



HEIDENHAIN



TNC7 basic

User's Manual
Measuring Cycles for
Workpieces and Tools

NC Software
81762x-20

English (en)
10/2025

Table of contents

1	New and modified functions.....	17
2	About the User's Manual.....	21
3	About the product.....	35
4	First steps.....	53
5	NC and programming fundamentals.....	73
6	Programming with variables.....	93
7	Touch probes.....	99
8	Touch-probe cycles for workpieces (#17 / #1-05-1).....	123
9	Touch-probe cycles for tools (#17 / #1-05-1).....	395
10	Touch-probe cycles for measuring the kinematics.....	419

1	New and modified functions.....	17
1.1	New functions.....	19
1.1.1	Process Tracking Interface (#3-04-1*).....	19
1.1.2	Dynamic Collision Monitoring DCM (#140 / #5-03-2).....	19
1.2	Modified or extended functions.....	20
1.2.1	Touch-probe cycles for workpieces (#17 / #1-05-1).....	20

2	About the User's Manual.....	21
2.1	Target group: Users.....	22
2.2	Available user documentation.....	23
2.3	Types of notes used.....	25
2.4	Notes on using NC programs.....	27
2.5	User's Manual as integrated product aid: TNCguide.....	28
2.5.1	Searching in TNCguide.....	31
2.5.2	Copying NC examples to clipboard.....	32
2.6	Contacting the editorial staff.....	33

3	About the product.....	35
3.1	The TNC7 basic.....	36
3.1.1	Proper and intended use.....	36
3.1.2	Data Architecture and Access Options.....	37
3.1.3	Intended place of operation.....	38
3.2	Safety precautions.....	39
3.3	Software.....	41
3.3.1	Software options.....	42
3.3.2	Information on licensing and use.....	48
3.4	Areas of the control's user interface.....	49
3.5	Overview of the operating modes.....	51

4	First steps.....	53
4.1	Programming and simulating a workpiece.....	54
4.1.1	Programming the workpiece.....	54
4.1.2	Simulating a workpiece.....	70

5	NC and programming fundamentals.....	73
5.1	Working with cycles.....	74
5.1.1	General information on cycles.....	74
5.1.2	General information about touch probe cycles.....	83
5.1.3	Machine-specific cycles.....	89
5.1.4	Available cycle groups.....	90

6	Programming with variables.....	93
6.1	Program defaults for cycles.....	94
6.1.1	Overview.....	94
6.1.2	Entering GLOBAL DEF definitions.....	94
6.1.3	Using GLOBAL DEF information.....	95
6.1.4	Global data valid everywhere.....	96
6.1.5	Global data for probing functions.....	97

7	Touch probes.....	99
7.1	Calibrating a workpiece touch probe (#17 / #1-05-1).....	100
7.1.1	Overview.....	100
7.1.2	Fundamentals.....	100
7.1.3	Cycle 460 CALIBRATION OF TS ON A SPHERE (#17 / #1-05-1).....	102
7.1.4	Cycle 461 TS CALIBRATION OF TOOL LENGTH (#17 / #1-05-1).....	109
7.1.5	Cycle 462 CALIBRATION OF A TS IN A RING (#17 / #1-05-1).....	111
7.1.6	Cycle 463 TS CALIBRATION ON STUD (#17 / #1-05-1).....	114
7.2	Calibrating a tool touch probe (#17 / #1-05-1).....	117
7.2.1	Overview.....	117
7.2.2	Fundamentals.....	117
7.2.3	Cycle 480 CALIBRATE TT (#17 / #1-05-1).....	118
7.2.4	Cycle 484 CALIBRATE IR TT (#17 / #1-05-1).....	120

8	Touch-probe cycles for workpieces (#17 / #1-05-1)	123
8.1	Overview	124
8.2	Conditional stops in touch probe cycles	130
8.3	Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)	131
8.3.1	Application	131
8.3.2	Evaluation	131
8.3.3	Logging	132
8.3.4	Notes	132
8.3.5	Semi-automatic mode	133
8.3.6	Evaluation of tolerances	139
8.3.7	Transferring the actual position	142
8.4	Determining workpiece misalignment (#17 / #1-05-1)	143
8.4.1	Fundamentals of touch probe cycles 400 to 405	143
8.4.2	Cycle 400 BASIC ROTATION (#17 / #1-05-1)	144
8.4.3	Cycle 401 ROT OF 2 HOLES (#17 / #1-05-1)	148
8.4.4	Cycle 402 ROT OF 2 STUDS (#17 / #1-05-1)	153
8.4.5	Cycle 403 ROT IN ROTARY AXIS (#17 / #1-05-1)	158
8.4.6	Cycle 404 SET BASIC ROTATION (#17 / #1-05-1)	163
8.4.7	Cycle 405 ROT IN C AXIS (#17 / #1-05-1)	165
8.4.8	Cycle 1410 PROBING ON EDGE (#17 / #1-05-1)	170
8.4.9	Cycle 1411 PROBING TWO CIRCLES (#17 / #1-05-1)	177
8.4.10	Cycle 1412 INCLINED EDGE PROBING (#17 / #1-05-1)	185
8.4.11	Cycle 1416 INTERSECTION PROBING (#17 / #1-05-1)	192
8.4.12	Cycle 1420 PROBING IN PLANE (#17 / #1-05-1)	200
8.4.13	Example: Determining a basic rotation from two holes	207
8.4.14	Example: Determining a basic rotation from a plane and two holes	208
8.4.15	Example: Aligning the rotary table from two holes	210

8.5	Determining the preset (#17 / #1-05-1).....	212
8.5.1	Fundamentals of touch probe cycles 408 to 419 for preset setting.....	212
8.5.2	Cycle 408 SLOT CENTER PRESET (#17 / #1-05-1).....	214
8.5.3	Cycle 409 RIDGE CENTER PRESET (#17 / #1-05-1).....	219
8.5.4	Cycle 410 PRESET INSIDE RECTAN (#17 / #1-05-1).....	224
8.5.5	Cycle 411 PRESET OUTS. RECTAN (#17 / #1-05-1).....	229
8.5.6	Cycle 412 PRESET INSIDE CIRCLE (#17 / #1-05-1).....	234
8.5.7	Cycle 413 PRESET OUTS. CIRCLE (#17 / #1-05-1).....	240
8.5.8	Cycle 414 PRESET OUTS. CORNER (#17 / #1-05-1).....	246
8.5.9	Cycle 415 PRESET INSIDE CORNER (#17 / #1-05-1).....	253
8.5.10	Cycle 416 PRESET CIRCLE CENTER (#17 / #1-05-1).....	259
8.5.11	Cycle 417 PRESET IN TS AXIS (#17 / #1-05-1).....	265
8.5.12	Cycle 418 PRESET FROM 4 HOLES (#17 / #1-05-1).....	269
8.5.13	Cycle 419 PRESET IN ONE AXIS (#17 / #1-05-1).....	274
8.5.14	Cycle 1400 POSITION PROBING (#17 / #1-05-1).....	278
8.5.15	Cycle 1401 CIRCLE PROBING (#17 / #1-05-1).....	282
8.5.16	Cycle 1402 SPHERE PROBING (#17 / #1-05-1).....	287
8.5.17	Cycle 1403 RECTANGLE PROBING (#17 / #1-05-1).....	292
8.5.18	Cycle 1404 PROBE SLOT/RIDGE (#17 / #1-05-1).....	298
8.5.19	Cycle 1430 PROBE POSITION OF UNDERCUT (#17 / #1-05-1).....	304
8.5.20	Cycle 1434 PROBE SLOT/RIDGE UNDERCUT (#17 / #1-05-1).....	310
8.5.21	Example: Presetting at center of a circular segment and on top surface of workpiece.....	316
8.5.22	Example: Presetting on top surface of workpiece and at center of a bolt hole circle.....	317
8.6	Checking the workpiece (#17 / #1-05-1).....	319
8.6.1	Fundamentals of touch probe cycles 0, 1, and 420 to 431.....	319
8.6.2	Cycle 0 REF. PLANE (#17 / #1-05-1).....	323
8.6.3	Cycle 1 POLAR PRESET (#17 / #1-05-1).....	325
8.6.4	Cycle 420 MEASURE ANGLE (#17 / #1-05-1).....	327
8.6.5	Cycle 421 MEASURE HOLE (#17 / #1-05-1).....	330
8.6.6	Cycle 422 MEAS. CIRCLE OUTSIDE (#17 / #1-05-1).....	335
8.6.7	Cycle 423 MEAS. RECTAN. INSIDE (#17 / #1-05-1).....	340
8.6.8	Cycle 424 MEAS. RECTAN. OUTS. (#17 / #1-05-1).....	345
8.6.9	Cycle 425 MEASURE INSIDE WIDTH (#17 / #1-05-1).....	350
8.6.10	Cycle 426 MEASURE RIDGE WIDTH (#17 / #1-05-1).....	354
8.6.11	Cycle 427 MEASURE COORDINATE (#17 / #1-05-1).....	358
8.6.12	Cycle 430 MEAS. BOLT HOLE CIRC (#17 / #1-05-1).....	363
8.6.13	Cycle 431 MEASURE PLANE (#17 / #1-05-1).....	367
8.6.14	Example: Measuring and reworking a rectangular stud.....	371
8.6.15	Example: Probing a rectangular pocket and recording the results.....	373
8.7	Probing a position in the plane or in space (#17 / #1-05-1).....	374
8.7.1	Cycle 3 MEASURING (#17 / #1-05-1).....	374
8.7.2	Cycle 4 MEASURING IN 3-D (#17 / #1-05-1).....	377
8.7.3	Cycle 444 PROBING IN 3-D (#17 / #1-05-1).....	379

8.8	Influencing cycle runs (#17 / #1-05-1).....	385
8.8.1	Cycle 441 FAST PROBING (#17 / #1-05-1).....	385
8.8.2	Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1).....	388

9	Touch-probe cycles for tools (#17 / #1-05-1)	395
9.1	Overview	396
9.2	Conditional stops in touch probe cycles	397
9.3	Fundamentals	398
9.3.1	Application	398
9.3.2	Measuring a tool of length 0	398
9.3.3	Setting machine parameters	399
9.3.4	Entries in the tool table for milling tools	401
9.4	Measurement of milling cutters (#17 / #1-05-1)	403
9.4.1	Cycle 481 CAL. TOOL LENGTH (#17 / #1-05-1)	403
9.4.2	Cycle 482 CAL. TOOL RADIUS (#17 / #1-05-1)	406
9.4.3	Cycle 483 MEASURE TOOL (#17 / #1-05-1)	410
9.5	Measuring turning tools	414
9.5.1	Cycle 485 MEASURE LATHE TOOL	414

10 Touch-probe cycles for measuring the kinematics.....	419
10.1 Overview.....	420
10.2 Conditional stops in touch probe cycles.....	421
10.3 Fundamentals of kinematics measurement (#48 / #2-01-1).....	422
10.3.1 Fundamentals.....	422
10.3.2 Requirements.....	423
10.3.3 Notes.....	424
10.4 Storing, measuring and optimizing kinematics (#48 / #2-01-1).....	425
10.4.1 Cycle 450 SAVE KINEMATICS (#48 / #2-01-1).....	425
10.4.2 Cycle 451 MEASURE KINEMATICS (#48 / #2-01-1).....	429
10.4.3 Cycle 452 PRESET COMPENSATION (#48 / #2-01-1).....	445
10.4.4 Cycle 453 KINEMATICS GRID (#48 / #2-01-1).....	457

1

**New and modified
functions**

Available documentation



TNC7 basic complete edition

The split editions of the User's Manual contain only the new and modified functions that are relevant to the corresponding User's Manual. The **complete edition** contains all new and modified functions of this software version that are relevant to the user.

ID: 1411730-xx

You can download this documentation free of charge from the HEIDENHAIN website.

https://content.heidenhain.de/doku/tnc_guide/html/en/



Overview of new and modified software functions

The additional documentation **Overview of New and Modified Software Functions** contains all new and modified functions of the current and previous software versions that are relevant to the user.

ID: 1443541-xx

You can download this documentation free of charge from the HEIDENHAIN website.

TNCguide

1.1 New functions

1.1.1 Process Tracking Interface (#3-04-1*)

Topic	Description
Software option PTI 8 channels (#3-04-1*)	This software option provides an interface for recording control and process signals at a scanning interval as short as 3 ms. With PTI, you can record live information about axes and spindles. The signals can be used, among other things, to create a digital workpiece twin in order to meet documentation requirements in the aerospace industry. This software option is available only for controls with SIK2. You can order this software option up to two times, each time selecting up to eight channels for recording.

1.1.2 Dynamic Collision Monitoring DCM (#140 / #5-03-2)

Topic	Description
The Collision Monitoring software option	The software option Collision Monitoring (#40 / #5-03-1) is no longer available. To be able to use the Dynamic Collision Monitoring (DCM) functions in the future, you need the software option Collision Monitoring v2 (#140 / #5-03-2) If the software option Collision Monitoring v2 (#140 / #5-03-2) is not available after a software update, the control will display a warning.

