277=+35.15	;MAXIMUM LIMIT ~
Q278=+34.9	;MINIMUM LIMIT ~
Q279=+0.05	;TOLERANCE 1ST CENTER ~
Q280=+0.05	;TOLERANCE 2ND CENTER ~
Q281=+1	;MEASURING LOG ~
Q309=+0	;PGM STOP TOLERANCE ~
Q330=+0	;TOOL ~
Q423=+4	;NO. OF PROBE POINTS ~
Q365=+1	;TYPE OF TRAVERSE ~
Q498=+0	;REVERSE TOOL ~
Q531=+0	;ANGLE OF INCIDENCE

6.7 Cycle 423 MEAS. RECTAN. INSIDE

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 sequence



- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
- Further information:** "Positioning logic", Page 50
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

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

Notes

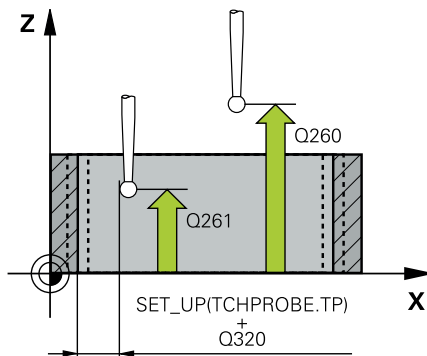
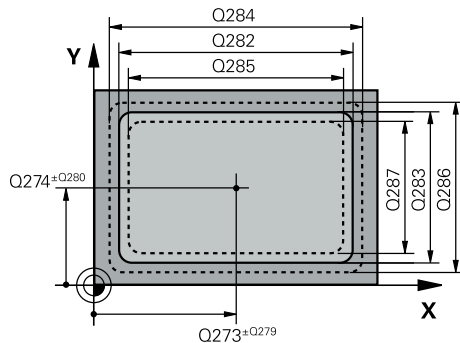
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points.
- Tool monitoring is dependent on the deviation of the first side length.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

6.7.1 Cycle parameters

Help graphic



Parameter

Q273 Center in 1st axis (nom. value)?

Center of the pocket in the main axis of the working plane. The value has an absolute effect.

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

Q274 Center in 2nd axis (nom. value)?

Center of the pocket in the secondary axis of the working plane. The value has an absolute effect.

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

Q282 1st side length (nominal value)?

Pocket length, parallel to the main axis of the working plane

Input: **0...99999.9999**

Q283 2nd side length (nominal value)?

Pocket length, parallel to the secondary axis of the working plane

Input: **0...99999.9999**

Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q301 Move to clearance height (0/1)?

Specify how the touch probe moves between measuring points:

0: Move at measuring height between measuring points

1: Move at clearance height between measuring points

Input: **0, 1**

Q284 Max. size limit 1st side length?

Maximum permissible length for the pocket

Input: **0...99999.9999**

Q285 Min. size limit 1st side length?

Minimum permissible length for the pocket

Input: **0...99999.9999**

Help graphic	Parameter
	Q286 Max. size limit 2nd side length? Maximum permissible width for the pocket Input: 0...99999.9999
	Q287 Min. size limit 2nd side length? Minimum permissible width for the pocket Input: 0...99999.9999
	Q279 Tolerance for center 1st axis? Permissible position deviation in the main axis of the working plane. Input: 0...99999.9999
	Q280 Tolerance for center 2nd axis? Permissible position deviation in the secondary axis of the working plane. Input: 0...99999.9999
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: 0: Do not create a measuring log. 1: Create a measuring log: The control will save the log file named TCHPR423.TXT in the folder that also contains the associated NC program. 2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with NC Start . Input: 0, 1, 2
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: 0: Do not interrupt program run; no error message 1: Interrupt program run and output an error message Input: 0, 1
	Q330 Tool for monitoring? Define whether the control will monitor the tool Page 207. 0: Monitoring not active > 0: Tool number in tool table TOOL.T Input: 0...99999.9 or max. 255 characters

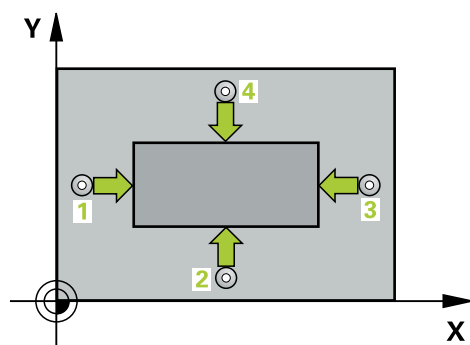
Example

11 TCH PROBE 423 MEAS. RECTAN. INSIDE ~	
Q273=+50	;CENTER IN 1ST AXIS ~
Q274=+50	;CENTER IN 2ND AXIS ~
Q282=+80	;FIRST SIDE LENGTH ~
Q283=+60	;2ND SIDE LENGTH ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+10	;CLEARANCE HEIGHT ~
Q301=+1	;MOVE TO CLEARANCE ~
Q284=+0	;MAX. LIMIT 1ST SIDE ~
Q285=+0	;MIN. LIMIT 1ST SIDE ~
Q286=+0	;MAX. LIMIT 2ND SIDE ~
Q287=+0	;MIN. LIMIT 2ND SIDE ~
Q279=+0	;TOLERANCE 1ST CENTER ~
Q280=+0	;TOLERANCE 2ND CENTER ~
Q281=+1	;MEASURING LOG ~
Q309=+0	;PGM STOP TOLERANCE ~
Q330=+0	;TOOL

6.8 Cycle 424 MEAS. RECTAN. OUTS.

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 sequence

- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
Further information: "Positioning logic", Page 50
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

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

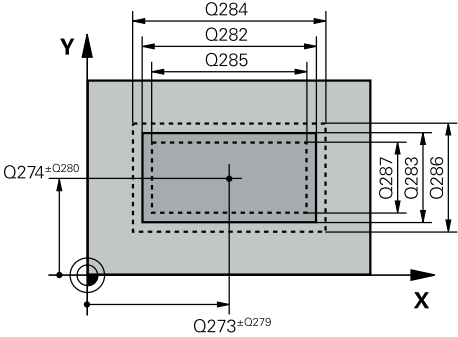
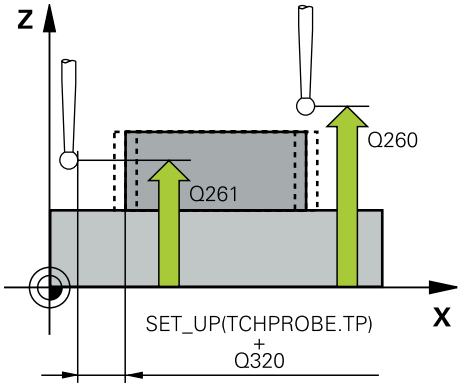
Notes

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

Note on programming

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

6.8.1 Cycle parameters

Help graphic	Parameter
	<p>Q273 Center in 1st axis (nom. value)? Center of the stud in the main axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q274 Center in 2nd axis (nom. value)? Center of the stud in the secondary axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q282 1st side length (nominal value)? Length of stud parallel to the main axis of the working plane Input: 0...99999.9999</p>
	<p>Q283 2nd side length (nominal value)? Length of stud parallel to the secondary axis of the working plane Input: 0...99999.9999</p>
	<p>Q261 Measuring height in probe axis? Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q320 Set-up clearance? Additional distance between touch point and ball tip. Q320 is in addition to the SET_UP column in the touch probe table. This value has an incremental effect. Input: 0...99999.9999 or PREDEF</p>
	<p>Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: -99999.9999...+99999.9999 or PREDEF</p>
	<p>Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: 0, 1</p>
	<p>Q284 Max. size limit 1st side length? Maximum permissible length for the stud Input: 0...99999.9999</p>
	<p>Q285 Min. size limit 1st side length? Minimum permissible length for the stud Input: 0...99999.9999</p>

Help graphic	Parameter
	Q286 Max. size limit 2nd side length? Maximum permissible width for the stud Input: 0...99999.9999
	Q287 Min. size limit 2nd side length? Minimum permissible width for the stud Input: 0...99999.9999
	Q279 Tolerance for center 1st axis? Permissible position deviation in the main axis of the working plane. Input: 0...99999.9999
	Q280 Tolerance for center 2nd axis? Permissible position deviation in the secondary axis of the working plane. Input: 0...99999.9999
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: 0: Do not create a measuring log 1: Create a measuring log: The control will save the log file named TCHPR424.TXT in the folder that also contains the .h file 2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with NC Start . Input: 0, 1, 2
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: 0: Do not interrupt program run; no error message 1: Interrupt program run and output an error message Input: 0, 1
	Q330 Tool for monitoring? Define whether the control will monitor the tool (see "Tool monitoring", Page 207): 0: Monitoring not active > 0: Number or name of the tool used for machining. Via a selection in the action bar, you have the option of applying a tool directly from the tool table. Input: 0...99999.9 or max. 255 characters

Example

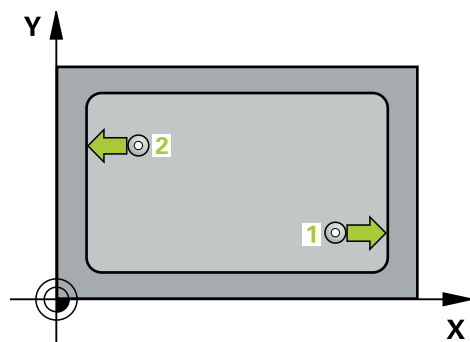
11 TCH PROBE 424 MEAS. RECTAN. OUTS. ~	
Q273=+50	;CENTER IN 1ST AXIS ~
Q274=+50	;2ND CENTER 2ND AXIS ~
Q282=+75	;FIRST SIDE LENGTH ~
Q283=+35	;2ND SIDE LENGTH ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;MOVE TO CLEARANCE ~
Q284=+75.1	;MAX. LIMIT 1ST SIDE ~
Q285=+74.9	;MIN. LIMIT 1ST SIDE ~
Q286=+35	;MAX. LIMIT 2ND SIDE ~
Q287=+34.95	;MIN. LIMIT 2ND SIDE ~
Q279=+0.1	;TOLERANCE 1ST CENTER ~
Q280=+0.1	;TOLERANCE 2ND CENTER ~
Q281=+1	;MEASURING LOG ~
Q309=+0	;PGM STOP TOLERANCE ~
Q330=+0	;TOOL

6.9 Cycle 425 MEASURE INSIDE WIDTH

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 sequence



- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control calculates the touch points from the data in the cycle and from the set-up clearance in the **SET_UP** column of the touch probe table.
Further information: "Positioning logic", Page 50
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The first probing is always in the positive direction of the programmed axis.
- 3 If you enter an offset for the second measurement, the control then moves the touch probe (if required, at clearance height) to the next touch point **2** and probes that point. If the nominal length is large, the control moves the touch probe to the second touch point at rapid traverse. If you do not enter an offset, the control measures the width in the exact opposite direction.
- 4 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

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

Notes

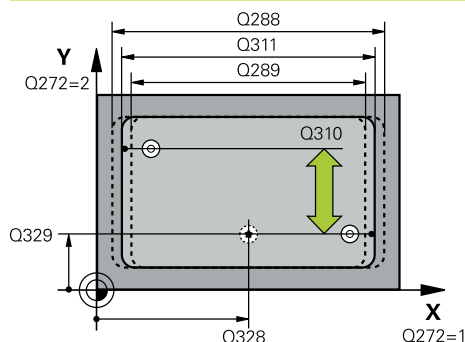
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

6.9.1 Cycle parameters

Help graphic



Parameter

Q328 Starting point in 1st axis?

Starting point for probing in the main axis of the working plane. The value has an absolute effect.

Input: -99999.9999...+99999.9999

Q329 Starting point in 2nd axis?

Starting point for probing in the secondary axis of the working plane. The value has an absolute effect.

Input: -99999.9999...+99999.9999

Q310 Offset for 2nd measurement (+/-)?

Distance by which the touch probe is offset before the second measurement. If you enter 0, the control does not offset the touch probe. This value has an incremental effect.

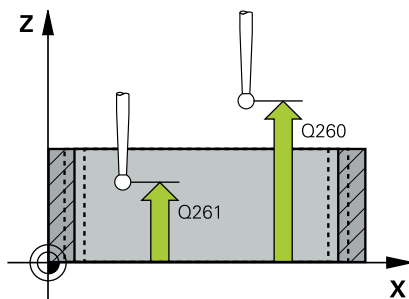
Input: -99999.9999...+99999.9999

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

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

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

Input: 1, 2



Q261 Measuring height in probe axis?

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

Input: -99999.9999...+99999.9999

Q260 Clearance height?

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

Input: -99999.9999...+99999.9999 or PREDEF

Q311 Nominal length?

Nominal value of the length to be measured

Input: 0...99999.9999

Q288 Maximum limit of size?

Maximum permissible length

Input: 0...99999.9999

Q289 Minimum limit of size?

Minimum permissible length

Input: 0...99999.9999

Help graphic	Parameter
	<p>Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: 0: Do not create a measuring log 1: Create a measuring log: The control will save the log file named TCHPR425.TXT in the folder that also contains the .h file 2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with NC Start. Input: 0, 1, 2</p>
	<p>Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: 0: Do not interrupt program run; no error message 1: Interrupt program run and output an error message Input: 0, 1</p>
	<p>Q330 Tool for monitoring? Define whether the control will monitor the tool (see "Tool monitoring", Page 207): 0: Monitoring not active > 0: Number or name of the tool used for machining. Via a selection in the action bar, you have the option of applying a tool directly from the tool table. Input: 0...99999.9 or max. 255 characters</p>
	<p>Q320 Set-up clearance? Additional distance between 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. This value has an incremental effect. Input: 0...99999.9999 or PREDEF</p>
	<p>Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: 0, 1</p>

Example

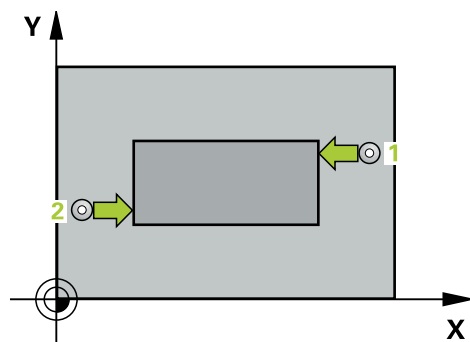
11 TCH PROBE 425 MEASURE INSIDE WIDTH ~	
Q328=+75	;STARTNG PNT 1ST AXIS ~
Q329=-12.5	;STARTNG PNT 2ND AXIS ~
Q310=+0	;OFFS. 2ND MEASUREMNT ~
Q272=+1	;MEASURING AXIS ~
Q261=-5	;MEASURING HEIGHT ~
Q260=+10	;CLEARANCE HEIGHT ~
Q311=+25	;NOMINAL LENGTH ~
Q288=+25.05	;MAXIMUM LIMIT ~
Q289=+25	;MINIMUM LIMIT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

6.10 Cycle 426 MEASURE RIDGE WIDTH

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 sequence



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

Further information: "Positioning logic", Page 50

- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The first probing is always in the negative direction of the programmed axis.
- 3 Then the touch probe moves at clearance height to the next touch point and probes it.
- 4 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

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

Notes

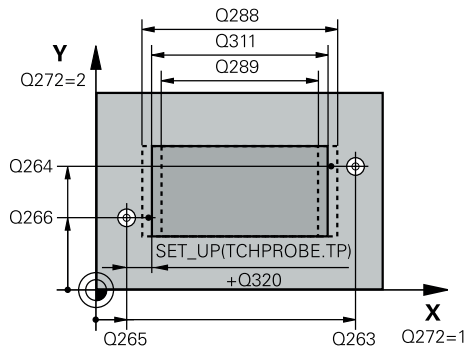
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

6.10.1 Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

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

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

Q264 1st measuring point in 2nd axis?

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

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

Q265 2nd measuring point in 1st axis?

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

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

Q266 2nd measuring point in 2nd axis?

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

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

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

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

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

Input: **1, 2**

Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q311 Nominal length?

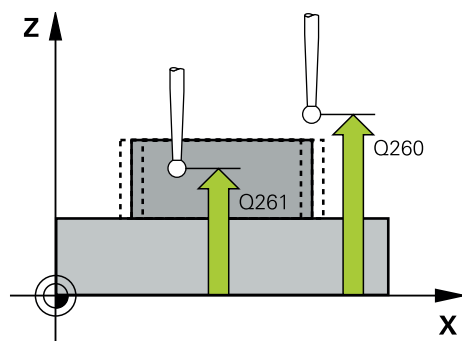
Nominal value of the length to be measured

Input: **0...99999.9999**

Q288 Maximum limit of size?

Maximum permissible length

Input: **0...99999.9999**



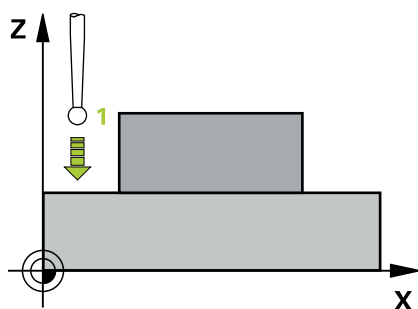
Help graphic	Parameter
	Q289 Minimum limit of size? Minimum permissible length Input: 0...99999.9999
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: 0: Do not create a measuring log 1: Create a measuring log: The control will save the log file named TCHPR426.TXT in the folder that also contains the associated NC program. 2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with NC Start . Input: 0, 1, 2
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: 0: Do not interrupt program run; no error message 1: Interrupt program run and output an error message Input: 0, 1
	Q330 Tool for monitoring? Define whether the control will monitor the tool (see "Tool monitoring", Page 207): 0: Monitoring not active > 0: Number or name of the tool used for machining. Via a selection in the action bar, you have the option of applying a tool directly from the tool table. Input: 0...99999.9 or max. 255 characters

Example

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

6.11 Cycle 427 MEASURE COORDINATE**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 sequence

- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) to the touch point **1**. The control offsets the touch probe by the set-up clearance in the direction opposite the defined traverse direction

Further information: "Positioning logic", Page 50

- 2 Then the control positions the touch probe to the specified touch point **1** in the working plane and measures the actual value in the selected axis.
- 3 Finally, the control returns the touch probe to the clearance height and saves the measured coordinate in the following Q parameter:

Q parameter number	Meaning
Q160	Measured coordinate

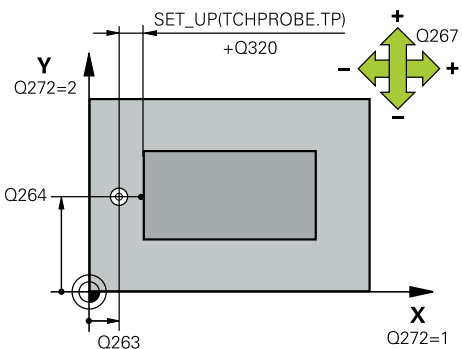
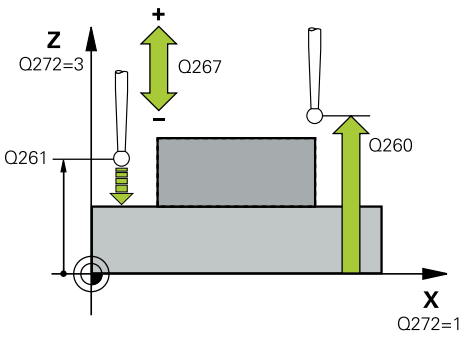
Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If an axis of the active working plane is defined as the measuring axis (**Q272** = 1 or 2), the control will perform a tool radius compensation. The control determines the direction of compensation from the defined traversing direction (**Q267**).
- If the touch probe axis is defined as the measuring axis (**Q272** = 3), the control will perform a tool length compensation.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

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

6.11.1 Cycle parameters

Help graphic	Parameter
	Q263 1st measuring point in 1st axis? Coordinate of the first touch point in the main axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q261 Measuring height in probe axis? Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q320 Set-up clearance? Additional distance between touch point and ball tip. Q320 is in addition to the SET_UP column in the touch probe table. This value has an incremental effect. Input: 0...99999.9999 or PREDEF
	Q272 Meas. axis (1/2/3, 1=ref. axis)? Axis in which the measurement will be made: 1: Main axis = measuring axis 2: Secondary axis = measuring axis 3: Touch probe axis = measuring axis Input: 1, 2, 3
	Q267 Trav. direction 1 (+1=+ / -1=-)? Direction in which the touch probe will approach the workpiece: -1: Negative traverse direction +1: Positive traverse direction Input: -1, +1
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: -99999.9999...+99999.9999 or PREDEF

Help graphic	Parameter
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: 0: Do not create a measuring log 1: Create a measuring log: The control will save the log file named TCHPR427.TXT in the folder that also contains the associated NC program. 2: Interrupt the program run and display the measuring log on the control screen. Resume the NC program run with NC Start . Input: 0, 1, 2
	Q288 Maximum limit of size? Maximum permissible value Input: -99999.9999...+99999.9999
	Q289 Minimum limit of size? Minimum permissible value Input: -99999.9999...+99999.9999
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: 0: Do not interrupt program run; no error message 1: Interrupt program run and output an error message Input: 0, 1
	Q330 Tool for monitoring? Define whether the control will monitor the tool (see "Tool monitoring", Page 207): 0: Monitoring not active > 0: Number or name of the tool used for machining. Via a selection in the action bar, you have the option of applying a tool directly from the tool table. Input: 0...99999.9 or max. 255 characters

Help graphic**Parameter****Q498 Reverse tool (0=no/1=yes)?**

Only relevant if you have entered a turning tool in parameter **Q330** before. For proper monitoring of the turning tool, the control requires the exact machining situation. Therefore, enter the following:

1: Turning tool is mirrored (rotated by 180°) by, for example, Cycle **800** and parameter **Reverse the tool Q498** = 1

0: Turning tool corresponds to the description in the turning tool table (toolturn.trn); no modification by, for example, Cycle **800** and parameter **Reverse the tool Q498** = 0

Input: **0, 1**

Q531 Angle of incidence?

Only relevant if you have entered a turning tool in parameter **Q330** before. Enter the angle of incidence (inclination angle) between turning tool and workpiece during machining (e.g., from Cycle **800**, **Angle of incidence? Q531**).

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

Example

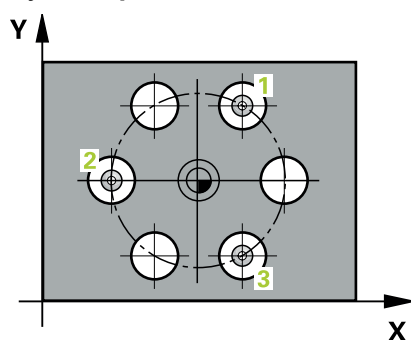
11 TCH PROBE 427 MEASURE COORDINATE ~	
Q263=+35	;1ST POINT 1ST AXIS ~
Q264=+45	;1ST POINT 2ND AXIS ~
Q261=+5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q272=+3	;MEASURING AXIS ~
Q267=-1	;TRAVERSE DIRECTION ~
Q260=+20	;CLEARANCE HEIGHT ~
Q281=+1	;MEASURING LOG ~
Q288=+5.1	;MAXIMUM LIMIT ~
Q289=+4.95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

6.12 Cycle 430 MEAS. BOLT HOLE CIRC

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 sequence



- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from **FMAX** column) to the programmed center point of the first hole **1**.

Further information: "Positioning logic", Page 50

- 2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
- 3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
- 4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
- 5 The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
- 6 The control moves the touch probe to the entered measuring height and probes four points to determine the third hole center point.
- 7 Finally, the control returns the touch probe to the clearance height and saves the actual values and deviations in the following Q parameters:

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

Notes

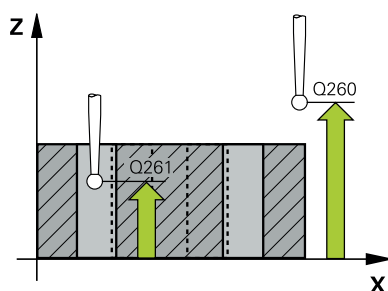
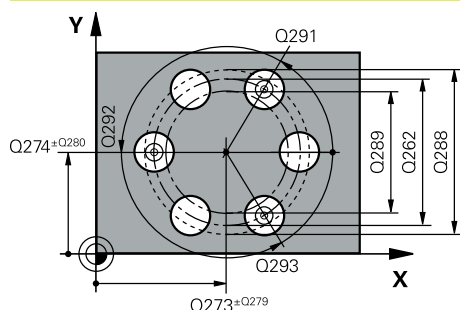
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Cycle **430** only monitors for tool breakage; there is no automatic tool compensation.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

6.12.1 Cycle parameters

Help graphic



Parameter

Q273 Center in 1st axis (nom. value)?

Bolt hole circle center (nominal value) in the main axis of the working plane. The value has an absolute effect.

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

Q274 Center in 2nd axis (nom. value)?

Bolt hole circle center (nominal value) in the secondary axis of the working plane. The value has an absolute effect.

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

Q262 Nominal diameter?

Enter the diameter of the hole.

Input: **0...99999.9999**

Q291 Polar coord. angle of 1st hole?

Polar coordinate angle of the first hole center in the working plane. The value has an absolute effect.

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

Q292 Polar coord. angle of 2nd hole?

Polar coordinate angle of the second hole center in the working plane. The value has an absolute effect.

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

Q293 Polar coord. angle of 3rd hole?

Polar coordinate angle of the third hole center in the working plane. The value has an absolute effect.

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

Q261 Measuring height in probe axis?

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

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

Q260 Clearance height?

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

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

Q288 Maximum limit of size?

Maximum permissible diameter of bolt hole circle

Input: **0...99999.9999**

Q289 Minimum limit of size?

Minimum permissible diameter of bolt hole circle

Input: **0...99999.9999**

Q279 Tolerance for center 1st axis?

Permissible position deviation in the main axis of the working plane.

Input: **0...99999.9999**

Help graphic	Parameter
	<p>Q280 Tolerance for center 2nd axis? Permissible position deviation in the secondary axis of the working plane. Input: 0...99999.9999</p>
	<p>Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: 0: Do not create a measuring log 1: Create a measuring log: The control will save the log file named TCHPR430.TXT in the folder that also contains the associated NC program 2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with NC Start. Input: 0, 1, 2</p>
	<p>Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: 0: Do not interrupt program run; no error message 1: Interrupt program run and output an error message Input: 0, 1</p>
	<p>Q330 Tool for monitoring? Define whether the control will monitor the tool (see "Tool monitoring", Page 207): 0: Monitoring not active > 0: Number or name of the tool used for machining. Via a selection in the action bar, you have the option of applying a tool directly from the tool table. Input: 0...99999.9 or max. 255 characters</p>

Example

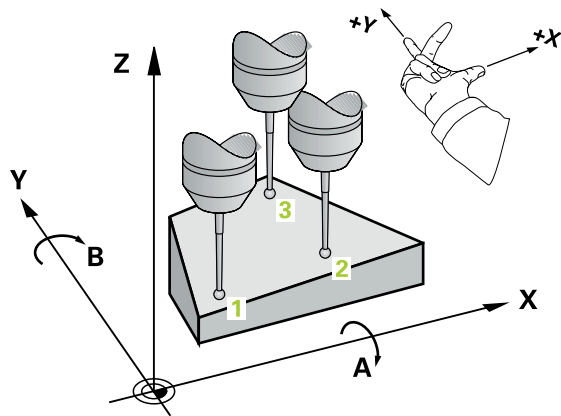
11 TCH PROBE 430 MEAS. BOLT HOLE CIRC ~	
Q273=+50	;CENTER IN 1ST AXIS ~
Q274=+50	;CENTER IN 2ND AXIS ~
Q262=+80	;NOMINAL DIAMETER ~
Q291=+0	;ANGLE OF 1ST HOLE ~
Q292=+90	;ANGLE OF 2ND HOLE ~
Q293=+180	;ANGLE OF 3RD HOLE ~
Q261=-5	;MEASURING HEIGHT ~
Q260=+10	;CLEARANCE HEIGHT ~
Q288=+80.1	;MAXIMUM LIMIT ~
Q289=+79.9	;MINIMUM LIMIT ~
Q279=+0.15	;TOLERANCE 1ST CENTER ~
Q280=+0.15	;TOLERANCE 2ND CENTER ~
Q281=+1	;MEASURING LOG ~
Q309=+0	;PGM STOP TOLERANCE ~
Q330=+0	;TOOL

6.13 Cycle 431 MEASURE PLANE

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 sequence



- 1 Following the positioning logic, the control positions the touch probe at rapid traverse (value from the **FMAX** column) at the programmed touch point **1** and measures the first point of the plane. The control offsets the touch probe by the set-up clearance in the direction opposite to the direction of probing.

Further information: "Positioning logic", Page 50

- 2 The touch probe returns to the clearance height and then moves in the working plane to touch point **2** and measures the actual value of the second touch point in the plane.
- 3 The touch probe returns to the clearance height and then moves in the working plane to touch point **3** and measures the actual value of the third touch point in the plane.
- 4 Finally the control returns the touch probe to the clearance height and saves the measured angle values in the following Q parameters:

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

Notes

NOTICE

Danger of collision!

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

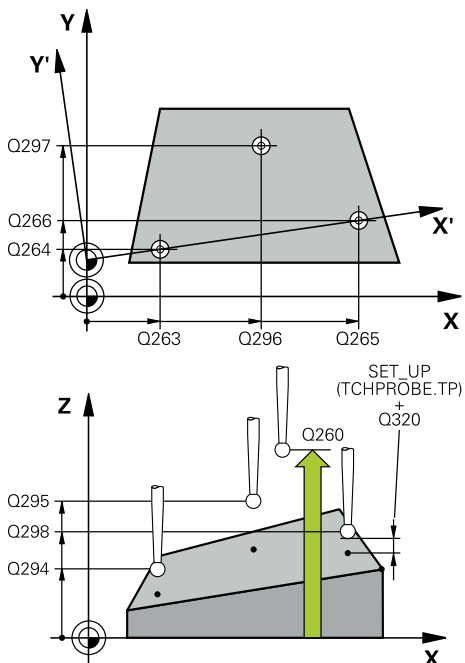
- Make sure to program **SYM (SEQ)** + or **SYM (SEQ)** -

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

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The spatial angles that are needed for the **Tilt working plane** function are saved in parameters **Q170** to **Q172**. With the first two measuring points, you also specify the direction of the main axis when tilting the working plane.
- The third measuring point determines the direction of the tool axis. Define the third measuring point in the direction of the positive Y axis to ensure that the position of the tool axis in a clockwise coordinate system is correct.

6.13.1 Cycle parameters

Help graphic	Parameter
	<p>Q263 1st measuring point in 1st axis? Coordinate of the first touch point in the main axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q294 1st measuring point in 3rd axis? Coordinate of the first touch point in the touch probe axis. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q265 2nd measuring point in 1st axis? Coordinate of the second touch point in the main axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q266 2nd measuring point in 2nd axis? Coordinate of the second touch point in the secondary axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q295 2nd measuring point in 3rd axis? Coordinate of the second touch point in the touch probe axis. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q296 3rd measuring point in 1st axis? Coordinate of the third touch point in the main axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q297 3rd measuring point in 2nd axis? Coordinate of the third touch point in the secondary axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q298 3rd measuring point in 3rd axis? Coordinate of the third touch point in the touch probe axis. The value has an absolute effect. Input: -99999.9999...+99999.9999</p>
	<p>Q320 Set-up clearance? Additional distance between touch point and ball tip. Q320 is in addition to the SET_UP column in the touch probe table. This value has an incremental effect. Input: 0...99999.9999 or PREDEF</p>

Help graphic

Parameter

Q260 Clearance height?

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

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

Q281 Measuring log (0/1/2)?

Define whether the control will create a measuring log:

0: Do not create a measuring log

1: Create a measuring log: The control will save the **log file named TCHPR431.TXT** in the folder that also contains the associated NC program

2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.