1.2 Modified or extended functions

1.2.1 Touch-probe cycles for workpieces (#17 / #1-05-1)

Topic	Description
Q1146 EXTRUSION LENGTH	In Cycle 1493 EXTRUSION PROBING (ISO: G1493), the input value of this parameter was expanded from ± 99 to ± 9999.99. Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)", Page 388
The Calibration mandrel (CAL-PIN) tool type	The following cycles support the Calibration mandrel (CAL-PIN) tool type: ■ Cycle 480 CALIBRATE TT (ISO: G480) (#17 / #1-05-1) ■ Cycle 484 CALIBRATE IR TT (ISO: G484) (#17 / #1-05-1) If you use the following cycles with the Calibration mandrel (CAL-PIN) tool type, you can check measured tool values, but can't correct them: ■ Cycle 481 CAL. TOOL LENGTH (ISO: G481) (#17 / #1-05-1) ■ Cycle 482 CAL. TOOL RADIUS (ISO: G482) (#17 / #1-05-1) ■ Cycle 483 MEASURE TOOL (ISO: G484) (#17 / #1-05-1)
The TCHPRAUTO.html log file	The maximum file size of the automatic log file TCHPRAUTO.html was increased to 512 kB. This change applies to the following cycles: ■ Touch-probe cycles 14xx (#17 / #1-05-1) ■ Cycle 444 PROBING IN 3-D (ISO: G444) (#17 / #1-05-1) ■ Calibrating touch probe cycles for the workpiece touch probe ■ Touch probe cycles for measuring the kinematics
Machine parameters	The optional machine parameter trackAsync (no. 122503) for machine manufacturers was removed from the control.

2

**About the
User's Manual**

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



HEIDENHAIN offers separate information products for other target groups:

- Leaflets and overview of the product portfolio 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.

2.2 Available user documentation

User's Manual

HEIDENHAIN refers to this information product as a 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 PDF file containing a **complete edition** of the User's Manual with all contents
ID: 1411730-xx
https://content.heidenhain.de/doku/tnc_guide/html/en/
- As an HTML file with all contents for use as the integrated product aid **TNCguide**: directly on the control
https://content.heidenhain.de/doku/tnc_guide/html/en/
- As a printed version, sub-divided into the modules below:
 - The **Setup and Program Run** User's Manual contains all information needed for setting up the machine and for executing NC programs.
ID: 1410286-xx
 - The **Programming and Testing** User's Manual contains all information needed for creating and testing NC programs. Touch probe cycles and machining cycles are not included.
ID: 1409856-xx
 - The **Machining Cycles** User's Manual contains all machining cycles functions.
ID: 1410289-xx
 - The **Measuring Cycles for Workpieces and Tools** User's Manual contains all functions related to the touch-probe cycles.
ID: 1410290-xx

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

Further information: "Proper and intended use", Page 36



Complete edition and TNCguide for the TNC7 basic

Binding of the printed version of the User's Manual is only possible if a certain number of pages is not exceeded.

Die **complete edition** and **TNCguide** may include additional sample programs for NC functions that are not contained in the printed version.

You can download this documentation free of charge from the HEIDENHAIN website.

Further information products for users

The following information products are available:

- The **Overview of New and Modified Software Functions** informs you about the innovations of specific software versions.

https://content.heidenhain.de/doku/tnc_guide/html/en/

- **Overview of the machine parameters, error numbers and system data**, providing the following functions:
 - Machine parameters of the **MPs for setters** application
 - Preassigned error numbers of the **FN 14: ERROR** NC function (ISO: **D14**)
 - System data readable with the **FN 18: SYSREAD** (ISO: **D18**) and **SYSSTR** NC functions

https://content.heidenhain.de/doku/tnc_guide/html/en/

- **HEIDENHAIN brochures** inform you about products and services from HEIDENHAIN (e.g., software options of the control).

HEIDENHAIN brochures

- The **NC Solutions** database offers solutions for frequently occurring tasks.
HEIDENHAIN NC solutions

2.3 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
Caution indicates danger to material or data. If you do not follow the avoidance instructions, the hazard could result in things other than personal injury, such as property damage .

Sequence of information in precautionary statements

All precautionary statements comprise the following four sections:

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

Informational notes

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

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



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



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



The book symbol indicates a **cross reference**.
A cross reference leads to external documentation, for example the documentation of your machine manufacturer or other supplier.

2.4 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 program can be used with the available software options, the active machine kinematics and the current machine configuration.

2.5 User's Manual as integrated product aid: TNCguide

Application

The integrated product aid **TNCguide** offers the full content of all User's Manuals.

Further information: "Available user documentation", Page 23

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

Further information: "Proper and intended use", Page 36

Related topics

- The **Help** workspace

Further information: Programming and Testing User's Manual

Requirement

In the factory default setting, the control offers the integrated product aid **TNCguide** in German and English language versions.

If the control cannot find a **TNCguide** language version matching the selected dialog language, it opens **TNCguide** in English.

If the control cannot find a **TNCguide** language version, it opens an information page with instructions. With the link available there and the steps provided, you can supplement the files missing in the control.



You can also open the information page manually by selecting the **index.html** file (for example, at **TNC:\tncguide\en\readme**). The path depends on the desired language version (e.g., **en** for English).

With the steps provided you can also update the **TNCguide** version. Updating may be required (e.g., after a software update).

Description of function

The integrated product aid **TNCguide** can be selected within the **Help** application or in the **Help** workspace.

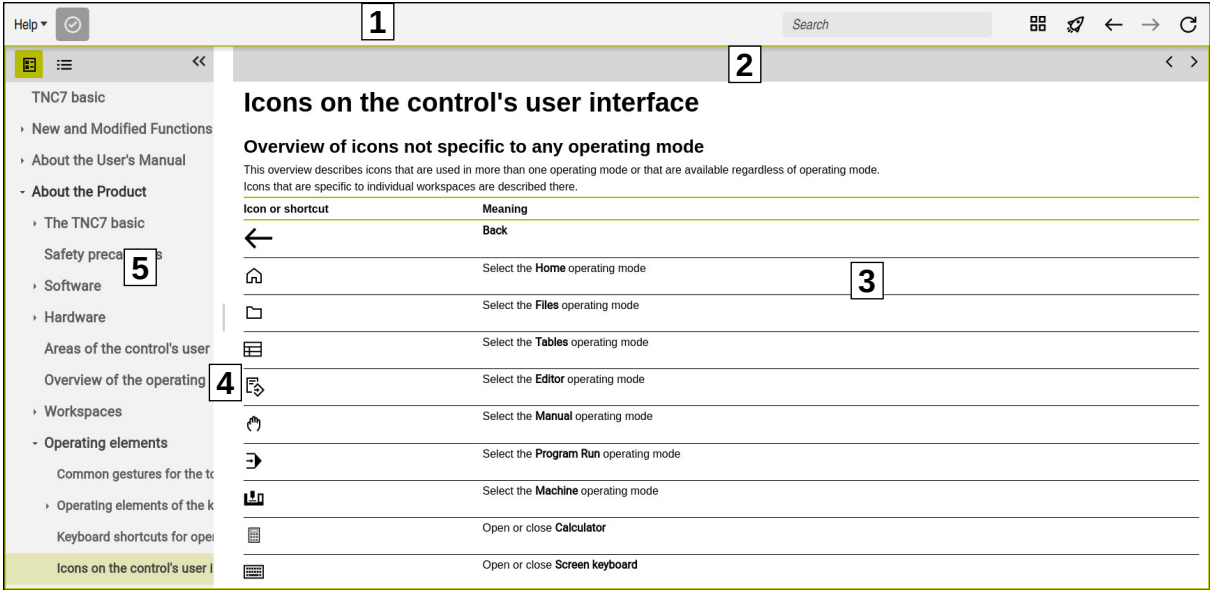
Further information: "The Help application", Page 29

Further information: Programming and Testing User's Manual

Operation of **TNCguide** is identical in both cases.

Further information: "Icons", Page 30

The Help application



Open **TNCguide** in the **Help** workspace

TNCguide includes the following areas:

- 1 Title bar of the **Help** workspace
Further information: "The Help workspace", Page 30
- 2 Title bar of the integrated product aid **TNCguide**
Further information: "TNCguide ", Page 30
- 3 Content column of the **TNCguide**
- 4 Separator between the columns of the **TNCguide**
Adjust the column width by means of the separator.
- 5 Navigation column of the **TNCguide**

Icons

The Help workspace

The **Help** workspace within the **Help** application includes the following icons:

Icon	Meaning
	Open or close the Search results column Further information: "Searching in TNCguide", Page 31
	Open Home page The start page displays all available documentation. Select the desired documentation using navigation tiles (e.g., TNCguide). If only one piece of documentation is available, the control opens the content directly. When a documentation is open, you can use the search function.
	Open Tutorials
	Navigate Navigate between the contents opened recently
	Refresh

TNCguide

The integrated **TNCguide** product aid includes the following icons:

Icon	Meaning
	Open Structure The structure consists of the content headings. The structure serves for main navigation within the documentation.
	Open Index The index consists of important keywords. The index serves as an alternative navigation within the documentation.
	Navigate Display previous or next page within the documentation
	Open or close Display or hide the navigation
	Copy Copy NC examples to the clipboard Further information: "Copying NC examples to clipboard", Page 32

Context-sensitive help

You can open **TNCguide** for the current context. Context-sensitive help means that the relevant information is displayed directly (e.g., for the selected item or the current NC function).

To call context-sensitive help, the following elements are available:

Icon or key	Meaning
	Help icon If you select the icon and then one of the items in the user interface, the control will open the associated information in TNCguide .
	HELP key If you press the HELP key while editing an NC block, the control will display the associated information in TNCguide .

If you call TNCguide in a certain context, the control opens the contents in a pop-up window. If you select the **Show more** button, the control will open **TNCguide** in the **Help** application.

Further information: "The Help application", Page 29

If the **Help** workspace is already open, the control displays **TNCguide** there and will not open a pop-up window.

Further information: Programming and Testing User's Manual

2.5.1 Searching in TNCguide

Using the search function, you can search for the entered search terms within the open documentation.

Use the search function as follows:

- ▶ Enter a string in **Search**



The search starts automatically after you enter a character.
If you wish to delete the entry, use the X symbol within the entry field.

- > The control opens the column containing the search results.
- > The control marks references also within open content pages.
- ▶ Select the reference
- > The control opens the selected content.
- > The control continues displaying the results of the last search.
- ▶ Select an alternative reference if necessary
- ▶ Enter a new character string if required

2.5.2 Copying NC examples to clipboard

Use the copy function to copy NC examples from the documentation to the NC editor.

To use the copy function:

- ▶ Navigate to the desired NC example
- ▶ Expand **Notes on using NC programs**
- ▶ Read and follow **Notes on using NC programs**

Further information: "Notes on using NC programs", Page 27



- ▶ Copy NC example to clipboard



- > The button switches colors while copying.
 - > The clipboard contains the entire content of the copied NC example.
 - ▶ Insert the NC example into the NC program
 - ▶ Adapt the inserted content according to the **Notes on using NC programs**
 - ▶ Use the Simulation mode to test the NC program
- Further information:** Programming and Testing User's Manual

2.6 Contacting the editorial staff

Would you like any changes, or have you found any errors?

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

tnc-userdoc@heidenhain.com

3

About the product

3.1 The TNC7 basic

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

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

Operation is made easier, for example, by 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) stands for the capability of entering 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.
basic	The addition basic indicates that the control provides all required basic functions for universal milling or drilling.

3.1.1 Proper and intended use

The information about proper and intended use supports you in safely handling 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 customer feedback into consideration. This results, for example, in function adaptations of the controls and safety precautions in the information products.



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

Further information: "Contacting the editorial staff", Page 33

3.1.2 Data Architecture and Access Options

Directories

While in use, the control will store various types of data on its internal data carrier.

Partition or directory	Data
root	Operating system
/mnt/tnc/	Process, application, and user data Examples: NC programs, tables or 3D models
/mnt/plc/	Machine manufacturer data Examples: PLC program, machine configuration or additional software
/mnt/sys/	NC software

The type, format, and amount of data stored depend on various factors. For example, the software options that are available and actually used as well as the functions used, including the current settings, have a great influence here.

Data acquisition

While in use, the control will generate different pieces of data. Depending on the functions in use, for example, data acquisition is performed continuously and in real time.

Data acquisition examples:

- Backup
You can manually create a backup copy of the TNC partition in order to avoid potential loss of data.
- Service file
In the event of a fault, you can manually create a service file that supports troubleshooting by a service technician.
- User administration
If user administration is active, the control assigns defined roles and permissions to the logged-on users. When a user logs on, the control will acquire personal or anonymous data in connection with its use.
- Tool-usage file
Using tool-usage files, the control can, for example, check whether the tool's remaining life is sufficient for the current NC program.
- Log functions or log files (e.g., from touch-probe cycles)
Using various log files, you can, for example, check and compare the results of touch-probe cycles.
The control needs other function-dependent log files to, for example, detect process variations.



For more details on data acquisition, please refer to the sections describing the individual functions and the **Settings** application.

Data transfer and data access

Machine manufacturers, machine operators, or users can transfer data to external data carriers or drives, for example, and set up and configure a data transfer functionality. The user has control over and responsibility for the stored data (e.g., with respect to safety and the duration of storage).

Examples of data transfer:

- **HEIDENHAIN DNC** (#18 / #3-03-1)
This software option enables external Windows applications to access data of the control via the TCP/IP protocol.
- **OPC UA NC Server** (#56-61 / #3-02-1*)
These software options include OPC UA, a standardized interface that allows external access to the data and functions of your control.
- **USB devices**
USB devices can be used to transfer data or make external backups.