Input: **0, 1, 2**

Example

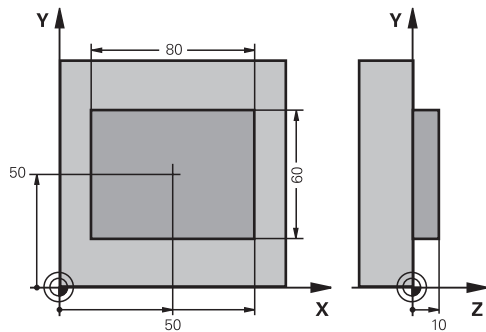
11 TCH PROBE 431 MEASURE PLANE ~	
Q263=+20	;1ST POINT 1ST AXIS ~
Q264=+20	;1ST POINT 2ND AXIS ~
Q294=-10	;1ST POINT 3RD AXIS ~
Q265=+50	;2ND PNT IN 1ST AXIS ~
Q266=+80	;2ND PNT IN 2ND AXIS ~
Q295=+0	;2ND PNT IN 3RD AXIS ~
Q296=+90	;3RD PNT IN 1ST AXIS ~
Q297=+35	;THIRD POINT 2ND AXIS ~
Q298=+12	;3RD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

6.14 Programming Examples

6.14.1 Example: Measuring and reworking a rectangular stud

Program sequence

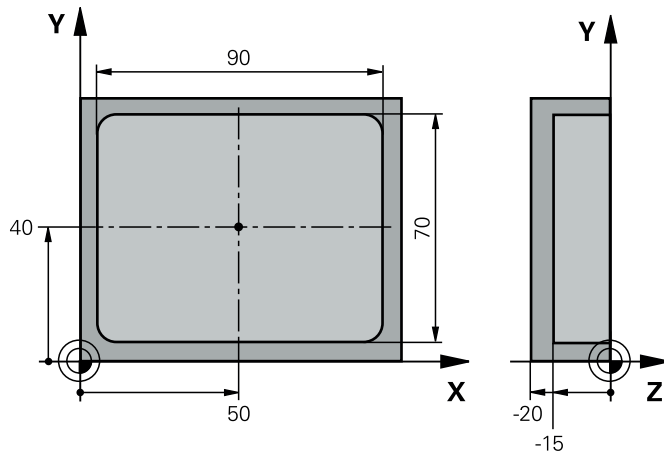
- Rough the rectangular stud with 0.5 mm finishing allowance
- Measure the rectangular stud
- Finish the rectangular stud, taking the measured values into account



0 BEGIN PGM TOUCHPROBE MM	
1 TOOL CALL 5 Z S6000	; Tool call: roughing
2 Q1 = 81	; Rectangle length in X (roughing dimension)
3 Q2 = 61	; Rectangle length in Y (roughing dimension)
4 L Z+100 R0 FMAX M3	; Retract the tool
5 CALL LBL 1	; Call the subprogram for machining
6 L Z+100 R0 FMAX	; Retract the tool
7 TOOL CALL 600 Z	; Call the touch probe
8 TCH PROBE 424 MEAS. RECTAN. OUTS. ~	
Q273=+50 ;CENTER IN 1ST AXIS ~	
Q274=+50 ;CENTER IN 2ND AXIS ~	
Q282=+80 ;FIRST SIDE LENGTH ~	
Q283=+60 ;2ND SIDE LENGTH ~	
Q261=-5 ;MEASURING HEIGHT ~	
Q320=+0 ;SET-UP CLEARANCE ~	
Q260=+30 ;CLEARANCE HEIGHT ~	
Q301=+0 ;MOVE TO CLEARANCE ~	
Q284=+0 ;MAX. LIMIT 1ST SIDE ~	
Q285=+0 ;MIN. LIMIT 1ST SIDE ~	
Q286=+0 ;MAX. LIMIT 2ND SIDE ~	
Q287=+0 ;MIN. LIMIT 2ND SIDE ~	
Q279=+0 ;TOLERANCE 1ST CENTER ~	
Q280=+0 ;TOLERANCE 2ND CENTER ~	
Q281=+0 ;MEASURING LOG ~	
Q309=+0 ;PGM STOP TOLERANCE ~	
Q330=+0 ;TOOL	

9 Q1 = Q1 - Q164	; Calculate the length in X based on the measured deviation
10 Q2 = Q2 - Q165	; Calculate the length in Y based on the measured deviation
11 L Z+100 R0 FMAX	; Retract the touch probe
12 TOOL CALL 25 Z S8000	; Tool call: finishing
13 L Z+100 R0 FMAX M3	; Retract the tool, end of program
14 CALL LBL 1	; Call the subprogram for machining
15 L Z+100 R0 FMAX	
16 M30	
17 LBL 1	; Subprogram with rectangular stud machining cycle
18 CYCL DEF 256 RECTANGULAR STUD ~	
Q218=+Q1 ;FIRST SIDE LENGTH ~	
Q424=+82 ;WORKPC. BLANK SIDE 1 ~	
Q219=+Q2 ;2ND SIDE LENGTH ~	
Q425=+62 ;WORKPC. BLANK SIDE 2 ~	
Q220=+0 ;RADIUS / CHAMFER ~	
Q368=+0.1 ;ALLOWANCE FOR SIDE ~	
Q224=+0 ;ANGLE OF ROTATION ~	
Q367=+0 ;STUD POSITION ~	
Q207=+500 ;FEED RATE MILLING ~	
Q351=+1 ;CLIMB OR UP-CUT ~	
Q201=-10 ;DEPTH ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q206=+3000 ;FEED RATE FOR PLNGNG ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+10 ;SURFACE COORDINATE ~	
Q204=+20 ;2ND SET-UP CLEARANCE ~	
Q370=+1 ;TOOL PATH OVERLAP ~	
Q437=+0 ;APPROACH POSITION ~	
Q215=+0 ;MACHINING OPERATION ~	
Q369=+0 ;ALLOWANCE FOR FLOOR ~	
Q338=+20 ;INFEEED FOR FINISHING ~	
Q385=+500 ;FINISHING FEED RATE	
19 L X+50 Y+50 R0 FMAX M99	; Cycle call
20 LBL 0	; End of subprogram
21 END PGM TOUCHPROBE MM	

6.14.2 Example: Measuring a rectangular pocket and recording the results



0 BEGIN PGM TOUCHPROBE_2 MM	
1 TOOL CALL 600 Z	; Tool call: touch probe
2 L Z+100 R0 FMAX	; Retract the touch probe
3 TCH PROBE 423 MEAS. RECTAN. INSIDE ~	
Q273=+50 ;CENTER IN 1ST AXIS ~	
Q274=+40 ;CENTER IN 2ND AXIS ~	
Q282=+90 ;FIRST SIDE LENGTH ~	
Q283=+70 ;2ND SIDE LENGTH ~	
Q261=-5 ;MEASURING HEIGHT ~	
Q320=+2 ;SET-UP CLEARANCE ~	
Q260=+20 ;CLEARANCE HEIGHT ~	
Q301=+0 ;MOVE TO CLEARANCE ~	
Q284=+90.15 ;MAX. LIMIT 1ST SIDE ~	
Q285=+89.95 ;MIN. LIMIT 1ST SIDE ~	
Q286=+70.1 ;MAX. LIMIT 2ND SIDE ~	
Q287=+69.9 ;MIN. LIMIT 2ND SIDE ~	
Q279=+0.15 ;TOLERANCE 1ST CENTER ~	
Q280=+0.1 ;TOLERANCE 2ND CENTER ~	
Q281=+1 ;MEASURING LOG ~	
Q309=+0 ;PGM STOP TOLERANCE ~	
Q330=+0 ;TOOL	
4 L Z+100 R0 FMAX	; Retract the tool, end of program
5 M30	
6 END PGM TOUCHPROBE_2 MM	

7

**Touch Probe Cycles:
Special Functions**

7.1 Fundamentals

7.1.1 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.

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:

Cycle		Call	Further information
3	MEASURING ■ Touch probe cycle for defining OEM cycles	DEF- active	Page 263
4	MEASURING IN 3-D ■ Measuring any position	DEF- active	Page 265
444	PROBING IN 3-D ■ Measuring any position ■ Determining the deviation from the nominal coordinates	DEF- active	Page 268
441	FAST PROBING ■ Touch probe cycle for defining various touch probe parameters	DEF- active	Page 274
1493	EXTRUSION PROBING ■ Touch probe cycle for defining an extrusion ■ Extrusion direction, length, and number of extrusion points can be programmed	DEF- active	Page 276

7.2 Cycle 3 MEASURING

Application

Touch probe cycle **3** measures any position on the workpiece in a selectable probing direction. Unlike other touch probe cycles, Cycle **3** enables you to enter the measuring range **SET UP** and feed rate **F** directly. Also, the touch probe retracts by a definable value **MB** after determining the measured value.

Cycle sequence

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

Notes



The exact behavior of touch probe cycle **3** is defined by your machine 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 sequences through the probe input X12 or X13.

7.2.1 Cycle parameters

Help graphic	Parameter
	<p>Parameter number for result?</p> <p>Enter the number of the Q parameter to which you want the control to assign the first measured coordinate (X). The Y and Z values will be written to the immediately following Q parameters.</p> <p>Input: 0...1999</p>
	<p>Probing axis?</p> <p>Enter the axis in whose direction the touch probe will move and confirm with the ENT key.</p> <p>Input: X, Y, or Z</p>
	<p>Probing angle?</p> <p>Angle measured from the defined probing axis in which the touch probe will move. Confirm with ENT.</p> <p>Input: -180...+180</p>
	<p>Maximum measuring range?</p> <p>Enter the maximum distance from the starting point by which the touch probe will move. Confirm with ENT.</p> <p>Input: -999999999...+999999999</p>
	<p>Feed rate measurement</p> <p>Enter the measuring feed rate in mm/min.</p> <p>Input: 0...3000</p>
	<p>Maximum retraction distance?</p> <p>Traverse path in the direction opposite the probing direction, after the stylus was deflected. The control returns the touch probe to a point no farther than the starting point, so that there can be no collision.</p> <p>Input: 0...999999999</p>
	<p>Reference system? (0=ACT/1=REF)</p> <p>Define whether the probing direction and measurement result will be referenced to the current coordinate system (ACT, can be shifted or rotated) or the machine coordinate system (REF):</p> <p>0: Perform the probing operation in the current system and save the measurement result in the ACT system</p> <p>1: Perform the probing operation in the machine-based REF system. Save the measurement result in the REF system.</p> <p>Input: 0, 1</p>

Help graphic**Parameter****Error mode? (0=OFF/1=ON)**

Define whether the control will issue an error message if the stylus is deflected at cycle start. If mode **1** is selected, the control saves the value **-1** in the 4th result parameter and continues the cycle:

0: Issue error message

1: Do not issue error message

Input: **0, 1**

Example

11 TCH PROBE 3.0 MEASURING

12 TCH PROBE 3.1 Q1

13 TCH PROBE 3.2 X ANGLE:+15

14 TCH PROBE 3.3 ABST+10 F100 MB1 REFERENCE SYSTEM:0

15 TCH PROBE 3.4 ERRORMODE1

7.3 Cycle 4 MEASURING IN 3-D

Application

Touch probe cycle **4** measures any position on the workpiece in the probing direction defined by a vector. Unlike other touch probe cycles, Cycle **4** enables you to enter the probing distance and probing feed rate directly. You can also define the distance by which the touch probe retracts after acquiring the probed value.

Cycle **4** is an auxiliary cycle that can be used for probing with any touch probe (TS or TT). The control does not provide a cycle for calibrating the TS touch probe in any probing direction.

Cycle sequence

- 1 The control moves the touch probe from the current position at the entered feed rate in the defined probing direction. Define the probing direction in the cycle by using a vector (delta values in X, Y and Z).
- 2 After the control has saved the position, the control stops the probe movement. The control saves the X, Y, Z coordinates of the probing position in three successive Q parameters. You define the number of the first parameter in the cycle. If you are using a TS touch probe, the probe result is corrected by the calibrated center offset.
- 3 Finally, the control retracts the touch probe in the direction opposite to the direction of probing. You define the traverse distance in parameter **MB**—the touch probe is moved to a point no farther than the starting point.



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

Notes

NOTICE

Danger of collision!

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

- 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.

7.3.1 Cycle parameters

Help graphic	Parameter
	<p>Parameter number for result?</p> <p>Enter the number of the Q parameter to which you want the control to assign the first measured coordinate (X). The Y and Z values will be written to the immediately following Q parameters.</p> <p>Input: 0...1999</p>
	<p>Relative measuring path in X?</p> <p>X component of the direction vector defining the direction in which the touch probe will move.</p> <p>Input: -999999999...+999999999</p>
	<p>Relative measuring path in Y?</p> <p>Y component of the direction vector defining the direction in which the touch probe will move.</p> <p>Input: -999999999...+999999999</p>
	<p>Relative measuring path in Z?</p> <p>Z component of the direction vector defining the direction in which the touch probe will move.</p> <p>Input: -999999999...+999999999</p>
	<p>Maximum measuring range?</p> <p>Enter the maximum distance from the starting point by which the touch probe will move along the direction vector.</p> <p>Input: -999999999...+999999999</p>
	<p>Feed rate measurement</p> <p>Enter the measuring feed rate in mm/min.</p> <p>Input: 0...3000</p>
	<p>Maximum retraction distance?</p> <p>Traverse path in the direction opposite the probing direction, after the stylus was deflected.</p> <p>Input: 0...999999999</p>
	<p>Reference system? (0=ACT/1=REF)</p> <p>Define whether the result of probing will be saved in the input coordinate system (ACT), or with respect to the machine coordinate system (REF):</p> <p>0: Save the measurement result in the ACT system</p> <p>1: Save the measurement result in the REF system</p> <p>Input: 0, 1</p>

Example

```

11 TCH PROBE 4.0 MEASURING IN 3-D
12 TCH PROBE 4.1 Q1
13 TCH PROBE 4.2 IX-0.5 IY-1 IZ-1
14 TCH PROBE 4.3 ABST+45 F100 MB50 REFERENCE SYSTEM:0

```

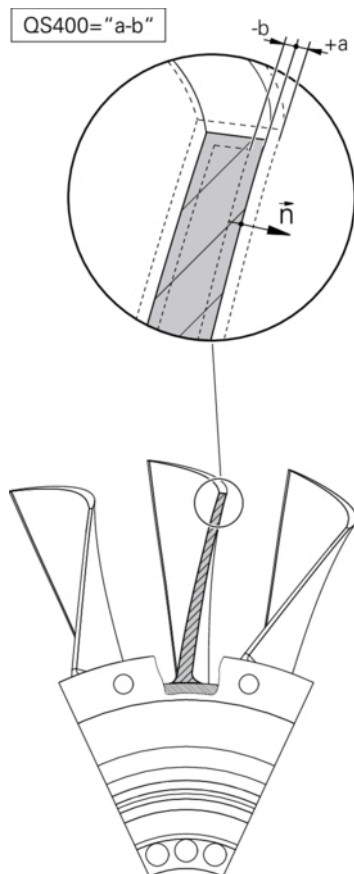
7.4 Cycle 444 PROBING IN 3-D

Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.



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

Cycle sequence

- 1 Starting from the current position, the touch probe traverses to a point on the normal vector that is at the following distance from the nominal coordinate:
Distance = ball-tip radius + **SET_UP** value from the tchprobe.tp table (TNC:\table\tchprobe.tp) + **Q320**. Pre-positioning takes a clearance height into account. **Further information:** "Executing touch probe cycles", Page 50
- 2 The touch probe then approaches the nominal coordinate. The probing distance is defined by DIST, not by the normal vector! The normal vector is only used for the correct calculation of the coordinates.
- 3 After the control has saved the position, the touch probe is retracted and stopped. The control saves the measured coordinates of the contact point in Q parameters.
- 4 Finally, the control retracts the touch probe by the value that you defined in parameter **MB** in the direction opposite to the probing direction.

Result parameters

The control stores the probing results in the following parameters:

Q parameter number	Meaning
Q151	Measured position in main axis
Q152	Measured position in secondary axis
Q153	Measured position in tool axis
Q161	Measured deviation in main axis
Q162	Measured deviation in secondary axis
Q163	Measured deviation in tool axis
Q164	Measured 3-D deviation <ul style="list-style-type: none"> ■ Less than 0: Undersize ■ Greater than 0: Oversize
Q183	Workpiece status: <ul style="list-style-type: none"> ■ - 1 = undefined ■ 0 = good ■ 1 = Rework ■ 2 = Scrap

Log function

Once probing has finished, the control generates a log in HTML format. The log includes the results from the main, secondary, and tool axes as well as the 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 main, secondary, and tool axes:

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

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- In order to obtain exact results from the touch probe being used, you need to perform 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 possible even if the position resulting from the **Tilt working plane** function is inconsistent with the current position of the rotary axes.
- If your machine is equipped with a feedback-controlled spindle, you should activate 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 268.
- The workpiece status (good/rework/scrap) is set via Q parameter **Q183**, independent of parameter **Q309** (see "Application", Page 268).