 For more details on data transfer, please refer to the sections describing the individual functions and the **Settings** application.

Data access via file management:

- **Drives, folders, and files**
You can access external data or make external backups by using connected drives or USB devices. For example, you can manage, create, edit, or delete folders and files.
- **Functions and settings**
In the file manager you can search for certain contents or display hidden or dependent files.

 For further details on file access, please refer to the section on the **Files** operating mode.
Please observe the safety instructions and information given by the machine manufacturer in order to avoid loss of data or jeopardize the functionality of your system.

3.1.3 Intended place of operation

In accordance with the DIN EN 50370-1 standard for 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.

3.2 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 length from the tool table for compensating for the tool length. Incorrect tool lengths will result in an incorrect tool length compensation. The control does not perform tool length compensation or a collision check for tools with a length of **0** and after a **TOOL CALL 0**. There is a risk 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 **Program Run** Single Block mode

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

If you do not properly remove a connected USB device during a data transfer, then data may be damaged or deleted!

- ▶ Use the USB port only for transferring or backing up data do not use it for editing and executing NC programs
- ▶ Use the **Eject** soft key to remove a USB device when data 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 regardless of the control's status!

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

3.3 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 42

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
817620-20	TNC7 basic
817625-20	TNC7 basic 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. The machine manufacturer can also change the colors in the control's GUI.

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

If later customization of the machine configuration by the machine manufacturer is intended, the machine operator might incur additional costs.

3.3.1 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

The TNC7 basic features various software options that the machine manufacturer may enable separately, even at a later point in time. The following overview includes only those software options that are relevant for you.

The software options are saved on the **SIK** (System Identification Key) plug-in board. The TNC7 basic can be equipped with a **SIK** or **SIK2** plug-in board. Depending on which one is used, the numbers of the software options differ.



The option numbers in parentheses given in the User's Manual show you that a function is not included in the standard range of available functions.

The parentheses enclose the **SIK** and **SIK2** option numbers, separated by a slash, for example: (#18 / #3-03-1).

The Technical Manual contains information about additional software options that are relevant to the machine manufacturer.

SIK2 definitions

SIK2 option numbers are structured by <class>-<option>-<version>:

Class	The function is effective for the following areas: ■ 1: Programming, simulation, and process setup ■ 2: Part quality and productivity ■ 3: Interfaces ■ 4: Technology functions and quality assessment ■ 5: Process stability and monitoring ■ 6: Machine configuration ■ 7: Developer tools
Option	Sequential number within each class
Version	New versions of software options are released if, for example, its features have been changed.

You can order some software options with **SIK2** more than once in order to obtain multiple variants of the same function (e.g., if you need to enable multiple control loops for the axes). In the User's Manual, these software option numbers are identified by an asterisk (*).

The control indicates in the **SIK** menu item of the **Settings** application whether a software option has been enabled, and if so, how often. The control also shows whether is equipped with the **SIK** or **SIK2**.

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

Overview



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
Control Loop Qty. (#0-3 / #6-01-1*)	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, for example, for detachable and motor-driven tilting tables. If your control features a SIK2, you can order this software option multiple times and enable up to 8 control loops.
Adv. Function Set 1 (#8 / #1-01-1)	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: ■ Tilting the working plane (e.g., with PLANE SPATIAL) Further information: Programming and Testing User's Manual ■ Programming of contours on an unrolled cylinder surface (e.g., with 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: Programming and Testing User's Manual ■ 3-axis circular interpolation with a tilted working plane The advanced functions (set 1) reduce the setup effort and increase the workpiece accuracy.
Adv. Function Set 2 (#9 / #4-01-1)	Advanced functions (set 2) On machines with rotary axes, this software option enables simultaneous 4-axis machining of workpieces. The software option includes the following functions: ■ TCPM (tool center point management): Automatic tracking of linear axes during rotary axis positioning Further information: Programming and Testing User's Manual ■ Running of NC programs with vectors, including optional 3D tool compensation Further information: Programming and Testing User's Manual ■ Manual moving of axes in the active tool coordinate system T-CS

Software option	Definition and application
Touch Probe Functions (#17 / #1-05-1)	Touch-probe functions This software option is used to program and execute automatic probing processes. If you are using a HEIDENHAIN touch probe with EnDat interface, then the Touch Probe Functions software option (#17 / #1-05-1) is automatically enabled. The software option includes the following functions: ■ Automatic compensation of workpiece misalignment ■ Automatic setting of workpiece presets ■ Automatic measurement of workpieces ■ Automatic measurement of tools The touch-probe functions reduce the setup effort and increase accuracy when machining workpieces.
HEIDENHAIN DNC (#18 / #3-03-1)	HEIDENHAIN DNC This software option enables external Windows applications to access data of the control via the TCP/IP protocol. Potential fields of application include: ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data HEIDENHAIN DNC is required in conjunction with external Windows applications.
Adv. Function Set 3 (#21 / #4-02-1)	Advanced functions (set 3) This software option offers additional ease of use with two powerful miscellaneous functions. The software option includes the following miscellaneous functions: ■ M120 for machining small contour steps without error message and contour damage Further information: Programming and Testing User's Manual ■ M118 for superimposed handwheel positioning during program run Further information: Programming and Testing User's Manual The advanced functions (set 3) reduce the setup effort and increase flexibility during program run.
CAD Import (#42 / #1-03-1)	CAD Import This software option is used to select positions and contours from CAD files and to transfer them into an NC program. 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. Further information: User's Manual for Setup and Program Run
Adaptive Feed Contr. (#45 / #2-31-1)	Adaptive Feed Control (AFC) 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. With AFC you can shorten machining times without adapting the NC program, while at the same time preventing machine damage from overload. Further information: User's Manual for Setup and Program Run

Software option	Definition and application
KinematicsOpt (#48 / #2-01-1)	KinematicsOpt This software option uses automatic probing processes to check and optimize the active kinematics. 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 for temperature-induced deviations through repeated measurements and corrections. Further information: "Touch-probe cycles for measuring the kinematics", Page 419
OPC UA NC Server (#56-61 / #3-02-1*)	OPC UA NC Server These software options include OPC UA, a standardized interface for remote access to the control's data and functions. Potential fields of application include: ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data Each software option enables one client connection. If more than one parallel connection is required, you need to enable multiple of these software options. If your control features the SIK2, you can order this software option multiple times and enable up to ten connections. Further information: User's Manual for Setup and Program Run
4 Additional Axes (#77 / #6-01-1*)	Four additional control loops Further information: "Control Loop Qty. (#0-3 / #6-01-1*)", Page 43
Ext. Tool Management (#93 / #2-03-1)	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: ■ 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 Extended tool management enables you to detect the tool requirements in time and thus prevent interruptions during program run.
Remote Desk. Manager (#133 / #3-01-1)	Remote Desktop Manager This software option is used to display and operate externally linked computer units. With Remote Desktop Manager you don't need to move between different workstations as much, and so increase your efficiency. Further information: User's Manual for Setup and Program Run

Software option	Definition and application
Collision Monitoring v2 (#140 / #5-03-2)	Dynamic Collision Monitoring (DCM) 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. The software option includes the following functions: ■ Automatic interruption of program run whenever a collision is imminent ■ Warnings in case of manual axis movements ■ Collision monitoring in Test Run mode ■ Collision monitoring of fixtures ■ Defining a reduced minimum distance between fixture and tool With DCM you can prevent collisions and thus avoid additional costs resulting from material damage or machine downtime. Further information: User's Manual for Setup and Program Run
Cross Talk Comp. (#141 / #2-20-1)	Compensation of axis couplings (CTC) Using this software option, the machine manufacturer can, for example, compensate for acceleration-induced deviations at the tool and thus increase accuracy and dynamic performance.
Position Adapt. Contr. (#142 / #2-21-1)	Position Adaptive Control (PAC) Using this software option, the machine manufacturer can, for example, compensate for position-induced deviations at the tool and thus increase accuracy and dynamic performance.
Load Adapt. Contr. (#143 / #2-22-1)	Load Adaptive Control (LAC) Using this software option, the machine manufacturer can, for example, compensate for load-induced deviations at the tool and thus increase accuracy and dynamic performance.
Motion Adapt. Contr. (#144 / #2-23-1)	Motion Adaptive Control (MAC) Using this software option, the machine manufacturer can, for example, change speed-dependent machine settings and thus increase the dynamic performance.
Active Chatter Contr. (#145 / #2-30-1)	Active Chatter Control (ACC) With this software option the chatter tendency of a machine used for heavy machining can be reduced. 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%. Further information: User's Manual for Setup and Program Run
Machine Vibr. Contr. (#146 / #2-24-1)	Vibration damping for machines (MVC) Damping of machine oscillations for improving the workpiece surface quality through the following functions: ■ AVD Active Vibration Damping ■ FSC Frequency Shaping Control
CAD Model Optimizer (#152 / #1-04-1)	Optimization of CAD models 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. Further information: User's Manual for Setup and Program Run

Software option	Definition and application
Batch Process Mngr. (#154 / #2-05-1)	Batch Process Manager (BPM) This software option makes it easy to plan and execute multiple production jobs. By extending and combining the pallet management and extended tool management functions (#93 / #2-03-1), BPM offers the following additional data, for example: ■ Machining time ■ Availability of necessary tools ■ Manual interventions to be made ■ Program test results of assigned NC programs Further information: Programming and Testing User's Manual
Component Monitoring (#155 / #5-02-1)	Component monitoring This software option enables the automatic monitoring of machine components configured by the machine manufacturer. Component monitoring assists the control in preventing machine damage due to overload by way of hazard warnings and error messages.
Model Aided Setup (#159 / #1-07-1)	Graphically supported setup This software option is used to determine the position and misalignment of a workpiece with only one touch-probe function. You can probe complex workpieces with, for example, free-form surfaces or undercuts, which is not possible with all of the other touch-probe functions. The control supports you additionally by showing the setup situation and possible touch points in the Simulation workspace by means of a 3D model. Further information: User's Manual for Setup and Program Run
Opt. Contour Milling (#167 / #1-02-1)	Optimized contour machining (OCM) This software option enables 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. The software option includes the following cycles: ■ 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 provides OCM STANDARD FIGURES for frequently needed contours With OCM you can shorten machining times while at the same time reducing tool wear. Further information: User's Manual for Machining Cycles

Software option	Definition and application
PTI 8 channels (#3-04-1*)	Process Tracking Interface PTI This software option provides an interface for recording control and process signals at a scanning interval as short as 3 ms. With PTI, you can record live information about axes and spindles. The signals can be used, among other things, to create a digital workpiece twin in order to meet documentation requirements in the aerospace industry. This software option is available only for controls with SIK2. You can order this software option up to two times, each time selecting up to eight channels for recording. Further information: User's Manual for Setup and Program Run  For more information, refer to the integrated help system of RemoTools SDK. ID 340442-xx

3.3.2 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 License Viewer** window.

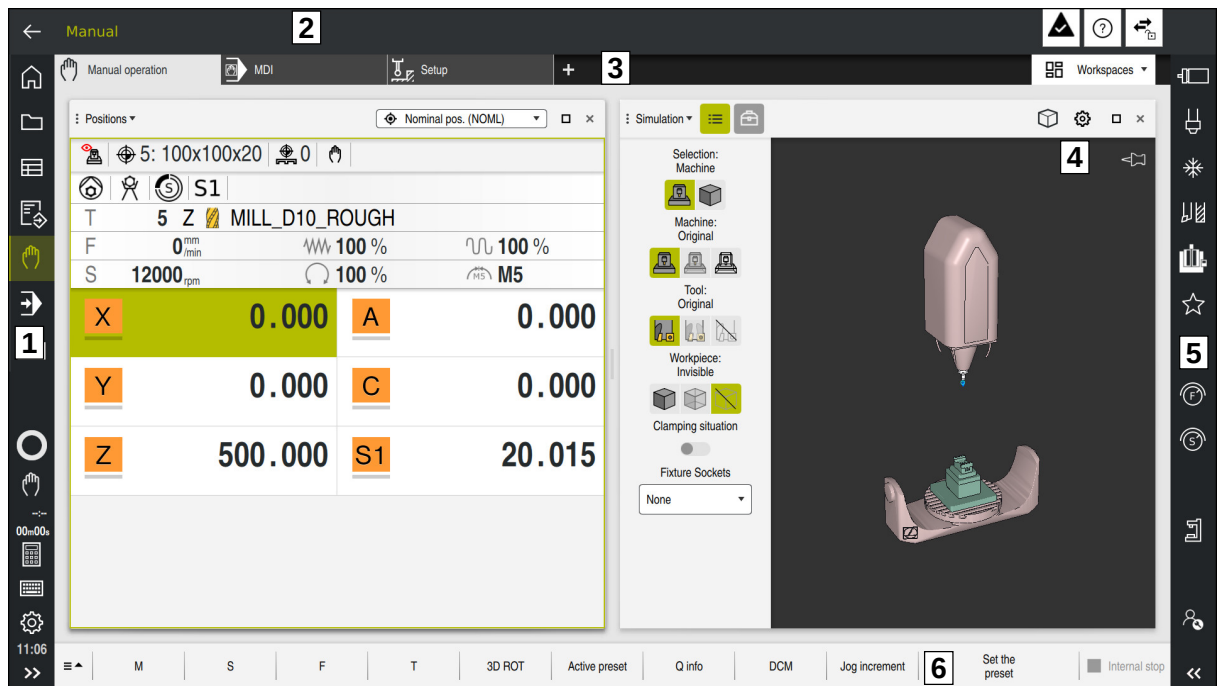
OPC UA

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

The control's behavior can be influenced by means of the OPC UA NC Server (#56-61 / #3-02-1*) and HEIDENHAIN DNC (#18 / #3-03-1). 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

3.4 Areas of the control's user interface



User interface of the control in the **Manual operation** application

The control's user interface shows the following areas:

1 TNC bar

■ Back

Use this function to go backwards in the application history since booting the control.

■ Operating modes

Further information: "Overview of the operating modes", Page 51

■ Status overview

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

■ Calculator

Further information: Programming and Testing User's Manual

■ Screen keyboard

■ Settings

The Settings menu enables you to change the control interface:

■ Left-hand mode

The control swaps the positions of the TNC bar and the machine manufacturer bar.

■ Dark Mode

In the machine parameter **darkModeEnable** (no. 135501), the machine manufacturer defines whether the **Dark Mode** function is available for selection.

This manual describes the state of the GUI with inactive **Dark Mode**. Your control might display other colors than shown in this User's Manual.

■ Font size

- Date and time
- 2 Information bar
 - Active operating mode
 - Message menu
 - Symbols
- 3 Application bar
 - Tabs of opened applications

The maximum number of simultaneously opened applications is limited to ten tabs. If you try to open an eleventh tab, the control shows a message.
 - Selection menu for workspaces

With the selection menu you define which workspaces are open in the active application.
- 4 Workspaces
- 5 Machine manufacturer bar

The machine manufacturer configures the machine manufacturer bar.
- 6 Function bar
 - Selection menu for buttons

With the selection menu you define which buttons the control displays in the function bar.
 - Button

With the buttons you activate individual functions of the control.



Refer to your machine manual.

The machine parameter **state** (no. 143601) allows the machine manufacturer to dim or hide the following user interface elements, for example:

- Operating modes
- Applications
- Workspaces
- Buttons

3.5 Overview of the operating modes

The control provides the following operating modes:

Icon	Operating modes	Further information
	The Home operating mode contains the following applications: ■ The Start/Login application During the startup process, the control is in the Start/Login application. ■ The Settings application ■ The Help application ■ Applications for machine parameters	See the User's Manual for Setup and Program Run See the User's Manual for Programming and Testing See the User's Manual for Setup and Program Run
	In the Files operating mode the control displays drives, folders and files. You can, for example, create or delete folders or files and can also connect drives.	See the User's Manual for Programming and Testing
	In the Tables operating mode you can open various tables and edit them as necessary.	
	In the Editor operating mode you can do the following: ■ Create, edit and simulate NC programs ■ Create and edit contours ■ Create and edit pallet tables	See the User's Manual for Programming and Testing
	The Manual operating mode contains the following applications: ■ The Manual operation application ■ The MDI Application ■ The Setup application ■ The Move to ref. point application ■ The Retract application You can move the tool away from the workpiece, for example after a power failure.	See the User's Manual for Setup and Program Run See the User's Manual for Setup and Program Run See the User's Manual for Setup and Program Run See the User's Manual for Setup and Program Run See the User's Manual for Setup and Program Run
	In the Program Run operating mode you produce workpieces by having the control execute NC programs either block-by-block or in full sequence. You also execute pallet tables in this operating mode.	See the User's Manual for Setup and Program Run
	If the machine manufacturer has defined an embedded workspace, then you can open full-screen mode with this operating mode. The machine manufacturer defines the name of the operating mode. Refer to your machine manual.	See the User's Manual for Setup and Program Run

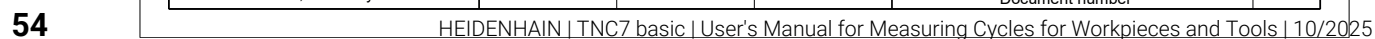
Icon	Operating modes	Further information
	In the Machine operating mode, the machine manufacturers define their own functions, such as diagnostic functions for spindle and axes, or other applications. Refer to your machine manual.	
	Refer to your machine manual. The machine parameter state (no. 143601) allows the machine manufacturer to gray out or hide the following user interface elements, for example: ■ Operating modes ■ Applications ■ Buttons	

4

First steps

4.1.1 Programming the workpiece

Sample task 1338459-00-A-01



Selecting the Editor operating mode

NC programs are always programmed in the **Editor** operating mode.

Requirement

- It must be possible to select the icon of the operating mode
In order to be able to select the **Editor** operating mode, the control must have already progressed enough during booting that the operating mode icon is no longer dimmed.

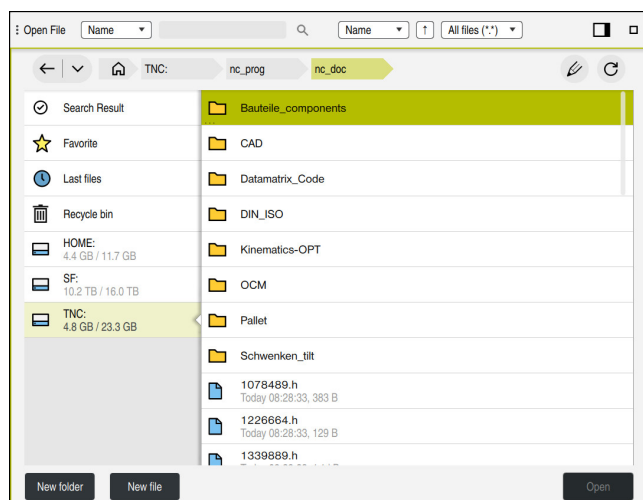
Selecting the Editor operating mode

To select the **Editor** operating mode:



- ▶ Select the **Editor** operating mode
- > The control displays the **Editor** operating mode and the most recently opened NC program.

Creating a new NC program



The **Open File** workspace in the **Editor** operating mode

To create an NC program in the **Editor** operating mode:



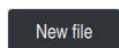
- ▶ Select **Add**
- The control shows the **Quick selection** and **Open File** workspaces.



- ▶ Select the desired drive in the **Open File** workspace



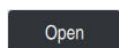
- ▶ Select a folder



- ▶ Select **New file**
- ▶ Enter a file name (e.g., 1338459.h)



- ▶ Confirm with the **ENT** key



- ▶ Select **Open**
- The control opens a new NC program and the **Insert NC function** window for definition of the workpiece blank.

More detailed information

- The **Open File** workspace
Further information: User's Manual for Setup and Program Run
- The **Editor** operating mode
Further information: Programming and Testing User's Manual

Configuring the control's user interface for programming

The **Editor** operating mode gives you several possibilities for writing an NC program.

i The first steps describe the procedure when you are in the **Klartext editor** mode with the **Form** column open.

Opening the Form column

You can open the **Form** column only if an NC program is open.

To open the **Form** column:

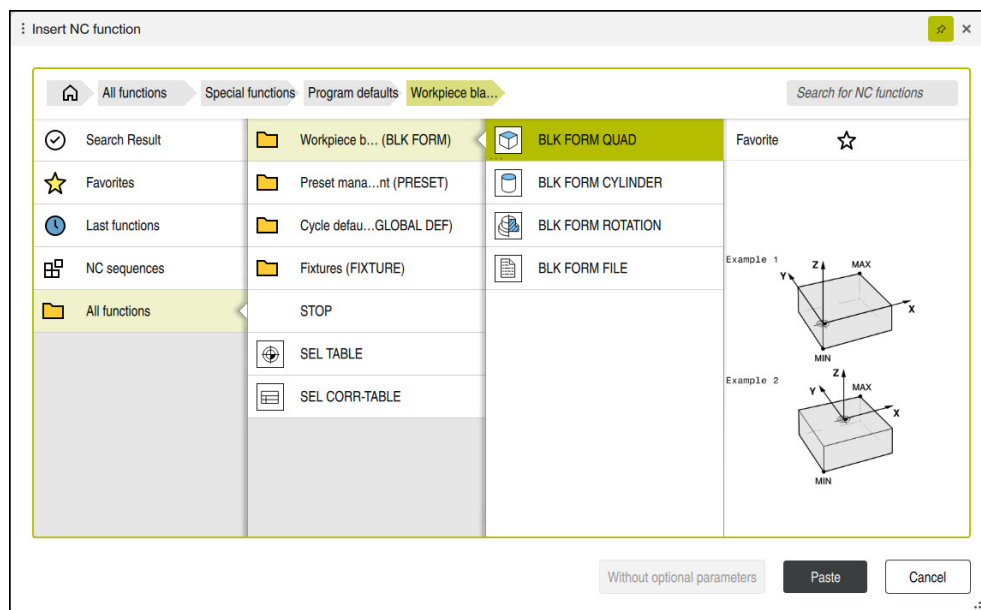


- ▶ Select **Form**
- > The control opens the **Form** column

Defining the workpiece blank

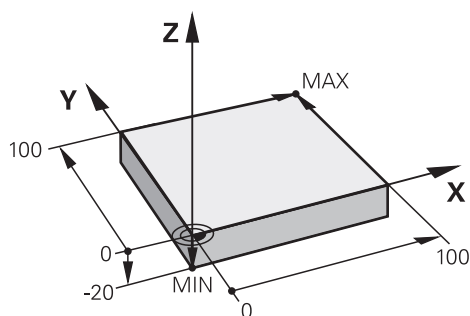
For the NC program you can define a workpiece blank that the control then uses for the simulation. When you create an NC program, the control automatically opens the **Insert NC function** window for workpiece blank definition.

i If you close the window without selecting a workpiece blank, you can use the **Insert NC function** button to select the workpiece blank definition later.



The **Insert NC function** window for workpiece blank definition

Defining a cuboid workpiece blank



Cuboid workpiece blank with minimum point and maximum point

You define a cuboid through a diagonal in space by entering the minimum point and maximum point relative to the active workpiece preset.



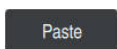
You can confirm the entries as follows:

- **ENT** key
- Right arrow key
- Click or tap the next syntax element

To define a cuboid workpiece blank:



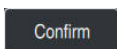
- ▶ Select **BLK FORM QUAD**



- ▶ Select **Paste**
- The control inserts the NC block for definition of the workpiece blank.



- ▶ Open the **Form** column
- ▶ Select the tool axis (e.g., **Z**)
- ▶ Confirm your input
- ▶ Enter the smallest X coordinate (e.g., **0**)
- ▶ Confirm your input
- ▶ Enter the smallest Y coordinate (e.g., **0**)
- ▶ Confirm your input
- ▶ Enter the smallest Z coordinate (e.g., **-20**)
- ▶ Confirm your input
- ▶ Enter the largest X coordinate (e.g., **100**)
- ▶ Confirm your input
- ▶ Enter the largest Y coordinate (e.g., **100**)
- ▶ Confirm your input
- ▶ Enter the largest Z coordinate (e.g., **0**)
- ▶ Confirm your input



- ▶ Select **Confirm**
- The control concludes the NC block.

Working spindle axis

X Y **Z**

Workpiece blank def.: MIN point

X

0

×

Y

0

×

Z

-40

×

Workpiece blank def.: MAX point

X

100

×

Y

100

×

Z

0

×

Comment

;

Confirm

Discard

Delete line

The **Form** column with the defined columns

0	BEGIN PGM	MM
1	BLK FORM 0.1	Z X+0 Y+0 Z-20
2	BLK FORM 0.2	X+100 Y+100 Z+0
3	END PGM	MM



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

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

More detailed information

- Inserting the workpiece blank
- Reference points in the machine

Structure of an NC program

Using a uniform structure for an NC program offers the following advantages:

- Improved overview
- Quicker programming
- Fewer sources of error

Recommended structure for a contouring program



The control automatically inserts the **BEGIN PGM** and **END PGM** NC blocks.

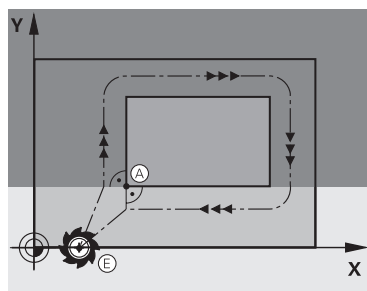
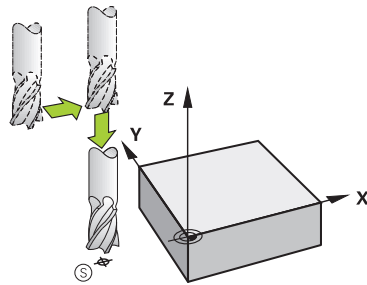
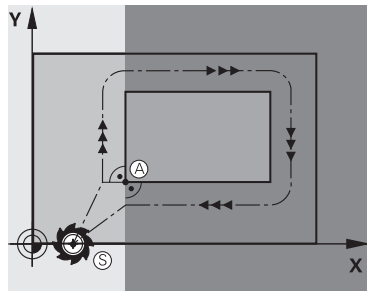
- 1 **BEGIN PGM** with selection of the unit of measure
- 2 Define the workpiece blank
- 3 Call the tool, with the tool axis and the technological data
- 4 Move the tool to a safe position, and switch the spindle on
- 5 Pre-position the tool in the working plane, near the first contour point
- 6 Pre-position the tool in the tool axis, turn coolant on if necessary
- 7 Approach the contour, activate tool radius compensation if necessary
- 8 Machine the contour
- 9 Depart from the contour, turn coolant off
- 10 Move the tool to a safe position
- 11 Conclude the NC program
- 12 **END PGM**

Contour approach and departure

When you program a contour, you need a starting point and end point outside the contour.