Note regarding machine parameters

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

7.4.1 Cycle parameters

Help graphic	Parameter
	Q263 1st measuring point in 1st axis? Coordinate of the first touch point in the main axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q294 1st measuring point in 3rd axis? Coordinate of the first touch point in the touch probe axis. The value has an absolute effect. Input: -99999.9999...+99999.9999
	Q581 Surface-normal in ref. axis? Enter here the surface normal in the direction of the main axis. The surface normal of a point is normally output by a CAD/CAM system. Input: -10...+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: -10...+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: -10...+10
	Q320 Set-up clearance? Additional distance between touch point and ball tip. Q320 is in addition to the SET_UP column in the touch probe table. This value has an incremental effect. Input: 0...99999.9999 or PREDEF
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. The value has an absolute effect. Input: -99999.9999...+99999.9999 or PREDEF

Help graphic

Parameter

QS400 Tolerance value?

Specify a tolerance band that will be monitored by the cycle. The tolerance defines the deviation permitted along the surface normal. This deviation is determined between the nominal coordinate and the actual coordinate of the workpiece. (The surface normal is defined by **Q581** to **Q583**, and the nominal coordinate is defined by **Q263**, **Q264**, and **Q294**.) The tolerance value is distributed over the axes, depending on the normal vector (see examples).

Examples

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

Input: Max. **255** characters

Q309 Reaction to tolerance error?

Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message:

- 0:** Do not interrupt program run when tolerance is exceeded; do not output an error message
- 1:** Interrupt program run when tolerance is exceeded and output an error message
- 2:** If the value of the measured actual coordinate along the surface normal vector is less than the nominal coordinate, the control displays a message and interrupts the NC program run. However, there will be no error message if the value of the measured actual coordinate is greater than the nominal coordinate.

Input: **0, 1, 2**

Example

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

7.5 Cycle 441 FAST PROBING

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.

Notes

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

Note regarding machine parameters

- The machine parameter **maxTouchFeed** (no. 122602) allows the machine manufacturer to limit the feed rate. You define the maximum absolute feed rate in this machine parameter.

7.5.1 Cycle parameters

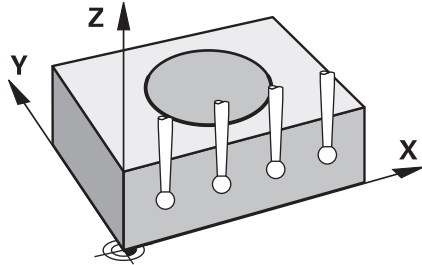
Help graphic	Parameter
	Q396 Positioning feed rate? Define the feed rate at which the touch probe will be moved to the specified positions. Input: 0...99999.999
	Q397 Pre-pos. at machine's rapid? Define whether the control, when prepositioning the touch probe, traverses at FMAX feed rate (machine's rapid traverse): 0: Pre-position at the feed rate from Q396 1: Pre-position at the machine's rapid traverse FMAX Input: 0, 1
	Q399 Angle tracking (0/1)? Define whether the control will orient the touch probe before every probing operation: 0: Do not orient the spindle 1: Orient the spindle before every probing operation (increased accuracy)? Input: 0, 1
	Q400 Automatic interruption? Define whether the control will interrupt program run and output the measurement results on the screen following a touch probe cycle for automatic workpiece measurement: 0: Do not interrupt program run even if, in the specific touch probe cycle, the output of measurement results on the screen is selected 1: Interrupt program run and output measurement results on the screen. You can then resume the NC program run with NC Start . Input: 0, 1

Example

11 TCH PROBE 441 FAST PROBING ~	
Q396=+3000	;POSITIONING FEEDRATE ~
Q397=+0	;SELECT FEED RATE ~
Q399=+1	;ANGLE TRACKING ~
Q400=+1	;INTERRUPTION

7.6 Cycle 1493 EXTRUSION PROBING

Application



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

The repetitions allow you, for example, to perform multiple measurements at different heights and to determine deviations based on the deflection of the tool. You can also use the extrusion to increase the accuracy during probing. Multiple measuring points help you ascertain contamination on the workpiece or rough surfaces.

In order to activate the repetition of specific touch points, you need to define Cycle **1493** before the probing cycle. Depending on the definition, this cycle will remain active for only the next cycle or for the entire NC program. The control interprets the extrusion in the input coordinate system **I-CS**.

The following cycles are capable of performing extrusions:

- **PROBING IN PLANE** (Cycle **1420**, option 17), see Page 67
- **PROBING ON EDGE** (Cycle **1410**), see Page 73
- **PROBING TWO CIRCLES** (Cycle **1411**), see Page 80
- **INCLINED EDGE PROBING** (Cycle **1412**), see Page 88
- **POSITION PROBING** (Cycle **1400**), see Page 121
- **CIRCLE PROBING** (Cycle **1401**), see Page 125

Result parameters

The control stores the results of the probing cycle in the following Q parameters:

Q parameter number	Meaning
Q970	Maximum deviation from the ideal line of touch point 1
Q971	Maximum deviation from the ideal line of touch point 2
Q972	Maximum deviation from the ideal line of touch point 3
Q973	Maximum deviation of diameter 1
Q974	Maximum deviation of diameter 2

QS parameter

In addition to saving the results of the probing cycle in the return parameters **Q97x**, the control saves individual results to the QS parameters **QS97x**. The control saves the results of all measuring points from **one** extrusion in the corresponding QS parameters. Each result is ten characters long and the results are separated from each other by a blank space. This makes it easy for the control to convert the individual values in the NC program via string processing and use them for special automated evaluations.

Result in a QS parameter:

QS970 = "0.12345678 -1.1234567 -2.1234567 -3.12345678"

Further information: User's Manual for Programming and Testing

Log function

Once probing has finished, the control generates a log file in HTML format. The log file contains the results of the 3-D deviation in graphical and tabular form. The control saves the log file in the same folder in which the NC program is located.

The log file contains the following data in the main axis, secondary axis and tool axis depending on the selected cycle (e.g., circle center point and diameter):

- Actual probing direction (as a vector in the input system). The value of the vector corresponds to the configured probing path
- Defined nominal coordinate
- Upper and lower dimensions, as well as the determined deviation along the normal vector
- Measured actual coordinate
- Color coding of the values:
 - Green: Good
 - Orange: Rework
 - Red: Scrap
- Extrusion points

Extrusion points:

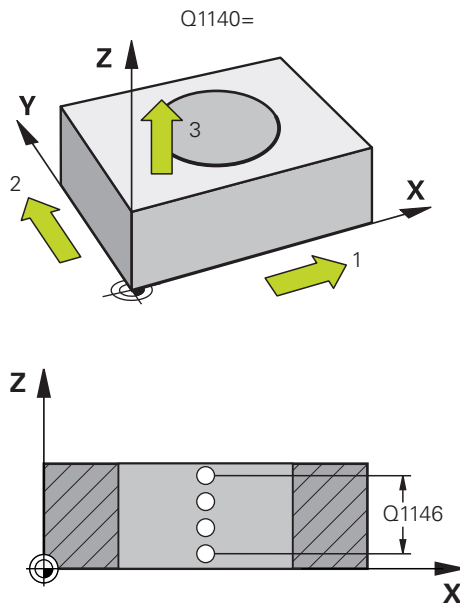
The horizontal axis represents the direction for the extrusion. The blue points are the individual measuring points. The red lines indicate the lower limit and the upper limit of the dimensions. If a value violates a specified tolerance, the control will show the area in red color in the graphic.

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- If **Q1145 > 0** and **Q1146 = 0**, then the control will perform the number of extrusion points at the same position.
- If you use Cycle **1401 CIRCLE PROBING** or **1411 PROBING TWO CIRCLES** to perform an extrusion, the direction for the extrusion must be **Q1140 = +3**; otherwise, the control will output an error message.

7.6.1 Cycle parameters

Help graphic



Parameter

Q1140 Direction for extrusion (1-3)?

- 1: Extrusion in the direction of the main axis
- 2: Extrusion in the direction of the secondary axis
- 3: Extrusion in the direction of the tool axis

Input: 1, 2, 3

Q1145 Number of extrusion points?

Number of measuring points that the cycle repeats over the length of the extrusion Q1146.

Input: 1...99

Q1146 Length of extrusion?

Length over which the measuring points are repeated.

Input: -99...+99

Q1149 Extrusion: Modal duration?

Effect of the cycle:

- 0: The extrusion is effective for only the next cycle.
- 1: The extrusion is effective until the end of the NC program.

Input: -99...+99

Example

11 TCH PROBE 1493 EXTRUSION PROBING ~	
Q1140=+3	;EXTRUSION DIRECTION ~
Q1145=+1	;EXTRUSION POINTS ~
Q1146=+0	;EXTRUSION LENGTH ~
Q1149=+0	;EXTRUSION MODAL

8

**Touch Probe Cycles:
Calibration**

8.1 Fundamentals

8.1.1 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.

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:

- Initial configuration
- Broken stylus
- Stylus replacement
- Change in the probe feed rate
- Irregularities caused, for example, when the machine heats up
- Change of active tool axis

The control assumes the calibration values for the active probe system directly after the calibration process. The updated tool data 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:

Cycle	Call	Further information
461 TS CALIBRATION OF TOOL LENGTH <ul style="list-style-type: none"> ■ Calibrating the length 	DEF- active	Page 282
462 CALIBRATION OF A TS IN A RING <ul style="list-style-type: none"> ■ Measuring the radius using a ring gauge ■ Measuring the center offset using a ring gauge 	DEF- active	Page 284
463 TS CALIBRATION ON STUD <ul style="list-style-type: none"> ■ Measuring the radius using a stud or a calibration pin ■ Measuring the center offset using a stud or a calibration pin 	DEF- active	Page 287
460 CALIBRATION OF TS ON A SPHERE <ul style="list-style-type: none"> ■ Measuring the radius using a calibration sphere ■ Measuring the center offset using a calibration sphere 	DEF- active	Page 290

8.1.2 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:

- Initial configuration
- Broken stylus
- Stylus replacement
- Change in the probe feed rate
- Irregularities caused, for example, when the machine heats up
- Change of active tool axis

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.



- 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.
- Ensure that the touch probe number in the tool table and the touch-probe number in the touch-probe table are identical.

Further information: User's Manual for Setup and Program Run

8.1.3 Displaying calibration values

The control saves the effective length and effective radius of the touch probe in the tool table. The control saves the touch probe center offset to the touch probe table in the columns **CAL_OF1** (main axis) and **CAL_OF2** (secondary axis).

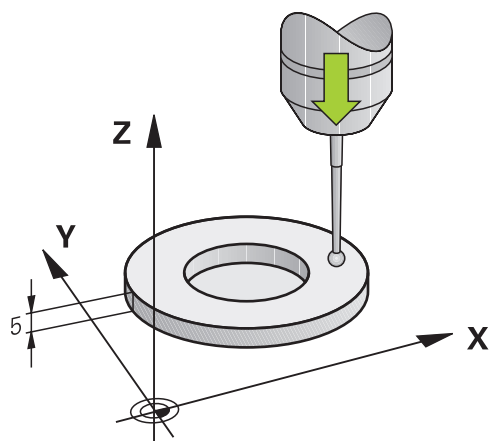
A measuring log is created automatically during calibration. The log file is named **TCHPRAUTO.html**. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe,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

8.2 Cycle 461 TS CALIBRATION OF TOOL LENGTH

Application



Refer to your machine manual.



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

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

Cycle sequence

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

Notes



HEIDENHAIN 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.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

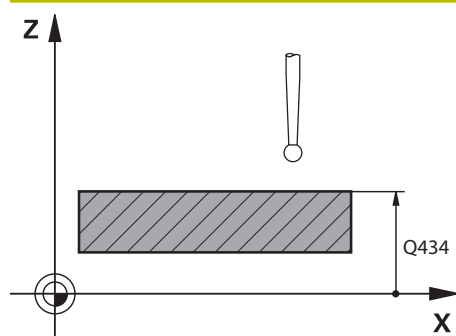
Note on programming

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

8.2.1 Cycle parameters

Cycle parameters

Help graphic



Parameter

Q434 Preset for length?

Preset for the length (e.g., height of the calibration ring). The value has an absolute effect.

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

Example

11 TCH PROBE 461 TS CALIBRATION OF TOOL LENGTH ~

Q434=+5

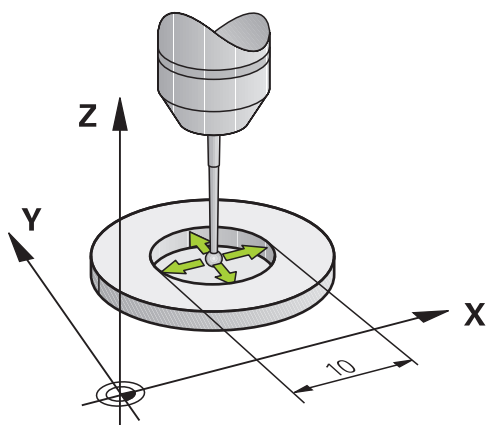
;PRESET

8.3 Cycle 462 CALIBRATION OF A TS IN A RING

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 touch probe table) is determined in addition to the radius by probing from opposite orientations.
- Any orientation possible (e.g. HEIDENHAIN infrared touch probes): Probing operation: see "Orientation possible in two directions").

Notes



In order to be able to determine the ball-tip center offset, the control needs to be specially prepared by the machine manufacturer.

The property of whether or how your touch probe can be oriented is predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine 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.

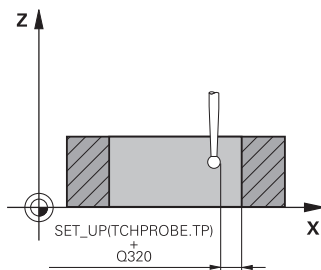
- This cycle can only be executed in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The center offset can be determined only with a suitable touch probe.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

Note on programming

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

8.3.1 Cycle parameters

Help graphic



Parameter

Q407 Radius of ring gauge?

Enter the radius of the ring gauge.

Input: **0.0001...99.9999**

Q320 Set-up clearance?

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

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

Q423 Number of probes?

Number of measuring points on the diameter. The value has an absolute effect.

Input: **3...8**

Q380 Ref. angle in ref. axis?

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

Input: **0...360**

Example

11 TCH PROBE 462 CALIBRATION OF A TS IN A RING ~	
Q407=+5	;RING RADIUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q423=+8	;NO. OF PROBE POINTS ~
Q380=+0	;REFERENCE ANGLE

8.4 Cycle 463 TS CALIBRATION ON STUD

Application



Refer to your machine manual.

Before starting the calibration cycle, you need to pre-position 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 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 touch probe table) 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").

Note:

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.

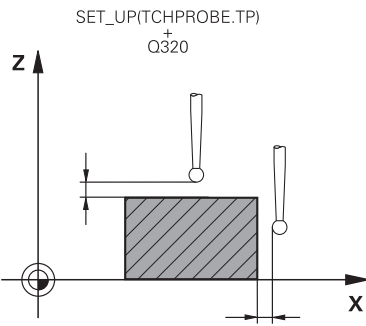
- ▶ The following cycles must not be activated before a touch probe cycle: Cycle **7 DATUM SHIFT**, Cycle **8 MIRRORING**, Cycle **10 ROTATION**, Cycle **11 SCALING FACTOR**, and Cycle **26 AXIS-SPECIFIC SCALING**.
- ▶ Reset any coordinate transformations beforehand.