The following positions are necessary for contour approach and departure:

Help graphic



Position

Starting point

The following preconditions apply for the starting point:

- No tool radius compensation
- Approachable without danger of collision
- Near to the first contour point

The graphic shows the following information:

If you define the starting point to be in the dark gray area, the contour will be damaged when the first contour point is approached.

Approaching the starting point in the tool axis

Before approaching the first contour point, you must position the tool to the working depth in the tool axis. If there is a danger of collision, approach the starting point in the tool axis separately.

First contour point

The control moves the tool from the starting point to the first contour point.

You need to program tool radius compensation for the tool movement to the first contour point.

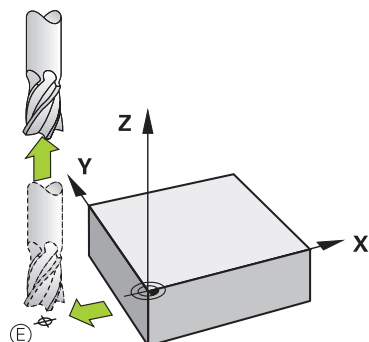
End point

The following preconditions apply for the end point:

- Approachable without danger of collision
- Near to the last contour point
- In order to make sure that the contour will not be damaged, the optimal ending point should lie on the extended tool path for machining the last contour element

The graphic shows the following information:

If you define the end point to be in the dark gray area, the contour will be damaged when the end point is approached.

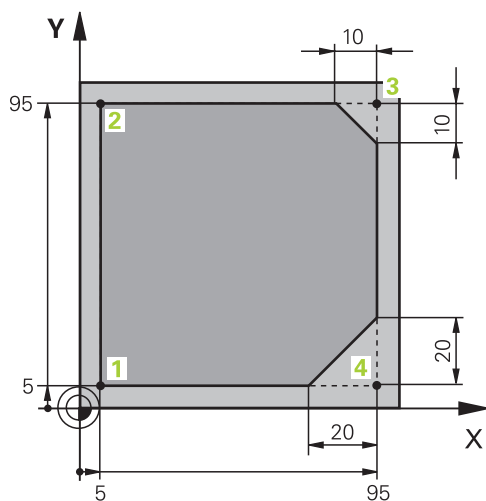
Help graphic**Position****Departing from the end point in the tool axis**

Program the tool axis separately when departing from the end point.

Identical starting and end points

Do not program any tool radius compensation if the starting point and end point are the same.

In order to make sure that the contour will not be damaged, the optimal starting point should lie between the extended tool paths for machining the first and last contour elements.

Programming a simple contour

Workpiece to be programmed

The following texts show you how to mill once at a depth of 5 mm around the contour shown here. You have already defined the workpiece blank.

Further information: "Defining the workpiece blank", Page 57

After you have inserted an NC function, the control shows an explanation about the current syntax element in the dialog bar. You can enter the data directly in the form.



Always write an NC program as if the tool were moving. This makes it irrelevant whether a head axis or a table axis performs the motion.

Calling a tool

The **Form** column with the syntax elements of the tool call

To call a tool:

TOOL
CALL

- ▶ Select **TOOL CALL**
- ▶ Select **Number** in the form
- ▶ Enter the tool number (e.g., **16**)
- ▶ Select the tool axis **Z**
- ▶ Select the spindle speed **S**
- ▶ Enter the spindle speed (e.g., **6500**)
- ▶ Select **Confirm**
- > The control concludes the NC block.

Confirm

3 TOOL CALL 16 Z S6500



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

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

Move the tool to a safe position

The screenshot shows the 'Form' column of the TNC7 basic control interface. It contains a list of axes (Z, A, B, C, U, V, W, & X, & Y, & Z) with input fields and a 'Radius compensation' section with buttons for R0, RL, and RR. The 'Confirm' button is highlighted in green.

The **Form** column with the syntax elements of a straight line

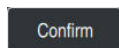
To move the tool to a safe position:



- ▶ Select the path function **L**



- ▶ Select **Z**
- ▶ Enter a value (e.g., **250**)
- ▶ Select tool radius compensation **R0**
- ▶ The control applies **R0**, which means there is no tool radius compensation.
- ▶ Select the **FMAX** feed rate
- ▶ The control adopts **FMAX** for rapid traverse.
- ▶ If needed, enter a miscellaneous function **M**, such as **M3** (turn spindle on)



- ▶ Select **Confirm**
- ▶ The control concludes the NC block.

4 L Z+250 R0 FMAX M3

Pre-positioning in the working plane

To pre-position in the working plane:



- Select the path function **L**



- ▶ Select **X**
- ▶ Enter a value (e.g., **-20**)



- ▶ Select **Y**
- ▶ Enter a value (e.g., **-20**)
- ▶ Select the **FMAX** feed rate



- ▶ Select **Confirm**
- The control concludes the NC block.

5 L X-20 Y-20 FMAX

Pre-positioning in the tool axis

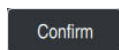
To pre-position in the tool axis:



- Select the path function **L**



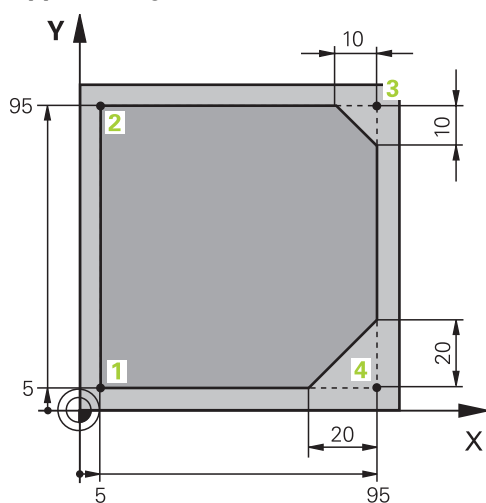
- ▶ Select **Z**
- ▶ Enter a value (e.g., **-5**)
- ▶ Select the feed rate **F**
- ▶ Enter the value for the positioning feed rate (e.g., **3000**)
- ▶ If needed, enter a miscellaneous function **M**, such as **M8** (turn coolant on)



- ▶ Select **Confirm**
- The control concludes the NC block.

6 L Z-5 F3000 M8

Approaching the contour



Workpiece to be programmed

The screenshot shows a 'Form' column in a CNC control interface. It contains several input fields and buttons for configuring an approach function. The fields are: 'Center angle' with 'CCA' set to '90'; 'Algebraic sign (+/-) and radius of the circular path' with 'R' set to '8'; 'Radius compensation' with 'RL' selected; 'Feed rate' with 'F' set to '700'; and 'M-Functions' with 'M' set to an empty field. At the bottom, there are three buttons: 'Confirm', 'Discard', and 'Delete line'.

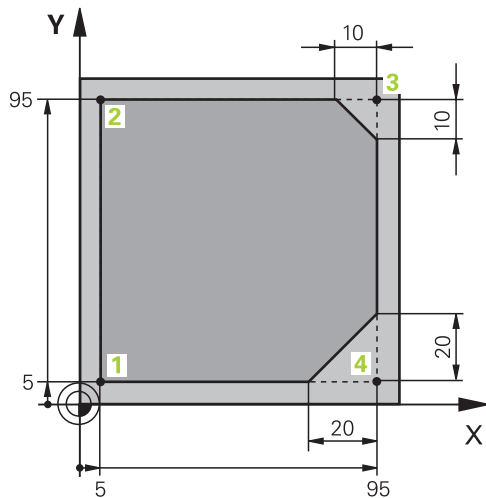
The **Form** column with the syntax elements of an approach function

To approach the contour:

- ▶ Select the **APPR DEP** path function
- ▶ The control opens the **Insert NC function** window.
- ▶ Select **APPR**
- ▶ Select an approach function (e.g., **APPR CT**)
- ▶ Select **Paste**
- ▶ Enter the coordinates of the starting point **1** (e.g., **X 5 Y 5**)
- ▶ For the center angle **CCA**, enter the approach angle (e.g., **90**)
- ▶ Enter the radius of the circular arc (e.g., **8**)
- ▶ Select **RL**
- ▶ The control applies tool radius compensation to the left.
- ▶ Select the feed rate **F**
- ▶ Enter the value for the machining feed rate (e.g., **700**)
- ▶ Select **Confirm**
- ▶ The control concludes the NC block.

7 APPR CT X+5 Y+5 CCA90 R+8 RL F700

Machining a contour



Workpiece to be programmed

To machine the contour:



Confirm

- ▶ Select the path function **L**
- ▶ Enter the coordinates of contour point **2** that differ (e.g., **Y 95**)
- ▶ Conclude the NC block with **Confirm**
- ▶ The control applies the changed value and retains all of the other information from the previous NC block.



Confirm

- ▶ Select the path function **L**
- ▶ Enter the coordinates of contour point **3** that differ (e.g., **X 95**)
- ▶ Conclude the NC block with **Confirm**



Confirm

- ▶ Select the path function **CHF**
- ▶ Enter the chamfer width (e.g., **10**)
- ▶ Conclude the NC block with **Confirm**



Confirm

- ▶ Select the path function **L**
- ▶ Enter the coordinates of contour point **4** that differ (e.g., **Y 5**)
- ▶ Conclude the NC block with **Confirm**



Confirm

- ▶ Select the path function **CHF**
- ▶ Enter the chamfer width (e.g., **20**)
- ▶ Conclude the NC block with **Confirm**



Confirm

- ▶ Select the path function **L**
- ▶ Enter the coordinates of contour point **1** that differ (e.g., **X 5**)
- ▶ Conclude the NC block with **Confirm**

8 L Y+95

9 L X+95

10 CHF 10

11 L Y+5

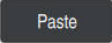
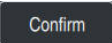
12 CHF 20

13 L X+5

Departing from the contour

The **Form** column with the syntax elements of a departure function

To depart from the contour:

- 
 - ▶ Select the **APPR DEP** path function
 - The control opens the **Insert NC function** window.
- 
 - ▶ Select **DEP**
- 
 - ▶ Select a departure function (e.g., **DEP CT**)
- 
 - ▶ Select **Paste**
 - ▶ For the center angle **CCA**, enter the departure angle (e.g., **90**)
 - ▶ Enter the departure radius (e.g., **8**)
 - ▶ Select the feed rate **F**
 - ▶ Enter the value for the positioning feed rate (e.g., **3000**)
 - ▶ If needed, enter a miscellaneous function **M**, such as **M9** (turn coolant off)
- 
 - ▶ Select **Confirm**
 - The control concludes the NC block.

14 DEP CT CCA90 R+8 F3000 M9

Solution 1338459-00-A-01

If you follow all steps described here, you will obtain the following NC program:

0 BEGIN PGM "1338459-00-A-01" MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 16 Z S6500	
4 L Z+250 R0 FMAX M3	
5 L X-20 Y-20 FMAX	
6 L Z-5 F3000 M8	
7 APPR CT X+5 Y+5 CCA90 R+8 RL F700	
8 L Y+95	
9 L X+95	
10 CHF 10	
11 L Y+5	
12 CHF 20	
13 L X+5	
14 DEP CT CCA90 R+8 F3000 M9	
15 L Z+250 R0 FMAX	
16 TOOL CALL 6 Z S6500	
17 L Z+250 R0 FMAX M3	
18 L X+50 Y+50 FMAX	
19 CYCL DEF 254 CIRCULAR SLOT ~	
Q215=+0 ;ROUGHING AND FINISHING ~	
Q219=+15 ;SLOT WIDTH ~	
Q368=+0.1 ;ALLOWANCE FOR SIDE ~	
Q375=+60 ;PITCH CIRCLE DIAMETR ~	
Q367=+0 ;REF. SLOT POSITION ~	
Q216=+50 ;CENTER IN 1ST AXIS ~	
Q217=+50 ;CENTER IN 2ND AXIS ~	
Q376=+45 ;STARTING ANGLE ~	
Q248=+225 ;ANGULAR LENGTH ~	
Q378=+0 ;STEPPING ANGLE ~	
Q377=+1 ;NR OF REPETITIONS ~	
Q207=+500 ;FEED RATE MILLING ~	
Q351=+1 ;CLIMB MILLING ~	
Q201=-5 ;DEPTH ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q369=+0.1 ;ALLOWANCE FOR FLOOR ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q338=+5 ;INFEEED FOR FINISHING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	

Q204=+50	;2ND SET-UP CLEARANCE ~	
Q366=+2	;RECIPROCATING PLUNGE ~	
Q385=+500	;FINISHING FEED RATE ~	
Q439=+0	;CENTER POINT PATH	
20 CYCL CALL		
21 L Z+250 R0 FMAX		
22 M30		
23 END PGM "1338459-00-A-01" MM		

4.1.2 Simulating a workpiece

Configuring the control's user interface for simulation

In the **Editor** operating mode you can test NC programs graphically. The control simulates the active NC program in the **Program** workspace.