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

Note on programming

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

8.4.1 Cycle parameters

Help graphic	Parameter
	Q407 Radius of calibr. stud? Diameter of the ring gauge Input: 0.0001...99.9999
	Q320 Set-up clearance? Additional distance between touch point and ball tip. Q320 is in addition to the SET_UP column in the touch probe table. This value has an incremental effect. Input: 0...99999.9999 or PREDEF
	Q301 Move to clearance height (0/1)? Specify how the touch probe moves between measuring points: 0: Move at measuring height between measuring points 1: Move at clearance height between measuring points Input: 0, 1
	Q423 Number of probes? Number of measuring points on the diameter. The value has an absolute effect. Input: 3...8
	Q380 Ref. angle in ref. axis? Angle between the main axis of the working plane and the first touch point. The value has an absolute effect. Input: 0...360

Example

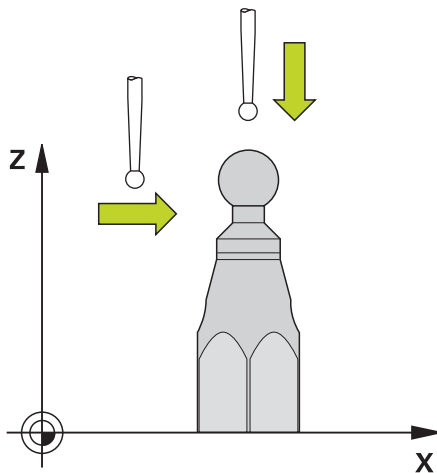
11 TCH PROBE 463 TS CALIBRATION ON STUD ~	
Q407=+5	;STUD RADIUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q301=+1	;MOVE TO CLEARANCE ~
Q423=+8	;NO. OF PROBE POINTS ~
Q380=+0	;REFERENCE ANGLE

8.5 Cycle 460 CALIBRATION OF TS ON A SPHERE (option 17)

Application



Refer to your machine manual.

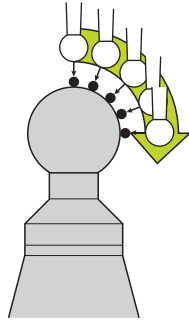


Before starting the calibration cycle, you need to pre-position the touch probe above the center of the calibration sphere. Position the touch probe in the touch probe axis by approximately the set-up clearance (value from touch probe table + value from cycle) above the calibration sphere.

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 "Cycle 444 PROBING IN 3-D ", Page 268).

Cycle sequence



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

Radius calibration Q433=0

- 1 Clamp the calibration sphere. Ensure the prevention of collisions
- 2 In the touch probe axis, position the touch probe over the calibration sphere, and in the working plane, approximately over the sphere center
- 3 The first movement is in the plane, depending on the reference angle (**Q380**)
- 4 The 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

Notes



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

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, 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.
- 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.

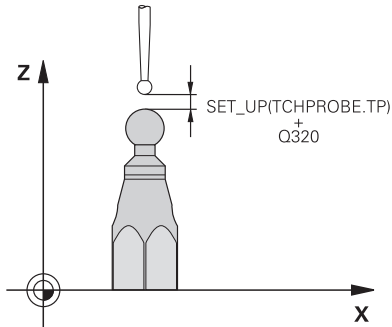
Note on programming

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

8.5.1 Cycle parameters

Cycle parameters

Help graphic



Parameter

Q407 Radius of calib. sphere?

Enter the exact radius of the calibration sphere being used.

Input: **0.0001...99.9999**

Q320 Set-up clearance?

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

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

Q301 Move to clearance height (0/1)?

Specify how the touch probe moves between measuring points:

0: Move at measuring height between measuring points

1: Move at clearance height between measuring points

Input: **0, 1**

Q423 Number of probes?

Number of measuring points on the diameter. The value has an absolute effect.

Input: **3...8**

Q380 Ref. angle in ref. axis?

Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. The value has an absolute effect.

Input: **0...360**

Q433 Calibrate length (0/1)?

Define whether the control will calibrate the touch probe length after radius calibration:

0: Do not calibrate touch probe length

1: Calibrate touch probe length

Input: **0, 1**

Q434 Preset for length?

Coordinate of the calibration sphere center. This value must be defined only if length calibration will be carried out. The value has an absolute effect.

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

Help graphic**Parameter****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: **0...30**

Example

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

9

**Touch Probe
Cycles: Automatic
Kinematics
Measurement**

9.1 Fundamentals (option 48)

9.1.1 Overview



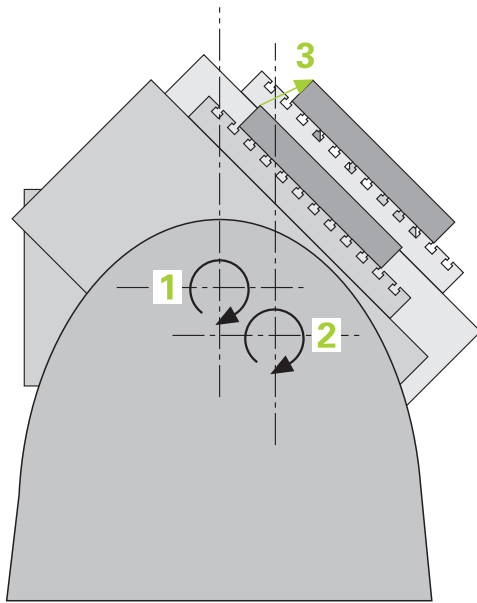
The control must be specifically prepared by the machine tool builder for the use of a 3-D touch probe.

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

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

Cycle	Call	Further information
450 SAVE KINEMATICS (option 48) <ul style="list-style-type: none"> ■ Storing the active machine kinematic configuration ■ Restoring previously saved kinematic configuration 	DEF- active	Page 302
451 MEASURE KINEMATICS (option 48) <ul style="list-style-type: none"> ■ Automatic checking of the machine kinematic configuration ■ Optimizing the machine kinematic configuration 	DEF- active	Page 305
452 PRESET COMPENSATION (option 48) <ul style="list-style-type: none"> ■ Automatic checking of the machine kinematic configuration ■ Optimizing the kinematic transformation chain of the machine 	DEF- active	Page 320
453 KINEMATICS GRID (option 48,option 52) <ul style="list-style-type: none"> ■ Automatic checking depending on the rotary axis position of the machine kinematic configuration ■ Optimizing the machine kinematic configuration 	DEF- active	Page 331

9.1.2 Fundamentals



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

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

The **KinematicsOpt** function of the control is an important component that helps you 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.

9.1.3 Requirements



Refer to your machine manual.
Advanced Function Set 1 (option 8) must have been enabled.
Option 48 must have been enabled.
Machine and control must be specially prepared by the machine manufacturer for use of this cycle.

Prerequisites for using KinematicsOpt:



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

- 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)



HEIDENHAIN recommends using the calibration spheres **KKH 250** (ID number 655475-01) or **KKH 80 (ID number 655475-03)**, which are particularly rigid and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.

9.1.4 Notes



HEIDENHAIN only guarantees the proper operation 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.

NOTICE

Danger of collision!

A change in the kinematics always changes the preset as well. Basic rotations will automatically be reset to 0. There is a danger of collision!

- ▶ After an optimization, reset the preset

Notes about machine parameters

- In the machine parameter **mStrobeRotAxPos** (no. 204803), the machine manufacturer defines the position of the rotary axes. If an M function has been defined in the machine parameter, you have to position the rotary axes to 0° (ACTUAL system) before starting one of the KinematicsOpt cycles (except for **450**).
- If machine parameters were changed through the KinematicsOpt cycles, the control must be restarted. Otherwise the changes could be lost in certain circumstances.

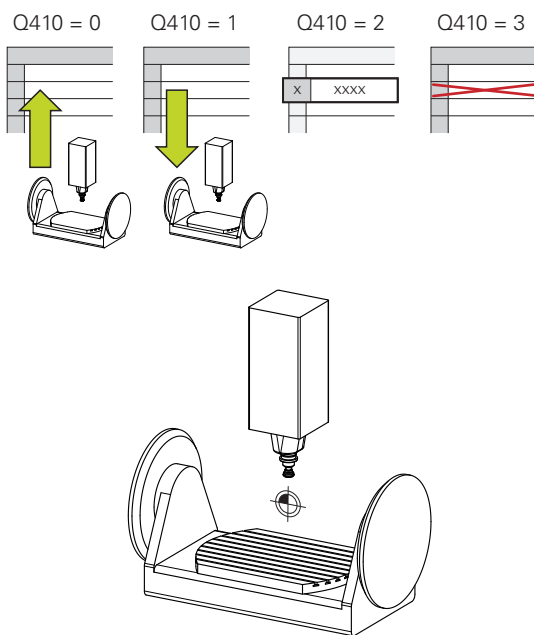
9.2 Cycle 450 SAVE KINEMATICS (option 48)

Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.



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

Notes



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

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

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).

9.2.1 Cycle parameters

Help graphic	Parameter
	Q410 Mode (0/1/2/3)? Define whether a kinematic model will be saved or restored: 0: Save active kinematics 1: Restore saved kinematics 2: Display the current memory status 3: Delete a data record Input: 0, 1, 2, 3
	Q409/QS409 Name of data record? Number or name of data record identifier. 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 selected data records after confirmation (mode 3). You can use the following wildcards in searches: ?: A single, undefined character \$: A single alphabetic character (letter) #: A single, undefined number * : An undefined string of any length Input: 0...99999 or max. 255 characters. A total of 16 memory locations is available.

Saving the current kinematics

11 TCH PROBE 450 SAVE KINEMATICS ~
Q410=+0 ;MODE ~
Q409=+947 ;MEMORY DESIGNATION

Restoring data records

11 TCH PROBE 450 SAVE KINEMATICS ~
Q410=+1 ;MODE ~
Q409=+948 ;MEMORY DESIGNATION

Displaying all saved data records

11 TCH PROBE 450 SAVE KINEMATICS ~
Q410=+2 ;MODE ~
Q409=+949 ;MEMORY DESIGNATION

Deleting data records

11 TCH PROBE 450 SAVE KINEMATICS ~
Q410=+3 ;MODE ~
Q409=+950 ;MEMORY DESIGNATION

9.2.2 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

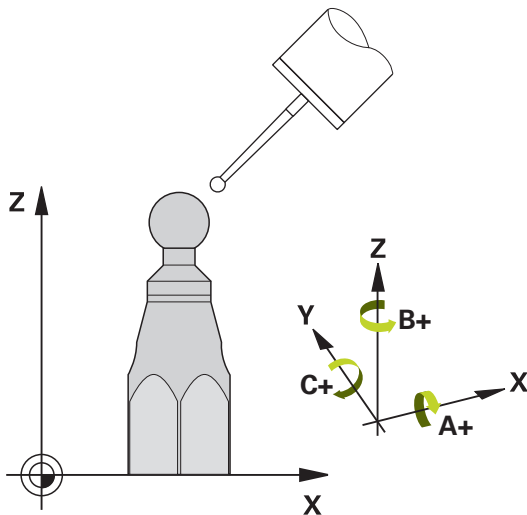
9.3 Cycle 451 MEASURE KINEMATICS (option 48)

Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.



Touch probe cycle **451** enables you to check and, if required, optimize the kinematics of your machine. Use the 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 sequence

- 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:

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

9.3.1 Positioning direction

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

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

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

9.3.2 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 manufacturer, enabling the control to move the rotary axes. The machine manufacturer must have entered the number of the M function in machine parameter **mStrobeRotAxPos** (no. 204803) for this purpose.

- ▶ Note the documentation of the machine tool builder



- 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.

9.3.3 Example calculation of measuring positions for an A axis:

Start angle **Q411** = -30

End angle **Q412** = +90

Number of measuring points **Q414** = 4

Hirth grid = 3°

Calculated stepping angle = $(\mathbf{Q412} - \mathbf{Q411}) / (\mathbf{Q414} - 1)$

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

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

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

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

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

9.3.4 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° .

9.3.5 Choice of the calibration sphere position on the machine table

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

- On machines with rotary tables/tilting tables: Clamp the 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.

9.3.6 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.

9.3.7 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 feedback-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.

9.3.8 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.

9.3.9 Backlash

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

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



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



Programming and operating notes:

- The control does not perform an automatic backlash compensation.
- If the measuring circle radius is < 1 mm, the control does not calculate the backlash. The larger the measuring circle radius, the more accurately the control can ascertain the rotary axis backlash (see "Log function", Page 320).

9.3.10 Notes



Angle compensation is only possible with option 52 KinematicsComp.

NOTICE

Danger of collision!

If you run this cycle, a basic rotation or 3-D basic rotation must not be active. The control will delete the values from the columns **SPA**, **SPB** and **SPC** of the preset table as needed. After the cycle, you need to set a basic rotation or 3-D basic rotation again; otherwise, there is a danger of collision.

- ▶ Deactivate the basic rotation before running the cycle.
- ▶ Set the preset and the basic rotation again after optimization.

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

Notes about machine parameters

- If the optional machine parameter **mStrobeRotAxPos** (no. 204803) is not equal to -1 (M function positions the rotary axis), then start a measurement only if all rotary axes are at 0°.
- In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the optional machine parameter **maxDevCalBall** (no. 204802), the control displays an error message and ends the measurement.
- For angle optimization, the machine manufacturer must adapt the configuration correspondingly.

9.3.11 Cycle parameters

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

Help graphic	Parameter
	<p>Q253 Feed rate for pre-positioning? Define the traversing speed of the tool during pre-positioning in mm/min. Input: 0...99999.9999 or FMAX, FAUTO, PREDEF</p>
	<p>Q380 Ref. angle in ref. axis? Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. The value has an absolute effect. Input: 0...360</p>
	<p>Q411 Starting angle in A axis? Starting angle in the A axis at which the first measurement will be made. The value has an absolute effect. Input: -359.9999...+359.9999</p>
	<p>Q412 End angle in A axis? End angle in the A axis at which the last measurement will be made. The value has an absolute effect. Input: -359.9999...+359.9999</p>
	<p>Q413 Angle of incidence in A axis? Angle of incidence in the A axis at which the other rotary axes will be measured. Input: -359.9999...+359.9999</p>
	<p>Q414 No. of meas. points in A (0...12)? Number of measuring points the control will use to measure the A axis. If the input value = 0, the control does not measure the respective axis. Input: 0...12</p>
	<p>Q415 Starting angle in B axis? Starting angle in the B axis at which the first measurement will be made. The value has an absolute effect. Input: -359.9999...+359.9999</p>
	<p>Q416 End angle in B axis? End angle in the B axis at which the last measurement will be made. The value has an absolute effect. Input: -359.9999...+359.9999</p>
	<p>Q417 Angle of incidence in B axis? Angle of incidence in the B axis at which the other rotary axes will be measured. Input: -359.999...+360.000</p>

Help graphic	Parameter
	Q418 No. of meas. points in B (0...12)? Number of measuring points the control will use to measure the B axis. If the input value = 0, the control does not measure the respective axis. Input: 0...12
	Q419 Starting angle in C axis? Starting angle in the C axis at which the first measurement will be made. The value has an absolute effect. Input: -359.9999...+359.9999
	Q420 End angle in C axis? End angle in the C axis at which the last measurement will be made. The value has an absolute effect. Input: -359.9999...+359.9999
	Q421 Angle of incidence in C axis? Angle of incidence in the C axis at which the other rotary axes will be measured. Input: -359.9999...+359.9999
	Q422 No. of meas. points in C (0...12)? Number of measuring points the control will use to measure the C axis. If the input value = 0, the control does not measure the respective axis. Input: 0...12
	Q423 Number of probes? Define the number of measuring points the control will use to measure the calibration sphere in the plane. Fewer measuring points increase speed and more measuring points increase measurement precision. Input: 3...8
	Q431 Preset (0/1/2/3)? Define whether the control will automatically set the active preset at the center of the sphere: 0: Do not set the preset automatically at the center of the sphere: Set the preset manually before the start of the cycle 1: Set the preset automatically at the center of the sphere before measurement (the active preset will be overwritten): Pre-position the touch probe manually above the calibration sphere before the start of the cycle 2: Set the preset automatically at the center of the sphere after measurement (the active preset will be overwritten): Set the preset manually before the start of the cycle 3: Set the preset at the center of the sphere before and after measurement (the active preset will be overwritten): Pre-position the touch probe manually above the calibration sphere before the start of the cycle Input: 0, 1, 2, 3

Help graphic**Parameter****Q432 Angular range of backlash comp.?**

Define the traversing angle the control will use to measure the rotary axis backlash. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the control does not measure the backlash.