In order to simulate the NC program you must open the **Simulation** workspace.



For the simulation you can close the **Form** column to get a better view of the NC program and the **Simulation** workspace.

Opening the Simulation workspace

You can open additional workspaces in the **Editor** operating mode only if an NC program is open.

To open the **Simulation** workspace:

- ▶ In the application bar, select **Workspaces**
- ▶ Select **Simulation**
- > The control then additionally displays the **Simulation** workspace.



You can also open the **Simulation** workpiece with the **Test Run** operating mode key.

Configuring the Simulation workspace

You can simulate the NC program without needing to enter any special settings. However, an adjustment to the simulation speed is recommended for best viewing of the simulation.

To adjust the speed of the simulation:

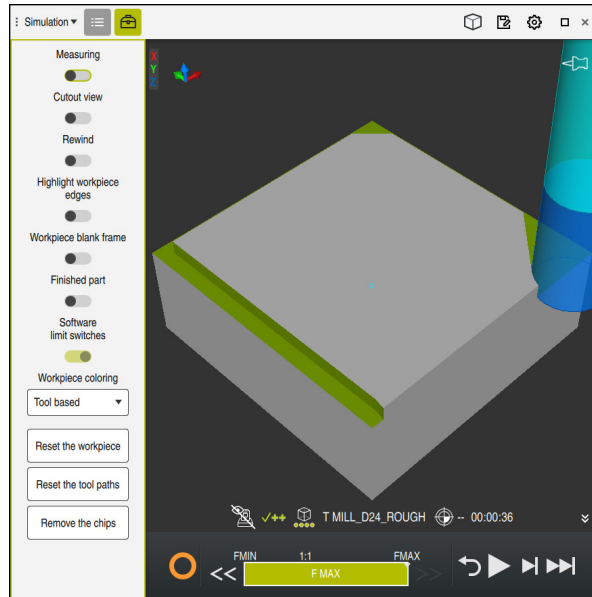
- ▶ Use the slider to select the factor (e.g., **5.0 * T**)
- > The control then performs the subsequent simulation at five times the speed of the programmed feed rate.

If you use different tables, such as tool tables, for program run and the simulation, then you can define the tables in the **Simulation** workspace.

Simulating an NC program

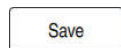
You can test the NC program in the **Simulation** workspace.

Starting the simulation



The **Simulation** workspace in the **Editor** operating mode

To start the simulation:



- ▶ Select **Start**
- > The control might ask whether the file should be saved.
- ▶ Select **Save**
- > The control starts the simulation.
- > The control uses the **Control-in-operation** symbol to show the simulation status.

Definition

Control-in-operation:

The control uses the **Control-in-operation** icon to show the current simulation status in the action bar and on the tab of the NC program:

- White: no movement command
- Green: active machining, axes are moving
- Orange: NC program interrupted
- Red: NC program stopped

5

**NC and
programming
fundamentals**

5.1 Working with cycles

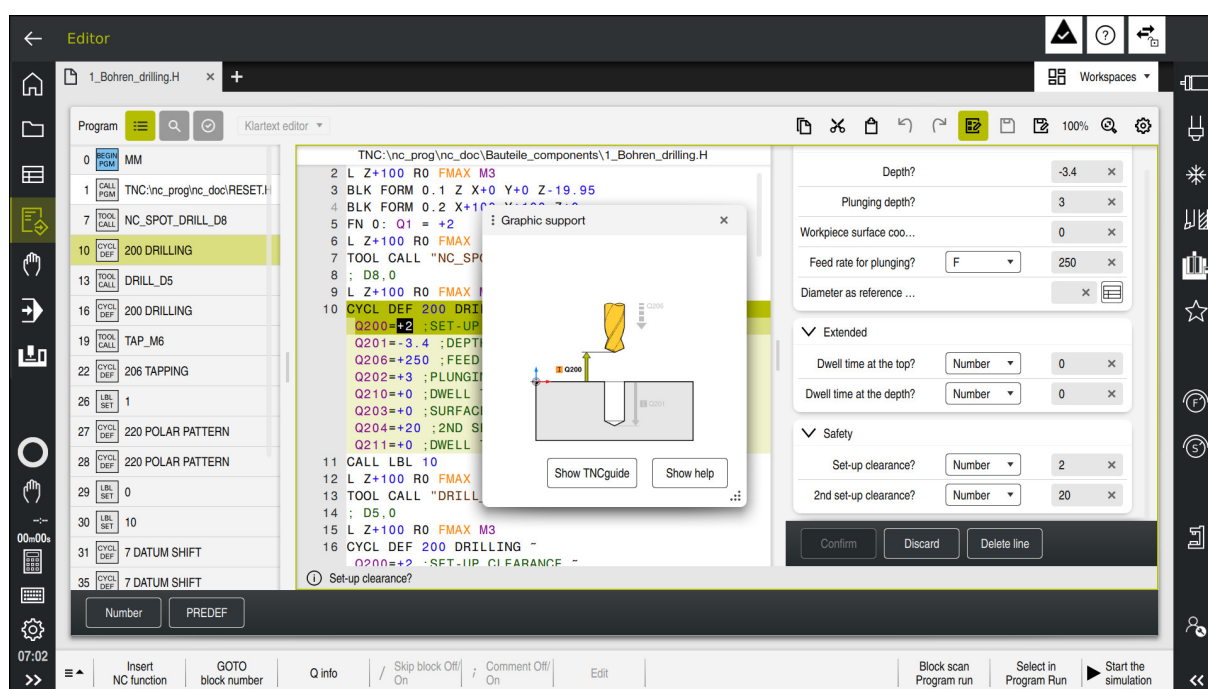
5.1.1 General information on cycles

General



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

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



Cycles are stored on the control as subprograms. The cycles can be used to execute different machining operations, thereby enabling the easy and efficient programming of recurring machining operations. Most cycles use Q parameters as transfer parameters. The control offers cycles for the following machining technologies:

- Drilling processes
- Thread machining
- Milling operations such as pockets and studs or even contours
- Cycles for coordinate transformation

NOTICE

Danger of collision!

Cycles sometimes perform complex machining movements. There is therefore a risk of collision!

- Simulate your program before executing it

NOTICE**Danger of collision!**

You can program variables as input values in HEIDENHAIN cycles. If you define cycle parameters with variables, then use only the input range of the cycle parameter. Otherwise, collisions may occur.

- ▶ Use only the input range of the cycle parameter
- ▶ Pay attention to the HEIDENHAIN documentation
- ▶ Check the machining sequence using a simulation



In inch programs, the feed rate for cycles must be defined in 0.1 inch/min.

Optional parameters

HEIDENHAIN continuously further develops the extensive package of cycles. Additional cycle parameters for cycles may become available with every new software version. These new Q parameters are **optional parameters**, some of which were not contained in previous software versions. These parameters are located at the end of the cycle definition.

To see which optional Q parameters were added to your current software version, refer to the "New and modified functions" overview.

You have the following alternatives when inserting a cycle:

- Insert the cycle with all optional parameters. Every optional parameter can be hidden using the **NO ENT** key.
- Insert the cycle with only its required parameters. In this case, you insert the cycle without optional parameters.



If you hide a Q parameter or insert a cycle with only its required parameters, then the values of the optional Q parameters that are not shown are their default values.

To remove optional parameters in the NC program:

Insert
NC function

Paste



NO
ENT

- ▶ Select **Insert NC function**
- The control opens the **Insert NC function** window.
- ▶ Select the desired cycle
- ▶ Select **Paste**
- The control inserts the cycle definition with all cycle parameters into the NC program.
- ▶ Navigate to the optional cycle parameters by pressing the arrow key that points to the right
- ▶ Press the **NO ENT** key to hide the optional cycle parameter

Insert the cycle with only its required parameters:

- ▶ Select **Insert NC function**
- The control opens the **Insert NC function** window.
- ▶ Select the desired cycle



- ▶ Select **Without optional parameters**
- The control inserts the cycle definition without optional cycle parameters into the NC program.

If you unintentionally hide an optional parameter or need it again at a later time, then you can once again show and edit it within the cycle.

To show optional parameters:

- ▶ Edit cycle



- ▶ Navigate to the desired optional cycle parameters by pressing the arrow key that points to the right
- ▶ Change the default value as needed



- ▶ Confirm the optional cycle parameter by pressing **ENT**



- ▶ Save the change by pressing **END**

Compatibility

The TNC7 basic can usually execute the NC programs of predecessor controls starting with the TNC 150 B and later even though the cycles have been extended to include additional optional parameters.

This is possible thanks to the stored default value. To execute a current NC program on an older control or software version, remove the optional cycle parameters. This gives you a backward-compatible NC program. If NC blocks contain invalid elements, then the control indicates that these elements are ERROR blocks.

Defining cycles

Cycles can be defined in several ways.

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 machining cycles via the CYCL DEF key:



- ▶ Press the **CYCL DEF** 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.

Inserting touch-probe cycles via the TOUCH PROBE key:



- ▶ Press 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



For some cycle parameters, the control provides selectable choices via the action bar or the form.

If an input option specifying a defined behavior is stored in certain cycle parameters, you can open a selection list with the **GOTO** key or in the form view. For example, in Cycle **200 DRILLING** the **Q395 DEPTH REFERENCE** parameter provides the following options:

- 0 | Tool tip
- 1 | Cutting edge corner

Cycle input form

The control provides a **FORM** for various functions and cycles. This **FORM** allows you to enter various syntax elements or cycle parameters.

The screenshot shows a 'Cycle input form' with two main sections: 'Geometry' and 'Default'. Each section contains several input fields with numerical values and a clear button (X). At the bottom, there are three buttons: 'Confirm', 'Discard', and 'Delete line'.

Section	Parameter	Value	Action
Geometry	First side length?	60	X
	Second side length?	20	X
	Corner radius?	0	X
	Depth?	-20	X
	Workpiece surface co...	0	X
Default	Machining operation (...)	0	X [Icon]
	Plunging depth?	5	X
	Infeed for finishing?	0	X
	Feed rate for milling?	F 500	X
	Finishing feed rate?	F 500	X
	Feed rate for plunging?	F 150	X

Buttons at the bottom: Confirm, Discard, Delete line

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.

To open the form:



- ▶ Select the **Editor** operating mode



- ▶ Select the desired **Program**
- ▶ 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.

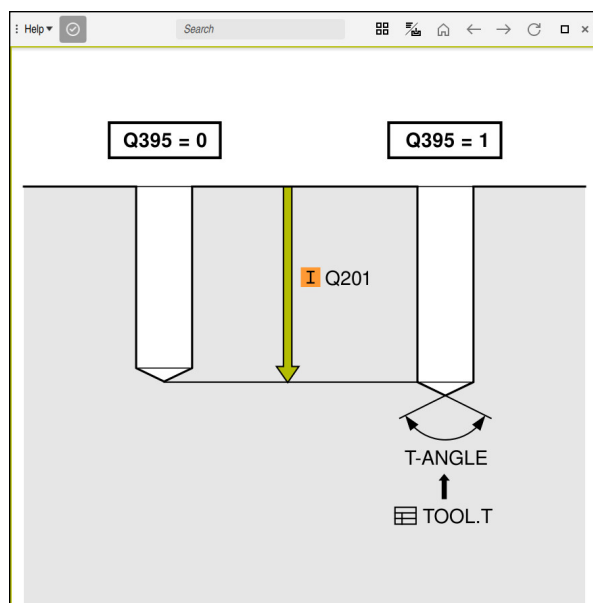
Help graphics

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.

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.



The **Help** workspace with a help graphic for a cycle parameter

Calling cycles

For cycles that remove material, you have to enter not only the cycle definition, but also the cycle call in the NC program. The call always refers to the machining cycle that was defined last in the NC program.

Requirements

Before calling a cycle, be sure to program:

- **BLK FORM** for graphic display (only required for simulation)
- Tool call
- Spindle direction of rotation (miscellaneous function **M3/M4**)
- Cycle definition (**CYCL DEF**)



For some cycles, additional requirements must be observed. They are detailed in the descriptions and overview tables for each cycle.

You can program the cycle call in the following ways:

Syntax	Further information
CYCL CALL	Page 80
CYCL CALL PAT	Page 80
CYCL CALL POS	Page 81
M89/M99	Page 81

Calling a cycle with **CYCL CALL**

The **CYCL CALL** function calls the most recently defined machining cycle once. The starting point of the cycle is the position that was programmed most recently before the **CYCL CALL** block.



- ▶ Select **Insert NC function**

or



- ▶ Press the **CYCL CALL** key

➤ The control opens the **Insert NC function** window.

- ▶ Select **CYCL CALL M**

- ▶ Define **CYCL CALL M** and add an M function, if necessary

Calling a cycle with **CYCL CALL PAT**

The **CYCL CALL PAT** function calls the most recently defined machining cycle at all positions that you defined in a **PATTERN DEF** pattern definition or in a point table.

Further information: User's Manual for Machining Cycles



- ▶ Select **Insert NC function**

or



- ▶ Press the **CYCL CALL** key

➤ The control opens the **Insert NC function** window.

- ▶ Select **CYCL CALL PAT**

- ▶ Define **CYCL CALL PAT** and add an M function , if necessary

Calling a cycle with CYCL CALL POS

The **CYCL CALL POS** function calls the most recently defined machining cycle once. The starting point of the cycle is the position that you defined in the **CYCL CALL POS** block.