Input: **-3...+3**

Saving and checking the kinematics

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

9.3.12 Various modes (Q406)

Test mode Q406 = 0

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

"Optimize position of rotary axes" mode Q406 = 1

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

Position and Angle Optimization mode Q406 = 2

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



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

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

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



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

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

11 TOOL CALL "TOUCH_PROBE" Z	
12 TCH PROBE 451 MEASURE KINEMATICS ~	
Q406=+1	;MODE ~
Q407=+12.5	;SPHERE RADIUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q408=+0	;RETR. HEIGHT ~
Q253=+750	;F PRE-POSITIONING ~
Q380=+0	;REFERENCE ANGLE ~
Q411=-90	;START ANGLE A AXIS ~
Q412=+90	;END ANGLE A AXIS ~
Q413=+0	;INCID. ANGLE A AXIS ~
Q414=+0	;MEAS. POINTS A AXIS ~
Q415=-90	;START ANGLE B AXIS ~
Q416=+90	;END ANGLE B AXIS ~
Q417=+0	;INCID. ANGLE B AXIS ~
Q418=+4	;MEAS. POINTS B AXIS ~
Q419=+90	;START ANGLE C AXIS ~
Q420=+270	;END ANGLE C AXIS ~
Q421=+0	;INCID. ANGLE C AXIS ~
Q422=+3	;MEAS. POINTS C AXIS ~
Q423=+3	;NO. OF PROBE POINTS ~
Q431=+1	;PRESET ~
Q432=+0.5	;BACKLASH, ANG. RANGE

9.3.13 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)

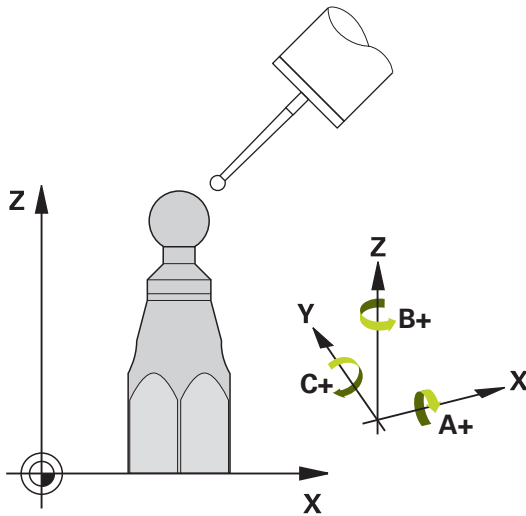
9.4 Cycle 452 PRESET COMPENSATION (option 48)

Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.



Touch probe cycle **452** optimizes the kinematic transformation chain of your machine (see "Cycle 451 MEASURE KINEMATICS (option 48)", Page 305). Then the control corrects the workpiece coordinate system in the kinematics model in such a way that the current preset is at the center of the calibration sphere after optimization.

Cycle sequence



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.

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

Notes



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

NOTICE

Danger of collision!

If you run this cycle, a basic rotation or 3-D basic rotation must not be active. The control will delete the values from the columns **SPA**, **SPB** and **SPC** of the preset table as needed. After the cycle, you need to set a basic rotation or 3-D basic rotation again; otherwise, there is a danger of collision.

- ▶ Deactivate the basic rotation before running the cycle.
- ▶ Set the preset and the basic rotation again after optimization.

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



- If you interrupt the cycle during the measurement, the kinematic data might no longer be in the original condition. Save the active kinematic configuration before an optimization with Cycle **450**, so that in case of a failure the most recently active kinematic configuration can be restored.

Notes about machine parameters

- In the machine parameter **maxModification** (no. 204801), the machine manufacturer defines the permissible limit value for modifications of a transformation. If the kinematics data determined exceed the permissible limit value, the control displays a warning. Then you have to confirm acceptance of the determined values by pressing **NC Start**.
- In the machine parameter **maxDevCalBall** (no. 204802), the machine manufacturer defines the maximum deviation of the calibration sphere radius. In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the machine parameter **maxDevCalBall** (no. 204802), the control displays an error message and ends the measurement.

9.4.1 Cycle parameters

Help graphic	Parameter
	Q407 Radius of calib. sphere? Enter the exact radius of the calibration sphere being used. Input: 0.0001...99.9999
	Q320 Set-up clearance? Additional distance between touch point and ball tip. Q320 is in addition to the SET_UP column in the touch probe table. This value has an incremental effect. Input: 0...99999.9999 or PREDEF
	Q408 Retraction height? 0: Do not move to any retraction height; the control moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C. > 0: Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter Q253 . The value has an absolute effect. Input: 0...99999.9999
	Q253 Feed rate for pre-positioning? Define the traversing speed of the tool during pre-positioning in mm/min. Input: 0...99999.9999 or FMAX, FAUTO, PREDEF
	Q380 Ref. angle in ref. axis? Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. The value has an absolute effect. Input: 0...360
	Q411 Starting angle in A axis? Starting angle in the A axis at which the first measurement will be made. The value has an absolute effect. Input: -359.9999...+359.9999
	Q412 End angle in A axis? End angle in the A axis at which the last measurement will be made. The value has an absolute effect. Input: -359.9999...+359.9999
	Q413 Angle of incidence in A axis? Angle of incidence in the A axis at which the other rotary axes will be measured. Input: -359.9999...+359.9999

Help graphic	Parameter
	Q414 No. of meas. points in A (0...12)? Number of measuring points the control will use to measure the A axis. If the input value = 0, the control does not measure the respective axis. Input: 0...12
	Q415 Starting angle in B axis? Starting angle in the B axis at which the first measurement will be made. The value has an absolute effect. Input: -359.9999...+359.9999
	Q416 End angle in B axis? End angle in the B axis at which the last measurement will be made. The value has an absolute effect. Input: -359.9999...+359.9999
	Q417 Angle of incidence in B axis? Angle of incidence in the B axis at which the other rotary axes will be measured. Input: -359.999...+360.000
	Q418 No. of meas. points in B (0...12)? Number of measuring points the control will use to measure the B axis. If the input value = 0, the control does not measure the respective axis. Input: 0...12
	Q419 Starting angle in C axis? Starting angle in the C axis at which the first measurement will be made. The value has an absolute effect. Input: -359.9999...+359.9999
	Q420 End angle in C axis? End angle in the C axis at which the last measurement will be made. The value has an absolute effect. Input: -359.9999...+359.9999
	Q421 Angle of incidence in C axis? Angle of incidence in the C axis at which the other rotary axes will be measured. Input: -359.9999...+359.9999
	Q422 No. of meas. points in C (0...12)? Number of measuring points the control will use to measure the C axis. If the input value = 0, the control does not measure the respective axis. Input: 0...12
	Q423 Number of probes? Define the number of measuring points the control will use to measure the calibration sphere in the plane. Fewer measuring points increase speed and more measuring points increase measurement precision. Input: 3...8

Help graphic**Parameter****Q432 Angular range of backlash comp.?**

Define the traversing angle the control will use to measure the rotary axis backlash. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the control does not measure the backlash.

Input: **-3...+3**

Calibration program

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

9.4.2 Adjustment of interchangeable heads



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

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

Adjusting an interchangeable head

11 TOOL CALL "TOUCH_PROBE" Z	
12 TCH PROBE 452 PRESET COMPENSATION ~	
Q407=+12.5	;SPHERE RADIUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q408=+0	;RETR. HEIGHT ~
Q253=+2000	;F PRE-POSITIONING ~
Q380=+45	;REFERENCE ANGLE ~
Q411=-90	;START ANGLE A AXIS ~
Q412=+90	;END ANGLE A AXIS ~
Q413=+45	;INCID. ANGLE A AXIS ~
Q414=+4	;MEAS. POINTS A AXIS ~
Q415=-90	;START ANGLE B AXIS ~
Q416=+90	;END ANGLE B AXIS ~
Q417=+0	;INCID. ANGLE B AXIS ~
Q418=+2	;MEAS. POINTS B AXIS ~
Q419=+90	;START ANGLE C AXIS ~
Q420=+270	;END ANGLE C AXIS ~
Q421=+0	;INCID. ANGLE C AXIS ~
Q422=+0	;MEAS. POINTS C AXIS ~
Q423=+4	;NO. OF PROBE POINTS ~
Q432=+0	;BACKLASH, ANG. RANGE

The goal of this procedure is 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

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

9.4.3 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

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

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

Drift compensation

11 TOOL CALL "TOUCH_PROBE" Z	
13 TCH PROBE 452 PRESET COMPENSATION ~	
Q407=+12.5	;SPHERE RADIUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q408=+0	;RETR. HEIGHT ~
Q253=+9999	;F PRE-POSITIONING ~
Q380=+45	;REFERENCE ANGLE ~
Q411=-90	;START ANGLE A AXIS ~
Q412=+90	;END ANGLE A AXIS ~
Q413=+45	;INCID. ANGLE A AXIS ~
Q414=+4	;MEAS. POINTS A AXIS ~
Q415=-90	;START ANGLE B AXIS ~
Q416=+90	;END ANGLE B AXIS ~
Q417=+0	;INCID. ANGLE B AXIS ~
Q418=+2	;MEAS. POINTS B AXIS ~
Q419=+90	;START ANGLE C AXIS ~
Q420=+270	;END ANGLE C AXIS ~
Q421=+0	;INCID. ANGLE C AXIS ~
Q422=+3	;MEAS. POINTS C AXIS ~
Q423=+3	;NO. OF PROBE POINTS ~
Q432=+0	;BACKLASH, ANG. RANGE

9.4.4 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 320)

9.5 Cycle 453 KINEMATICS GRID (option 48), (option 52)

Application



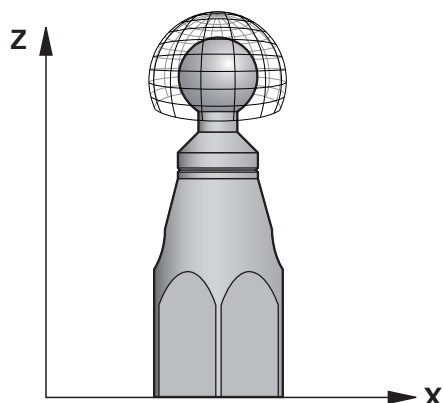
Refer to your machine manual.

KinematicsOpt (software option 48) is required.

KinematicsComp (software option 52) is required.

This function must be enabled and adapted by the machine manufacturer.

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 sequence

- 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.

9.5.1 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

9.5.2 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.

9.5.3 Notes

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

This function must be enabled and adapted by the machine manufacturer.

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

NOTICE**Danger of collision!**

If you run this cycle, a basic rotation or 3-D basic rotation must not be active. The control will delete the values from the columns **SPA**, **SPB** and **SPC** of the preset table as needed. After the cycle, you need to set a basic rotation or 3-D basic rotation again; otherwise, there is a danger of collision.

- ▶ Deactivate the basic rotation before running the cycle.
 - ▶ Set the preset and the basic rotation again after optimization.
- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
 - Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.
 - As with Cycles **451** and **452**, Cycle **453** ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.
 - Before defining the cycle, you must set the preset to the center of the calibration sphere and activate it, or you set input parameter **Q431** to 1 or 3, respectively.
 - For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while probe monitoring is inactive.
 - Programming in inches: The control always records the log data and results of measurement in millimeters.
 - If you have activated preset setting before the calibration (**Q431** = 1/3), then move the touch probe by the set-up clearance (**Q320** + **SET_UP**) to a position approximately above the center of the calibration sphere before the start of the cycle.



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

Notes about machine parameters

- In the machine parameter **mStrobeRotAxPos** (no. 204803), the machine manufacturer defines the maximum permissible modification of a transformation. If the value is not equal to -1 (M function positions the rotary axis), then start a measurement only if all rotary axes are at 0°.
- In the machine parameter **maxDevCalBall** (no. 204802), the machine manufacturer defines the maximum deviation of the calibration sphere radius. In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the machine parameter **maxDevCalBall** (no. 204802), the control displays an error message and ends the measurement.

9.5.4 Cycle parameters

Help graphic	Parameter
	<p>Q406 Mode (-1/0/+1)</p> <p>Define whether the control will write a value of 0 to the values of the compensation table (*.kco), will check the currently existing deviations, or will perform a compensation. A log file (*.html) is created.</p> <p>-1: Delete values in the compensation table (*.kco). The compensation values for TCP positioning errors are set to 0 in the compensation table (*.kco). The control will not perform any probing. No results will be output to the log (*.html).</p> <p>0: Check TCP positioning errors. The control measures the TCP positioning errors based on the rotary axis positions but does not write values to the compensation table (*.kco). The control displays the standard and maximum deviation in a log (*.html).</p> <p>1: Compensate TCP positioning errors. The control measures the TCP positioning errors based on the rotary axis positions and writes the deviations to the compensation table (*.kco). The compensations are then immediately effective. The control displays the standard and maximum deviation in a log (*.html).</p> <p>Input: -1, 0, +1</p>
	<p>Q407 Radius of calib. sphere?</p> <p>Enter the exact radius of the calibration sphere being used.</p> <p>Input: 0.0001...99.9999</p>
	<p>Q320 Set-up clearance?</p> <p>Additional distance between touch point and ball tip. Q320 is in addition to the SET_UP column in the touch probe table. This value has an incremental effect.</p> <p>Input: 0...99999.9999 or PREDEF</p>
	<p>Q408 Retraction height?</p> <p>0: Do not move to any retraction height; the control moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C.</p> <p>> 0: Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter Q253. The value has an absolute effect.</p> <p>Input: 0...99999.9999</p>
	<p>Q253 Feed rate for pre-positioning?</p> <p>Define the traversing speed of the tool during pre-positioning in mm/min.</p> <p>Input: 0...99999.9999 or FMAX, FAUTO, PREDEF</p>

Help graphic**Parameter****Q380 Ref. angle in ref. axis?**

Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. The value has an absolute effect.

Input: **0...360**

Q423 Number of probes?

Define the number of measuring points the control will use to measure the calibration sphere in the plane. Fewer measuring points increase speed and more measuring points increase measurement precision.

Input: **3...8**

Q431 Preset (0/1/2/3)?

Define whether the control will automatically set the active preset at the center of the sphere:

0: Do not set the preset automatically at the center of the sphere: Set the preset manually before the start of the cycle

1: Set the preset automatically at the center of the sphere before measurement (the active preset will be overwritten): Pre-position the touch probe manually above the calibration sphere before the start of the cycle

2: Set the preset automatically at the center of the sphere after measurement (the active preset will be overwritten): Set the preset manually before the start of the cycle

3: Set the preset at the center of the sphere before and after measurement (the active preset will be overwritten): Pre-position the touch probe manually above the calibration sphere before the start of the cycle

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

Probing with Cycle 453

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

9.5.5 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

10

**Touch Probe Cycles:
Automatic Tool
Measurement**

10.1 Fundamentals

10.1.1 Overview



Refer to your machine manual.
Some cycles and functions may not be provided on your machine.
Option 17 is required.
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.

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

Cycle	Call	Further information
480 CALIBRATE TT 30 ■ Calibrating the tool touch probe	DEF- active	Page 344
481 CAL. TOOL LENGTH 31 ■ Measuring the tool length	DEF- active	Page 346
482 CAL. TOOL RADIUS 32 ■ Measuring the tool radius	DEF- active	Page 349
483 MEASURE TOOL 33 ■ Measuring the tool length and radius	DEF- active	Page 353
484 CALIBRATE IR TT ■ Calibrating the tool touch probe (e.g., infrared tool touch probe)	DEF- active	Page 356
485 MEASURE LATHE TOOL (option 50) ■ Measurement of turning tools	DEF- active	Page 360

10.1.2 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**:

- Instead of a selectable parameter for the status of the measurement, Cycles **481** to **483** use the fixed parameter **Q199**.

10.1.3 Setting machine parameters



The touch probe cycles **480**, **481**, **482**, **483**, **484** can be hidden with the optional **hideMeasureTT** machine parameter (no. 128901).



Programming and operating notes:

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

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

The spindle speed is calculated as follows:

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

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

The probing feed rate is calculated as follows:

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

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

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

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

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

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

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

Tool radius	Measuring tolerance
Up to 30 mm	measureTolerance1
30 to 60 mm	$2 \cdot \text{measureTolerance1}$
60 to 90 mm	$3 \cdot \text{measureTolerance1}$
90 to 120 mm	$4 \cdot \text{measureTolerance1}$

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

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

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

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

10.1.4 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.

10.2 Cycle 30 or 480 CALIBRATE TT

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 341). 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.

You calibrate the TT with touch probe cycle **30** or **480**.

Touch probe

For the touch probe you use a spherical or cuboid probe contact

Cuboid probe contact

For a cuboid probe contact, the machine manufacturer can store in the optional machine parameters **detectStylusRot** (no. 114315) and **tippingTolerance** (no. 114319) whether the angle of misalignment and tilt angle are determined.

Determining the angle of misalignment enables compensation for it when measuring tools. The control displays a warning if the tilt angle is exceeded. The values determined can be seen in the status display of the **TT**.

Further information: User's Manual for Setup and Program Run



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 sequence

- 1 Clamp the calibration tool. The calibration tool must be a precisely cylindrical part, for example a cylindrical pin
- 2 Manually position the calibration tool in the working plane over the center of the TT
- 3 Position the calibration tool in the tool axis at approximately 15 mm plus set-up clearance over the TT
- 4 The first movement of the tool is along the tool axis. The tool is first moved to clearance height, i.e. set-up clearance + 15 mm.
- 5 The calibration process along the tool axis starts
- 6 This is followed by calibration in the working plane
- 7 The control positions the calibration tool in the working plane at a position of TT radius + set-up clearance + 11 mm
- 8 Then the control moves the tool downwards along the tool axis and the calibration process starts
- 9 During probing, the control moves in a square pattern
- 10 The control saves the calibration values and considers them during subsequent tool measurement
- 11 The control then retracts the stylus along the tool axis to set-up clearance and moves it to the center of the TT

Notes

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before calibrating the touch probe, you must enter the exact length and radius of the calibration tool into the TOOL.T tool table.

Notes about machine parameters

- Use the machine parameter **CfgTTRoundStylus** (no. 114200) or **CfgT-TRectStylus** (no. 114300) to define the functionality of the calibration cycle. Refer to your machine manual.
 - Use the machine parameter **centerPos** to define the position of the TT within the machine's working space.
- The TT needs to be recalibrated if you change the position of the TT on the table and/or a **centerPos** machine parameter.
- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.

10.2.1 Cycle parameters

Help graphic	Parameter
	<p>Q260 Clearance height?</p> <p>Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height value that the tool tip would lie below the top of the probe contact, the control automatically positions the calibration tool above the top of the probe contact (safety zone from safetyDistToolAx (no. 114203)).</p> <p>Input: -99999.9999...+99999.9999</p>

Example of new format

11 TOOL CALL 12 Z
12 TCH PROBE 480 CALIBRATE TT ~
Q260=+100 ;CLEARANCE HEIGHT

Example of old format

11 TOOL CALL 12 Z
12 TCH PROBE 30.0 CALIBRATE TT
13 TCH PROBE 30.1 HEIGHT:+90

10.3 Cycle 31 or 481 CAL. TOOL LENGTH

Application



Refer to your machine manual!

If you want to measure the tool length, program the touch probe cycle **31** or **482** (see "Differences between Cycles 30 to 33 and Cycles 480 to 483", Page 341). Input parameters allow you to select which of the three following methods will be used to measure the tool length:

- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the tool while it is rotating.
- If the tool diameter is smaller than the diameter of the measuring surface of the TT, or if you are measuring the length of a drill or spherical cutter, you measure the tool while it is stationary.
- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the individual teeth of the tool while it is stationary.

Cycle for measuring a tool during rotation

The control determines the longest tooth of a rotating tool by positioning the tool to be measured at an offset to the center of the touch probe and then moving it toward the measuring surface of the TT until it contacts the surface. The offset is programmed in the tool table under Tool offset: Radius (**R-OFFS**).

Cycle for measuring a stationary tool (e.g. for drills)

The control positions the tool to be measured above the center of the measuring surface. It then moves the non-rotating tool toward the measuring surface of the TT until contact is made. For this measurement, enter 0 in the tool table under Tool offset: radius (**R-OFFS**).

Cycle for measuring individual teeth

The control pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the tool to the upper edge of the touch probe head is defined in **offsetToolAxis** (no. 122707). You can enter an additional offset in Tool offset: Length (**L-OFFS**) in the tool table. The control probes the tool radially while it is rotating to determine the starting angle for measuring the individual teeth. It then measures the length of each tooth by changing the corresponding angle of spindle orientation. To activate this function, set the parameter **PROBING THE TEETH** = 1 in Cycle **31**.

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

If you set **stopOnCheck** (no. 122717) to **FALSE**, the control does not evaluate the result parameter **Q199** and the NC program is not stopped if the breakage tolerance is exceeded. There is a danger of collision!

- ▶ Set **stopOnCheck** (no. 122717) to **TRUE**
- ▶ You must then take steps to ensure that the NC program stops if the breakage tolerance is exceeded

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.
- You can run an individual tooth measurement for tools with **up to 20 teeth**.
- Cycles **31** and **481** do not support touch probes, turning or dressing tools.

Measuring grinding tools


- The cycle takes into account the basic and compensation data from the **TOOL-GRIND.GRD** table, as well as the wear and compensation data (**LBREAK** and **LTOL**) from the **TOOL.T** table.

Q340: 0 and 1

- This cycle will modify compensation or basic data, depending on whether or not an initial dressing operation (**INIT_D**) is defined. This cycle will enter the values automatically at the correct locations in the **TOOLGRIND.GRD** table.

Note the following sequence for setting up grinding tools. **Further information:** User's Manual for Setup and Program Run

10.3.1 Cycle parameters

Help graphic	Parameter
	<p>Q340 Tool measurement mode (0-2)?</p> <p>Define whether and how the measured data will be entered in the tool table.</p> <p>0: The measured tool length is written to column L of tool table TOOL.T, and the tool compensation is set to DL = 0. If there is already a value in TOOL.T, it will be overwritten.</p> <p>1: The measured tool length is compared to the tool length L from TOOL.T. The control calculates the deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation is also available in the Q parameter Q115. If the delta value is greater than the permissible tool length tolerance for wear or break detection, the control will lock the tool (status L in TOOL.T).</p> <p>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.</p> <p>Input: 0, 1, 2</p> <div style="border: 1px solid black; padding: 5px; margin-top: 10px;"> <p> Note the behavior with grinding tools, see "Measuring grinding tools", Page 347</p> </div> <p>Q260 Clearance height?</p> <p>Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from safetyDistStylus).</p> <p>Input: -99999.9999...+99999.9999</p> <p>Q341 Probe the teeth? 0=no/1=yes</p> <p>Define whether the control will measure the individual teeth (maximum of 20 teeth)</p> <p>Input: 0, 1</p>

Example of new format

11 TOOL CALL 12 Z	
12 TCH PROBE 481 CAL. TOOL LENGTH ~	
Q340=+1	;CHECK ~
Q260=+100	;CLEARANCE HEIGHT ~
Q341=+1	;PROBING THE TEETH

Cycle **31** includes an additional parameter:

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

Measuring a rotating tool for the first time; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 31.0 CAL. TOOL LENGTH
13 TCH PROBE 31.1 CHECK:0
14 TCH PROBE 31.2 HEIGHT::+120
15 TCH PROBE 31.3 PROBING THE TEETH:0
```

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
11 TOOL CALL 12 Z
12 TCH PROBE 31.0 CAL. TOOL LENGTH
13 TCH PROBE 31.1 CHECK:1 Q5
14 TCH PROBE 31.2 HEIGHT:+120
15 TCH PROBE 31.3 PROBING THE TEETH:1
```

10.4 Cycle 32 or 482 CAL. TOOL RADIUS

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 341). Input parameters allow you to select which of the two following methods will be used to measure the tool radius:

- 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.

Notes

NOTICE

Danger of collision!

If you set **stopOnCheck** (no. 122717) to **FALSE**, the control does not evaluate the result parameter **Q199** and the NC program is not stopped if the breakage tolerance is exceeded. There is a danger of collision!

- ▶ Set **stopOnCheck** (no. 122717) to **TRUE**
- ▶ You must then take steps to ensure that the NC program stops if the breakage tolerance is exceeded

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before measuring a tool for the first time, enter the following data on the tool into the **TOOL.T** tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.
- Cycles **32** and **482** do not support touch probes, turning or dressing tools.

Measuring grinding tools

- The cycle takes into account the basic and compensation data from the **TOOL-GRIND.GRD** table, as well as the wear and compensation data (**RBREAK** and **RTOL**) from the **TOOL.T** table.

Q340: 0 and 1

- This cycle will modify compensation or basic data, depending on whether or not an initial dressing operation (**INIT_D**) is defined. This cycle will enter the values automatically at the correct locations in the **TOOLGRIND.GRD** table.

Note the following sequence for setting up grinding tools. **Further information:** User's Manual for Setup and Program Run

Notes about machine parameters

- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.
- Cylindrical tools with diamond surfaces can be measured while the spindle is stationary. To do so, in the tool table define the number of teeth **CUT** as 0 and adjust the machine parameter **CfgTT**. Refer to your machine manual.

10.4.1 Cycle parameters

Help graphic	Parameter
	<p>Q340 Tool measurement mode (0-2)?</p> <p>Define whether and how the measured data will be entered in the tool table.</p> <p>0: The measured tool radius is written to column R of the TOOL.T tool table, and the tool compensation is set to DR = 0. If there is already a value in TOOL.T, it will be overwritten.</p> <p>1: The measured tool radius is compared to the tool radius R from TOOL.T. The control calculates the deviation from the stored value and enters it into TOOL.T as the delta value DR. The deviation is also available in the Q parameter Q116. If the delta value is greater than the permissible tool radius tolerance for wear or break detection, the control will lock the tool (status L in TOOL.T).</p> <p>2: The measured tool radius is compared to the tool radius from TOOL.T. The control calculates the deviation from the stored value and writes it to Q parameter Q116. Nothing is entered under R or DR in the tool table.</p> <p>Input: 0, 1, 2</p>
	<p>Q260 Clearance height?</p> <p>Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from safetyDistStylus).</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q341 Probe the teeth? 0=no/1=yes</p> <p>Define whether the control will measure the individual teeth (maximum of 20 teeth)</p> <p>Input: 0, 1</p>

Example of new format

11 TOOL CALL 12 Z	
12 TCH PROBE 482 CAL. TOOL RADIUS ~	
Q340=+1	;CHECK ~
Q260=+100	;CLEARANCE HEIGHT ~
Q341=+1	;PROBING THE TEETH

Cycle **32** includes an additional parameter:

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

Measuring a rotating tool for the first time; old format

11 TOOL CALL 12 Z
12 TCH PROBE 32.0 CAL. TOOL RADIUS
13 TCH PROBE 32.1 CHECK:0
14 TCH PROBE 32.2 HEIGHT:+120
15 TCH PROBE 32.3 PROBING THE TEETH:0

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

11 TOOL CALL 12 Z
12 TCH PROBE 32.0 CAL. TOOL RADIUS
13 TCH PROBE 32.1 CHECK:1 Q5
14 TCH PROBE 32.2 HEIGHT:+120
15 TCH PROBE 32.3 PROBING THE TEETH:1

10.5 Cycle 33 or 483 MEASURE TOOL

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 341). This cycle is particularly suitable for the first measurement of tools, as it saves time when compared with individual measurement of length and radius. Input parameters allow you to select which of the two following methods will be used to measure the tool:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth

Measuring the tool while it is rotating:

The control measures the tool in a fixed programmed sequence. First, if possible, it measures the tool length, and then the tool radius.

Measuring the individual teeth:

The control measures the tool in a fixed programmed sequence. First it measures the tool radius, then the tool length. The sequence of measurement is the same as for touch probe cycles **31** and **32** as well as **481** and **482**.

Notes

NOTICE

Danger of collision!

If you set **stopOnCheck** (no. 122717) to **FALSE**, the control does not evaluate the result parameter **Q199** and the NC program is not stopped if the breakage tolerance is exceeded. There is a danger of collision!

- ▶ Set **stopOnCheck** (no. 122717) to **TRUE**
- ▶ You must then take steps to ensure that the NC program stops if the breakage tolerance is exceeded

- This cycle can only be executed in the **FUNCTION MODE MILL** machining mode.
- Before measuring a tool for the first time, enter the following data on the tool into the **TOOL.T** tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.
- Cycles **33** and **483** do not support touch probes, turning or dressing tools.

Measuring grinding tools

- The cycle takes into account the basic and compensation data from the **TOOL-GRIND.GRD** table, as well as the wear and compensation data (**LBREAK**, **RBREAK**, **LTOL**, and **RTOL**) from the **TOOL.T** table.

Q340: 0 and 1

- This cycle will modify compensation or basic data, depending on whether or not an initial dressing operation (**INIT_D**) is defined. This cycle will enter the values automatically at the correct locations in the **TOOLGRIND.GRD** table.

Note the following sequence for setting up grinding tools. **Further information:** User's Manual for Setup and Program Run

Notes about machine parameters

- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.
- Cylindrical tools with diamond surfaces can be measured while the spindle is stationary. To do so, in the tool table define the number of teeth **CUT** as 0 and adjust the machine parameter **CfgTT**. Refer to your machine manual.

10.5.1 Cycle parameters

Help graphic	Parameter
	<p>Q340 Tool measurement mode (0-2)?</p> <p>Define whether and how the measured data will be entered in the tool table.</p> <p>0: The measured tool length and the measured tool radius are written to columns L and R of the TOOL.T tool table, and the tool compensation is set to DL = 0 and DR = 0. If there is already a value in TOOL.T, it will be overwritten.</p> <p>1: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R in TOOL.T. The control calculates the deviation from the stored value and enters them into TOOL.T as the delta values DL and DR. The deviation is also available in the Q parameters Q115 and Q116. If the delta value is greater than the permissible tool length or tool radius tolerance for wear or break detection, the control will lock the tool (status L in TOOL.T).</p> <p>2: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R in TOOL.T. The control calculates the deviation from the stored values and writes it to the Q parameter Q115 or Q116. Nothing is entered under L, R, or DL, DR in the tool table.</p> <p>Input: 0, 1, 2</p>
	<p>Q260 Clearance height?</p> <p>Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from safetyDistStylus).</p> <p>Input: -99999.9999...+99999.9999</p>
	<p>Q341 Probe the teeth? 0=no/1=yes</p> <p>Define whether the control will measure the individual teeth (maximum of 20 teeth)</p> <p>Input: 0, 1</p>

Example of new format

11 TOOL CALL 12 Z	
12 TCH PROBE 483 MEASURE TOOL ~	
Q340=+1	;CHECK ~
Q260=+100	;CLEARANCE HEIGHT ~
Q341=+1	;PROBING THE TEETH

Cycle **33** includes an additional parameter:

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

Measuring a rotating tool for the first time; old format

11 TOOL CALL 12 Z
12 TCH PROBE 33.0 MEASURE TOOL
13 TCH PROBE 33.1 CHECK:0
14 TCH PROBE 33.2 HEIGHT:+120
15 TCH PROBE 33.3 PROBING THE TEETH:0

Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

11 TOOL CALL 12 Z
12 TCH PROBE 33.0 MEASURE TOOL
13 TCH PROBE 33.1 CHECK:1 Q5
14 TCH PROBE 33.2 HEIGHT:+120
15 TCH PROBE 33.3 PROBING THE TEETH:1

10.6 Cycle 484 CALIBRATE IR TT

Application

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

- **With manual intervention:** If you define **Q536** = 0, then the control will stop before the calibration process. You then need to position the calibration tool manually above the center of the tool touch probe.
- **Without manual intervention:** If you define **Q536** = 1, then the control will automatically execute the cycle. You may have to program a prepositioning movement before. This depends on the value of the parameter **Q523 POSITION TT**.

Cycle sequence



Refer to your machine manual.
The machine manufacturer defines the functionality of the cycle.

To calibrate the tool touch probe, program the touch probe cycle **484**. In input parameter **Q536**, you can specify whether you want to run the cycle with or without manual intervention.