- ▶ Select **Insert NC function**

or



- ▶ Press the **CYCL CALL** key

- The control opens the **Insert NC function** window.

- ▶ Select **CYCL CALL POS**

- ▶ Define **CYCL CALL POS** and add an M function as needed

Using positioning logic, the control moves to the position defined in the **CYCL CALL POS** block:

- If the tool's current position in the tool axis is above the upper edge of the workpiece (**Q203**), the control first moves the tool to the programmed position in the working plane and then to the programmed position in the tool axis
- If the tool's current position in the tool axis is below the upper edge of the workpiece (**Q203**), the control first moves the tool to the clearance height in the tool axis and then to the programmed position in the working plane



Programming and operating notes

- Three coordinate axes must always be programmed in the **CYCL CALL POS** block. Using the coordinate in the tool axis, you can easily change the starting position. It serves as an additional datum shift.
- The feed rate most recently defined in the **CYCL CALL POS** block is only used to traverse to the start position programmed in this block.
- As a rule, the control moves without radius compensation (R0) to the position defined in the **CYCL CALL POS** block.
- If you use **CYCL CALL POS** to call a cycle in which a start position is defined (e.g., Cycle **212**), then the position defined in the cycle serves as an additional shift of the position defined in the **CYCL CALL POS** block. You should therefore always define the start position in the cycle as 0.

Calling cycles with additional functions

M99

The **M99** miscellaneous function calls the most recently defined machining cycle once. **M99** is effective blockwise and at the end of the block (e.g., after the traverse movement)

Example

```
11 CYCL DEF 257 CIRCULAR STUD
```

```
...
```

```
12 L X+50 Y+50 R0 FMAX M99
```

The control traverses at **FMAX** to the position **X+50** and **Y+50**. Then the control calls Machining Cycle **257 CIRCULAR STUD** with **M99**.

M89

If the control is to execute the cycle automatically after every positioning block, program the first cycle call with **M89**.

You can cancel **M89** with the following functions:

- **M99** at the last position
- New machining cycle with **CYCL DEF**

Defining and calling an NC program as cycle

With **SEL CYCLE**, you can define any NC program as a machining cycle.

To define an NC program as a cycle:



- ▶ Select **Insert NC function**
- The control opens the **Insert NC function** window.
- ▶ Select **SEL CYCLE**
- ▶ Select file name, string parameter, or file

To call an NC program as a cycle:



- ▶ Press the **CYCL CALL** key
- The control opens the **Insert NC function** window.
or
- ▶ Program **M99**



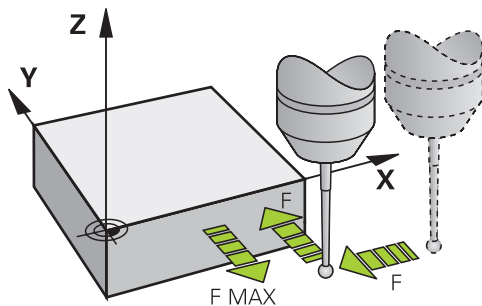
- If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path.
- **CYCL CALL PAT** and **CYCL CALL POS** use a positioning logic before the respective cycle is executed. With respect to the positioning logic, the **SEL CYCLE** and Cycle **12 PGM CALL** show the same behavior. In point pattern cycles, the clearance height for approaching is calculated based on:
 - The maximum Z position at the start of the pattern
 - All Z positions in the point pattern
- With **CYCL CALL POS**, there will be no pre-positioning in the tool-axis direction. This means that you need to manually program any pre-positioning in the file you call.

5.1.2 General information about touch probe cycles

Description of function



- Refer to your machine manual.
- The control must be specifically prepared by the machine manufacturer for the use of a touch probe.
- HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.
- If you are using a HEIDENHAIN touch probe with EnDat interface, then the Touch Probe Functions software option (#17 / #1-05-1) is automatically enabled.
- The control's full range of functions is available only if the **Z** tool axis is used.
- Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.



The touch-probe functions allow you to determine workpiece misalignment and compensate for it as well as set presets on the workpiece and measure the workpiece.

Whenever the control executes a touch probe cycle, the 3D touch probe approaches the workpiece parallel to a machine axis. This is also true during an active basic rotation or with a tilted working plane.

When the probe stylus contacts the workpiece,

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

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

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

- Touch Probe Functions (#17 / #1-05-1) software option
- Calibrated workpiece touch probe

Working with an L-shaped stylus

In addition to a simple stylus (**SIMPLE**), the touch-probe cycles **444** and **14xx** support the L-shaped stylus (**L-TYPE**). The L-shaped stylus must be calibrated prior to use.

HEIDENHAIN recommends calibrating the stylus with the following cycles:

- Radius calibration:
- Length calibration:

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



- The control does not show the arm of the stylus in the simulation. The arm is the angled part of the L-shaped stylus.
- The Collision Monitoring (#140 / #5-03-2) software option does not monitor the L-shaped stylus.
- In order to achieve maximum accuracy, the feed rate during calibration must be identical to the feed rate during probing.

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

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

General information on the touch-probe table

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 touch-probe cycle
- Maximum measuring range
- Set-up 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

Touch probe cycles in the Manual Operation and Electronic Handwheel modes

In the **Setup** application, the control provides touch probe cycles in **Manual** mode 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

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 functionality, which the control requires in various cycles, always have the same number: For example, **Q260** is always 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 **Editor** mode of operation.

Further information: "Defining cycles", Page 77



For the various cycle parameters, the control provides selectable choices via the action bar or the form.

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 while executing the program.

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

NOTICE**Danger of collision!**

When running touch probe cycles **444** and **14xx**, no NC functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following simple transformations before using the touch probe cycle:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations before the cycle call.

Notes about machine parameters

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

Notes in connection with programming and execution

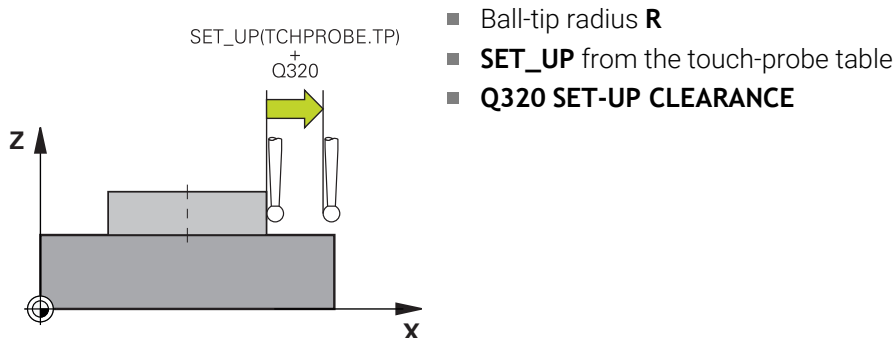
- Please note that the units of measure in the measuring log and in return parameters depend on the setting in the main program.
- The touch probe cycles **40x** to **43x** will reset an active basic rotation at the beginning of the cycle.
- The control interprets a basic transformation as a basic rotation,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

Pre-positioning

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

Pre-positioning is done in the inverse probing direction.

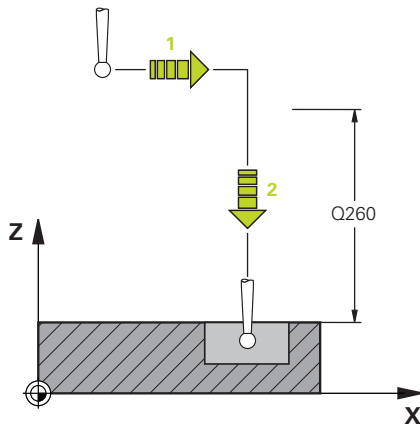
The distance between the probing point and the pre-position results from the following values:



Positioning logic

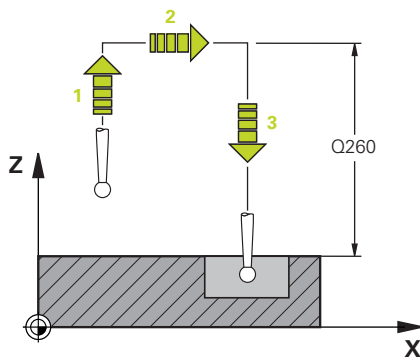
Touch-probe cycles with numbers from **400** through **499** or **1400** through **1499** pre-position the touch probe according to the following positioning logic:

Current position > Q260 CLEARANCE HEIGHT



- 1 The control positions the touch probe at **FMAX** at the pre-position in the working plane.
Further information: "Pre-positioning", Page 87
- 2 Then, the control positions the touch probe at **FMAX** in the tool axis, directly at probing height.

Current position < Q260 CLEARANCE HEIGHT



- 1 The control positions the touch probe at **FMAX** at **Q260 CLEARANCE HEIGHT**.
- 2 The control positions the touch probe at **FMAX** to the pre-position in the working plane.
Further information: "Pre-positioning", Page 87
- 3 Then, the control positions the touch probe at **FMAX** in the tool axis, directly to the probing height.

5.1.3 Machine-specific cycles

 Refer to your machine manual for a description of the specific functionality.

Cycles are available for many machines. Your machine manufacturer can implement these cycles into the control, in addition to the HEIDENHAIN cycles. These cycles are available in a separate cycle-number range:

Cycle-number range	Description
300 to 399	Machine-specific cycles that are to be selected through the CYCL DEF key
500 to 599	Machine-specific touch probe cycles that are to be selected through the TOUCH PROBE key

NOTICE

Danger of collision!

HEIDENHAIN cycles, machine manufacturer cycles and third-party functions use variables. You can also program variables within NC programs. Using variables outside the recommended ranges can lead to intersections, resulting in undesired behavior. Danger of collision during machining!

- ▶ Only use variable ranges recommended by HEIDENHAIN
- ▶ Do not use pre-assigned variables
- ▶ Comply with the documentation from HEIDENHAIN, the machine manufacturer and third-party providers
- ▶ Check the machining sequence using the simulation

Further information: "Calling cycles", Page 80
Further information: Programming and Testing User's Manual

5.1.4 Available cycle groups

Machining cycles

Cycle group	Further information
Drilling/Thread ■ Drilling, reaming ■ Boring ■ Counterboring, centering ■ Tapping ■ Thread milling	Further information: User's Manual for Machining Cycles
Pockets/studs/slots ■ Pocket milling ■ Stud milling ■ Slot milling ■ Face milling	Further information: User's Manual for Machining Cycles
Coordinate transformations ■ Mirroring ■ Rotating ■ Magnifying / Reducing	Further information: User's Manual for Machining Cycles
SL cycles ■ 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 Further information: User's Manual for Machining Cycles Further information: User's Manual for Machining Cycles
Point patterns ■ Bolt hole circle ■ Linear hole pattern ■ Data Matrix code	Further information: User's Manual for Machining Cycles
Special cycles ■ Dwell time ■ Oriented spindle stop ■ Tolerance ■ Ascertain the load ■ Program call ■ Engraving	Further information: User's Manual for Machining Cycles

Measuring cycles

Cycle group	Further information
Rotation	
■ Probing of plane, edge, two circles, beveled edge ■ Basic rotation ■ Two holes or studs ■ Via rotary axis ■ Via C-axis	Page 143
Preset/Position	
■ 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 212
Measuring	
■ Angle ■ Circle, inside or outside ■ Rectangle, inside or outside ■ Slot or ridge ■ Bolt hole circle ■ Plane or coordinate	Page 319
Special cycles	
■ Measuring or measuring in 3D ■ Probing in 3D ■ Fast probing ■ Extrusion probing	Page 374 Page 385
Calibrating the touch probe	
■ Calibrating the length ■ Calibration in a ring ■ Calibration on a stud ■ Calibration on a sphere	Page 100
Measuring kinematics	
■ Saving the kinematics ■ Measure kinematics ■ Preset compensation ■ Kinematics grid	Page 419
Measuring the tool (TT)	
■ Calibrating the TT ■ Tool length, radius or measuring completely ■ Calibrating the IR-TT	Page 395 Page 117

6

**Programming with
variables**

6.1 Program defaults for cycles

6.1.1 Overview

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

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

Cycle	Call	Further information
100 GENERAL Definition of generally valid cycle parameters ■ Q200 SET-UP CLEARANCE ■ Q204 2ND SET-UP CLEARANCE ■ Q253 F PRE-POSITIONING ■ Q208 RETRACTION FEED RATE	DEF-active	Page 96
120 PROBING Definition of specific touch probe cycle parameters ■ Q320 SET-UP CLEARANCE ■ Q260 CLEARANCE HEIGHT ■ Q301 MOVE TO CLEARANCE	DEF-active	Page 97

6.1.2 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

6.1.3 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. There is a danger of collision!

- ▶ 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**.