Touch probe

For the touch probe you use a spherical or cuboid probe contact

Cuboid probe contact:

For a cuboid probe contact, the machine manufacturer can store in the optional machine parameters **detectStylusRot** (no. 114315) and **tippingTolerance** (no. 114319) whether the angle of misalignment and tilt angle are determined. Determining the angle of misalignment enables compensation for it when measuring tools. The control displays a warning if the tilt angle is exceeded. The values determined can be seen in the status display of the **TT**.

Further information: User's Manual for Setup and Program Run



When clamping the tool touch probe, make sure that the edges of the cuboid probe contact are aligned as parallel to the machine axes as possible. The angle of misalignment should be less than 1° and the tilt angle should be less than 0.3°.

Calibration tool:

The calibration tool must be a precisely cylindrical part, for example a cylindrical pin. Enter the exact length and radius of the calibration tool into the TOOL.T tool table. After the calibration, the control stores the calibration values and takes them into account during subsequent tool measurements. The calibration tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck.

Q536 = 0: With manual intervention before calibration

Proceed as follows:

- ▶ Insert the calibration tool
- ▶ Start the calibration cycle
- > The control interrupts the calibration cycle and displays a dialog.
- ▶ Manually position the calibration tool above the center of the tool touch probe.



Ensure that the calibration tool is located above the measuring surface of the probe contact.

- ▶ Press **NC start** to resume cycle sequence
- > If you have programmed **Q523 = 2**, then the control writes the calibrated position to the machine parameter **centerPos** (no. 114200)

Q536 = 1: Without manual intervention before calibration

Proceed as follows:

- ▶ Insert the calibrating tool
- ▶ Position the calibration tool above the center of the tool touch probe before the start of the cycle.



- Ensure that the calibration tool is located above the measuring surface of the probe contact.
- For a calibration process without manual intervention, you do not need to position the calibration tool above the center of the tool touch probe. The cycle adopts the position from the machine parameters and automatically moves the tool to this position.

- ▶ Start the calibration cycle
- > The calibration cycle is executed without stopping.
- > If you have programmed **Q523 = 2**, then the control writes the calibrated position to the machine parameter **centerPos** (no. 114200).

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

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.

Note regarding machine parameters

- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual teeth measurements.

10.6.1 Cycle parameters

Help graphic	Parameter
	<p>Q536 Stop before running (0=Stop)?</p> <p>Define whether the control will stop before the calibration process or whether the cycle will automatically be executed without a stop:</p> <p>0: Stop before the calibration process. The control prompts you to position the calibration tool manually above the tool touch probe. After moving the tool to the approximate position above the tool touch probe, press NC Start to continue the calibration process or press the CANCEL button to cancel the calibration process.</p> <p>1: Without stopping before the calibration process. The control starts the calibration process depending on Q523. Before running Cycle 484, you may have to position the tool above the tool touch probe.</p> <p>Input: 0, 1</p>
	<p>Q523 Position of tool probe (0-2)?</p> <p>Position of the tool touch probe:</p> <p>0: Current position of the calibration tool. The tool touch probe is below the current position of the calibration tool. If Q536 = 0, position the calibration tool manually above the center of the tool touch probe during the cycle. If Q536 = 1, you need to position the calibration tool above the center of the tool touch probe before the start of the cycle.</p> <p>1: Configured position of the tool touch probe. The control adopts the position from the machine parameter centerPos (no. 114201). You do not need to preposition the tool. The calibration tool approaches the position automatically.</p> <p>2: Current position of the calibration tool. See Q523 = 0.</p> <p>0: The control additionally writes the determined position (where applicable) to the machine parameter centerPos (no. 114201) after calibration.</p> <p>Input: 0, 1, 2</p>

Example

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

10.7 Cycle 485 MEASURE LATHE TOOL (option 50)

Application



Refer to your machine manual!

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

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

Cycle sequence

- 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 main axis; traverse movement is interpolated in the main 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:

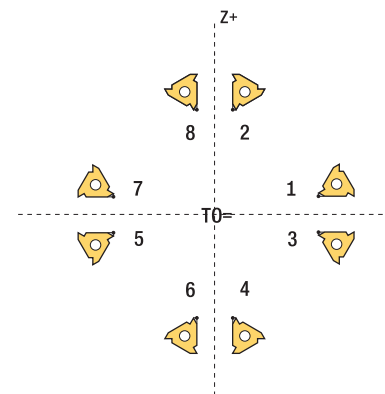
Abbr.	Entries	Dialog
ZL	Tool length 1 (Z direction)	Tool length 1?
XL	Tool length 2 (X direction)	Tool length 2?
DZL	Delta value of tool length 1 (Z direction), is added to ZL	Oversize in tool length 1?
DXL	Delta value of tool length 2 (X direction), is added to XL	Oversize in tool length 2?
RS	Cutting 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 362

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

TYPE	Supported TO with possible limitations	Non-supported TO	
ROUGH, FINISH	<ul style="list-style-type: none"> ■ 1 ■ 7 ■ 2, only XL ■ 3, only XL ■ 5, only XL ■ 6, only XL ■ 8, only ZL 	<ul style="list-style-type: none"> ■ 4 ■ 9 	
BUTTON	<ul style="list-style-type: none"> ■ 1 ■ 7 ■ 2, only XL ■ 3, only XL ■ 5, only XL ■ 6, only XL ■ 8, only ZL 	<ul style="list-style-type: none"> ■ 4 ■ 9 	
RECESS, RECTURN	<ul style="list-style-type: none"> ■ 1 ■ 7 ■ 8 ■ 2 ■ 3, only XL ■ 5, only XL 	<ul style="list-style-type: none"> ■ 4 ■ 6 ■ 9 	

TYPE	Supported TO with possible limitations	Non-supported TO
THREAD	<ul style="list-style-type: none"> ■ 1 ■ 7 ■ 8 ■ 2 ■ 3, only XL ■ 5, only XL 	<ul style="list-style-type: none"> ■ 4 ■ 6 ■ 9



Notes

NOTICE

Danger of collision!

If you set **stopOnCheck** (no. 122717) to **FALSE**, the control does not evaluate the result parameter **Q199** and the NC program is not stopped if the breakage tolerance is exceeded. There is a danger of collision!

- ▶ Set **stopOnCheck** (no. 122717) to **TRUE**
- ▶ You must then take steps to ensure that the NC program stops if the breakage tolerance is exceeded

NOTICE

Danger of collision!

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

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

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

Note regarding machine parameters

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

10.7.1 Cycle parameters

Help graphic	Parameter
	<p>Q340 Tool measurement mode (0-2)?</p> <p>Use of the measured values:</p> <p>0: The measured values are entered in ZL and XL. If values are already entered in the tool table, they will be overwritten. DZL and DXL will be reset to 0. TL will not be changed</p> <p>1: The measured values ZL and XL are compared with the values from the tool table. These values will not be changed. The control then calculates the deviations of ZL and XL, and enters these in DZL and DXL. If the delta values are larger than the permissible wear or breakage tolerance, the control locks the tool (TL = Tool Locked). In addition, the deviation is also entered in the Q parameters Q115 and Q116</p> <p>2: The measured values ZL and XL as well as DZL and DXL are compared with the values from the tool table, but are not changed. If the values are larger than the permissible wear or breakage tolerance, the control locks the tool (TL = Tool Locked).</p> <p>Input: 0, 1, 2</p>
	<p>Q260 Clearance height?</p> <p>Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from safetyDistStylus).</p> <p>Input: -99999.9999...+99999.9999</p>

Example

11 TOOL CALL 12 Z	
12 TCH PROBE 485 MEASURE LATHE TOOL ~	
Q340=+1	;CHECK ~
Q260=+100	;CLEARANCE HEIGHT

11

Special Cycles

11.1 Fundamentals

11.1.1 Overview

The control provides the following cycles for special purposes:

Cycle		Sequence	Further information
9	DWELL TIME ■ Delay execution by the programmed dwell time	DEF-active	Further information: User's Manual for Machining Cycles
12	PGM CALL ■ Call any NC program	DEF-active	Further information: User's Manual for Machining Cycles
13	ORIENTATION ■ Rotate spindle to a specific angle	DEF-active	"Cycle 13 ORIENTATION "
32	TOLERANCE ■ Program the permissible contour deviation for jerk-free machining operations	DEF-active	Further information: User's Manual for Machining Cycles
291	COUPLG.TURNG.INTERP. (option 96) ■ Coupling of the tool spindle with the positions of the linear axes ■ Or, rescind the spindle coupling	CALL-active	Further information: User's Manual for Machining Cycles
292	CONTOUR.TURNG.INTRP. (option 96) ■ 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 working plane	CALL-active	Further information: User's Manual for Machining Cycles
225	ENGRAVING ■ Engrave texts on a plane surface ■ Arranged in a straight line or along a circular arc	CALL-active	Further information: User's Manual for Machining Cycles
232	FACE MILLING ■ Face mill a level surface in multiple infeeds ■ Selection of the milling plan	CALL-active	Further information: User's Manual for Machining Cycles
285	DEFINE GEAR (option 157) ■ Define the geometry of the gear wheel	DEF-active	Further information: User's Manual for Machining Cycles
286	GEAR HOBGING (option 157) ■ Definition of the tool data ■ Selection of the machining strategy and side ■ Possibility of using the entire cutting edge	CALL-active	Further information: User's Manual for Machining Cycles
287	GEAR SKIVING (option 157) ■ Definition of the tool data ■ Selection of the machining side ■ Definition of the first and last infeed ■ Definition of the number of cuts	CALL-active	Further information: User's Manual for Machining Cycles

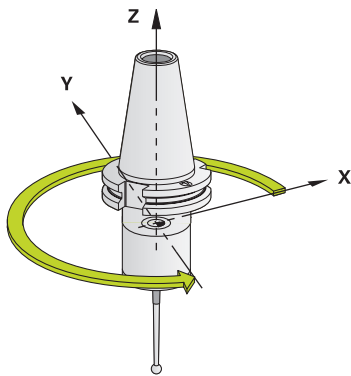
Cycle	Sequence	Further information
238 MEASURE MACHINE STATUS (option 155) <ul style="list-style-type: none"> Determine the current machine status or test the measuring sequence 	DEF-active	Further information: User's Manual for Machining Cycles
239 ASCERTAIN THE LOAD (option 143) <ul style="list-style-type: none"> Selection for a weighing run Reset the load-dependent feedforward and controller parameters 	DEF-active	Further information: User's Manual for Machining Cycles
18 THREAD CUTTING <ul style="list-style-type: none"> With controlled spindle Spindle stops at the bottom of the hole 	CALL-active	Further information: User's Manual for Machining Cycles

11.2 Cycle 13 ORIENTATION

Application



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



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

Oriented spindle stops are required for purposes such as:

- Tool changing systems with a defined tool change position
- Orientation of the 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.

Notes

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

11.2.1 Cycle parameters

Help graphic	Parameter
	Orientation angle Enter the angle relative to the angle reference axis of the working plane. Input: 0...360

Example

```
11 CYCL DEF 13.0 ORIENTATION
```

```
12 CYCL DEF 13.1 ANGLE180
```


Index

A

Additional documentation.....	21
Automatic preset setting	
Bolt hole circle.....	170
Center of 4 holes.....	181
Circle probing.....	125
Circular pocket (hole).....	147
Circular stud.....	153
Fundamentals of 4xx.....	134
Inside corner.....	164
Outside corner.....	159
Rectangular pocket.....	136
Rectangular stud.....	141
Ridge center.....	194
Single axis.....	186
Single position probing.....	121
Slot center.....	189
Sphere probing.....	129
Touch probe axis.....	177
Automatic workpiece inspection	
Fundamentals.....	204

B

Basic rotation.....	95
Setting directly.....	117
Using two holes.....	97
Using two studs.....	101
Via rotary axis.....	106

C

Calibration cycles.....	280
Calibrating TS.....	290
Calibrating TS in a ring.....	284
Calibrating TS length.....	282
Calibrating TS on a stud.....	287
Checking for workpiece misalignment	
Measuring angles.....	212
Measuring a plane.....	253
Measuring bolt hole circles....	248
Measuring circles.....	221
Measuring coordinates.....	243
Measuring holes.....	215
Measuring rectangular pockets....	227
Measuring rectangular studs	231
Measuring the ridge width....	240
Measuring the slot width.....	236
Polar preset.....	210
Reference plane.....	209
Classification of results.....	207
Comparison of controls.....	38
Contact.....	23

D

Determine inclined workpiece	
------------------------------	--

position

Touch probe cycles 4xx	
fundamentals.....	94
Determining workpiece misalignment	
Basic rotation.....	95
Basic rotation using two holes	97
Basic rotation using two studs.....	101
Basic rotation via rotary axis.	106
Fundamentals of touch probe cycles 14xx.....	57
Inclined edge probing.....	88
Probing in plane.....	67
Probing on edge.....	73
Probing two circles.....	80
Rotation via C axis.....	112
Setting basic rotation.....	117
Differences in controls.....	38

E

Extrusion probing.....	276
------------------------	-----

F

Fast probing.....	274
FCL.....	37
Feature Content Level.....	37

K

Kinematic measurement	
Accuracy.....	311
Backlash.....	312
Kinematics measurement	
Fundamentals.....	298
Hirth coupling.....	308
Kinematics grid.....	331
Preset compensation.....	320
Saving kinematics.....	302
KinematicsOpt.....	298

L

Licensing terms.....	37
----------------------	----

M

Measuring	
Angle.....	212
Bolt hole circle.....	248
Circle outside.....	221
Coordinate.....	243
Hole.....	215
Inside width.....	236
Measuring rectangles on the inside.....	227
Plane.....	253
Rectangle outside.....	231
Ridge width.....	240
Measuring circles on the inside.	215
Measuring circles on the outside.....	221

Measuring in 3-D.....	265
Measuring rectangular pockets.	227
Measuring rectangular studs.....	231
Measuring the inside width.....	236
Measuring the ridge width.....	240
Measuring the slot width.....	236
Measuring with Cycle 3.....	263

N

Notes, types of.....	22
----------------------	----

P

Place of operation.....	27
Positioning logic.....	50
Probing in 3-D.....	268
Proper and intended operation....	26

R

Recording measurement results....	205
-----------------------------------	-----

S

Safety precaution.....	28
Content.....	22
Software number.....	30
Software option.....	31
Spindle orientation.....	367
Split screen layout of User's Manual.....	21

T

Target group.....	20
Tolerance monitoring.....	207
Tool compensation.....	208
Tool measurement	
Fundamentals.....	340
IR TT calibration.....	356
Machine parameters.....	341
Measuring tool length and radius.....	353
Measuring turning tools.....	360
Tool length.....	346
Tool radius.....	349
TT calibration.....	344
Tool table.....	343
Touch probe cycles 14xx	
Fundamentals.....	57
Inclined edge probing.....	88
Probing in plane.....	67
Probing on edge.....	73
Probing two circles.....	80

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.

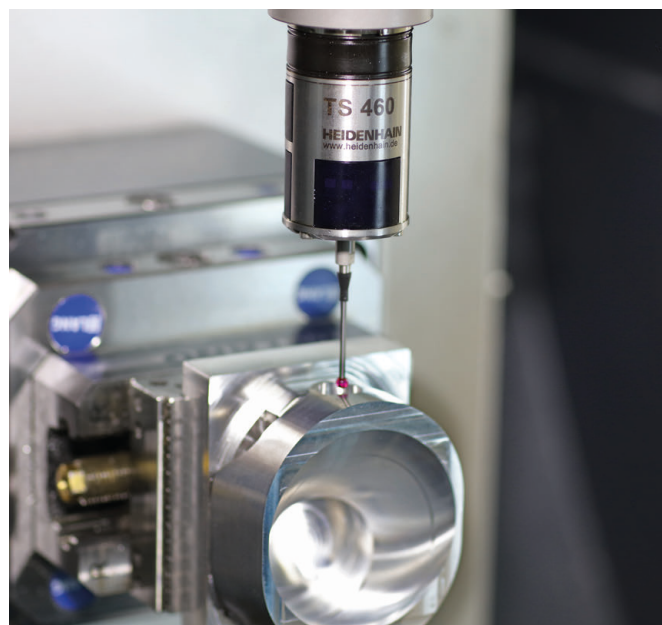
Workpiece touch probes

TS 150, TS 260, Signal transmission by cable
TS 750

TS 460, TS 760 Radio or infrared transmission

TS 642, 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