6.1.4 Global data valid everywhere

Parameters valid for all machining cycles **2xx** and the touch probe cycles **451, 452**

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 the tool and the 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

6.1.5 Global data for probing functions

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

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

Example

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

7

Touch probes

7.1 Calibrating a workpiece touch probe (#17 / #1-05-1)

7.1.1 Overview

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

Cycle	Call	Further information
460 CALIBRATION OF TS ON A SPHERE (#17 / #1-05-1) ■ Measuring the radius using a calibration sphere ■ Measuring the center offset using a calibration sphere	DEF-active	Page 102
461 TS CALIBRATION OF TOOL LENGTH (#17 / #1-05-1) ■ Calibrating the length	DEF-active	Page 109
462 CALIBRATION OF A TS IN A RING (#17 / #1-05-1) ■ Measuring the radius using a ring gauge ■ Measuring the center offset using a ring gauge	DEF-active	Page 111
463 TS CALIBRATION ON STUD (#17 / #1-05-1) ■ Measuring the radius using a stud or a calibration pin ■ Measuring the center offset using a stud or a calibration pin	DEF-active	Page 114

7.1.2 Fundamentals

Application



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

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

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



Always calibrate a touch probe in the following cases:

- 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 stylus tip. To calibrate the 3D touch probe, clamp a Calibration ring or a stud of known height and known radius to the machine table.

Calibrating a touch trigger probe

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

Always calibrate a touch probe in the following cases:

- 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 stylus tip. To calibrate the 3D touch probe, clamp a Calibration ring 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 applies 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

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.

7.1.3 Cycle 460 CALIBRATION OF TS ON A SPHERE (#17 / #1-05-1)

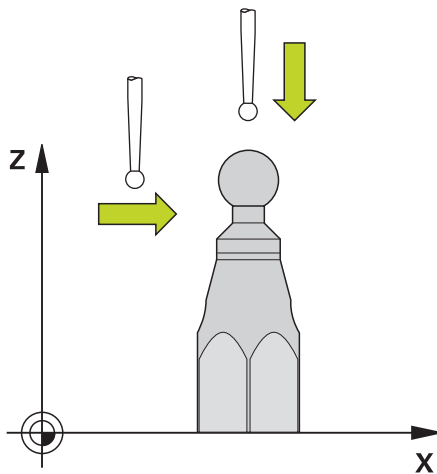
ISO programming

G460

Application



Refer to your machine manual.



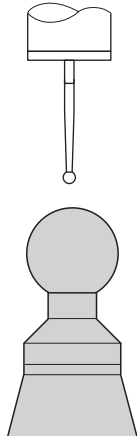
Before starting the calibration cycle, you need to pre-position the touch probe above the center of the calibration sphere. Position the touch probe in the touch probe axis by approximately the amount of the set-up clearance (value from touch probe table + value from cycle) above the calibration sphere.

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

Before calibrating with a normal stylus:

Before starting the calibration cycle, you need to pre-position the touch probe:

- ▶ Define the approximate value of the radius R and length L of the touch probe
- ▶ In the working plane, center the touch probe above the calibration sphere
- ▶ Position the touch probe in the touch probe axis by approximately the amount of the set-up clearance above the calibration sphere. The set-up clearance consists of the value from the touch probe table plus the value from the cycle.



Pre-positioning with a normal stylus

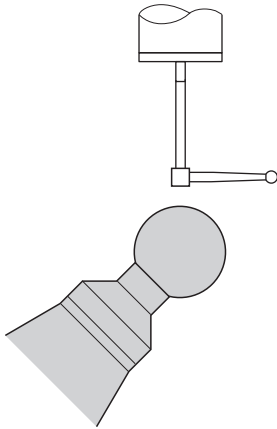
Before calibrating with an L-shaped stylus:

- ▶ Clamp the calibration sphere

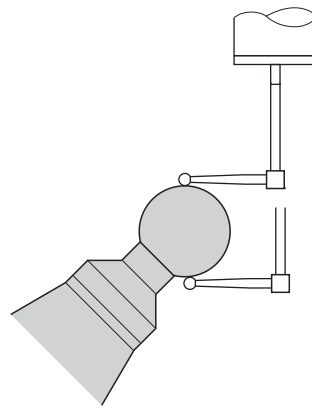


It must be possible to probe the north pole and south pole during calibration. If this is not possible, the control cannot determine the sphere radius. Ensure that no collision can occur.

- ▶ Define the approximate value of the radius **R** and length **L** of the touch probe. You can determine these with a tool presetter.
- ▶ Enter the approximate center offset in the touch probe table:
 - **CAL_OF1**: length of the extension
 - **CAL_OF2**: 0
- ▶ Insert the touch probe and orient it parallel to the main axis, for example with Cycle **13 ORIENTATION**
- ▶ Enter the calibration angle in the **CAL_ANG** column of the tool table.
- ▶ Position the center of the touch probe over the center of the calibration sphere
- ▶ Since the stylus is angled, the touch probe sphere is not centered over the calibration sphere.
- ▶ Position the touch probe in the tool axis by approximately the amount of the set-up clearance (value from touch probe table + value from cycle) above the calibration sphere

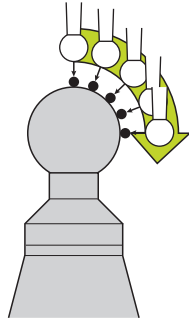


Pre-positioning with an L-shaped stylus



Calibration process with an L-shaped stylus

Cycle sequence



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

Radius calibration Q433=0

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

Radius and length calibration Q433=1

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

10 At the end of the cycle the control retracts the touch probe in the touch-probe axis to the height at which it had been pre-positioned.



- For an L-shaped stylus, the calibration takes place between the north and south pole.
- In order to calibrate the length, the position of the center point (**Q434**) of the calibration sphere relative to the active datum must be known. If this is not the case, then performing length calibration with Cycle **460** is not recommended!
- One application example for calibrating the length with Cycle **460** is the comparison of two touch probes

Notes



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

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

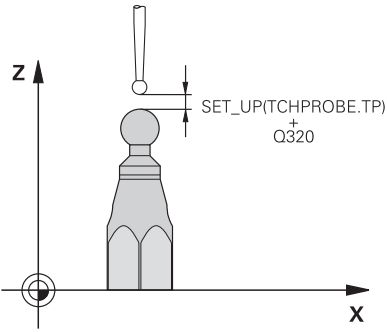
- This cycle can be executed only in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- A measuring log is created automatically during calibration. The log file is named **TCHPRAUTO.html**. This file is stored in the same location as the original file. The measuring log can be displayed in the browser on the control. If an NC program uses more than one cycle to calibrate the touch probe, **TCHPRAUTO.html** will contain all the measuring logs.
- The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose, the face of the spindle. The machine manufacturer may also place the tool reference point at a different point.
- Depending on the accuracy of the pre-positioning, finding the equator of the calibration sphere will require a different number of touch points.
- In order to achieve optimum accuracy results with an L-shaped stylus, HEIDENHAIN recommends calibrating and probing at identical speeds. Note the setting of the feed override if it is active for probing.
- If you program **Q455=0**, the control will not perform a 3D calibration.

- If you program **Q455=1** to **30**, the control will perform a 3D calibration of the touch probe. Deviations of the deflection behavior will thus be determined under various angles.
- If you program **Q455=1** to **30**, a table will be stored under TNC:\system\3D-ToolComp*.
- If there is already a reference to a calibration table (entry in **DR2TABLE**), this table will be overwritten.
- If there is no reference to a calibration table (entry in **DR2TABLE**), then, in dependence of the tool number, a reference and the associated table will be created.

Note on programming

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

Cycle parameters

Help graphic	Parameter
	Q407 Radius of calib. sphere? Enter the exact radius of the calibration sphere being used. Input: 0.0001...99.9999
	Q320 Set-up clearance? Additional distance between touch point and ball tip. Q320 is added to SET_UP (touch probe table), and is only active when the preset is probed in the touch probe axis. This value has an incremental effect. Input: 0...99999.9999 (or PREDEF)
	Q301 Move to clearance height (0/1)? Define how the touch probe will move between the measuring points: 0 : Move to measuring height between measuring points 1 : Move to clearance height between measuring points Input: 0, 1
	Q423 Number of probes? Number of measuring points on the diameter. This value has an absolute effect. Input: 3...8
	Q380 Ref. angle in ref. axis? Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. This value has an absolute effect. Input: 0...360
	Q433 Calibrate length (0/1)? Define whether the control will calibrate the touch probe length after radius calibration: 0 : Do not calibrate touch probe length 1 : Calibrate touch probe length Input: 0, 1

Help graphic	Parameter
	Q434 Preset for length? Coordinate of the calibration sphere center. This value must be defined only if length calibration will be carried out. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q455 No. of points for 3-D calibrtn.? (optional) Enter the number of touch points for 3D calibration. A value of about 15 touch points is useful. If you enter 0, the control will not perform a 3D calibration. During 3D calibration, the deflecting behavior of the touch probe is determined under various angles, and the values are stored in a table. 3D-ToolComp is required for 3D calibration. Input: 0...30

Example

11 TCH PROBE 460 TS CALIBRATION OF TS ON A SPHERE ~	
Q407=+12.5	;SPHERE RADIUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q301=+1	;TO CLEARANCE HEIGHT ~
Q423=+4	;NO. OF PROBE POINTS ~
Q380=+0	;REFERENCE ANGLE ~
Q433=+0	;DO NOT CALIBRATE THE LENGTH ~
Q434=-2.5	;PRESET ~
Q455=+15	;NO. POINTS 3-D CAL.

7.1.4 Cycle 461 TS CALIBRATION OF TOOL LENGTH (#17 / #1-05-1)

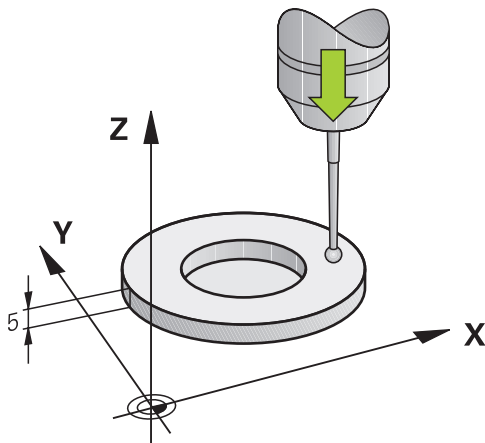
ISO programming

G461

Application



Refer to your machine manual.



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

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

Cycle sequence

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

Notes



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

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

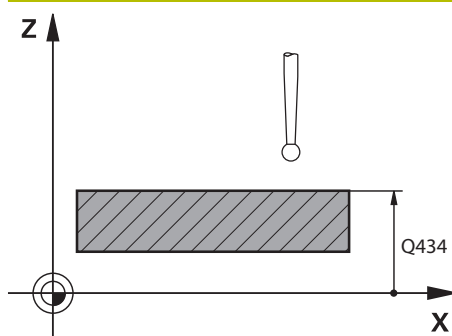
- This cycle can be executed only in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose, the face of the spindle. The machine manufacturer may also place the tool reference point at a different point.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q434 Preset for length?

Preset for the length (e.g. height of the Calibration ring). This value has an absolute effect.

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

Example

11 TCH PROBE 461 TS CALIBRATION OF TOOL LENGTH ~

Q434=+5

;PRESET

7.1.5 Cycle 462 CALIBRATION OF A TS IN A RING (#17 / #1-05-1)

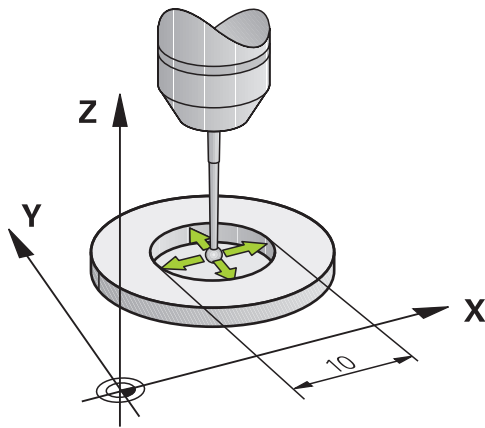
ISO programming

G462

Application



Refer to your machine manual.



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

When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run, the control finds the center point of the calibration ring or pin (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is determined. If the touch probe allows a reversal measurement, 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 reversal measurement determines not only the radius but also the center offset (**CAL_OF** in the touch-probe table).
- Any orientation possible (e.g., HEIDENHAIN infrared touch probes): Probing operation: see "Orientation possible in two directions").

Notes



In order to be able to determine the ball-tip center offset, the control needs to be specially prepared by the machine manufacturer.

The property of whether or how your touch probe can be oriented is predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine manufacturer.

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

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no functions for coordinate transformation must be active. Risk of collision!

► Do not activate the following cycles before the use of touch probe cycles:

- **TRANS DATUM**
- **TRANS MIRROR**
- **TRANS ROTATION**
- **TRANS SCALE**
- Cycle **8 MIRRORING**
- Cycle **10 ROTATION**
- Cycle **11 SCALING FACTOR**
- Cycle **26 AXIS-SPECIFIC SCALING**

► Reset any coordinate transformations beforehand.

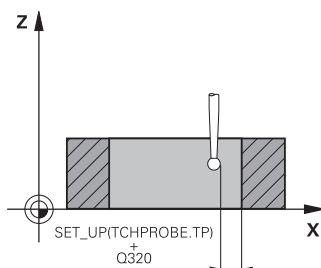
- This cycle can be executed only in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The center offset can be determined only with a suitable touch probe.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q407 Radius of ring gauge?

Enter the radius of the ring gauge.

Input: **0.0001...99.9999**

Q320 Set-up clearance?

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

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

Q423 Number of probes?

Number of measuring points on the diameter. This value has an absolute effect.

Input: **3...8**

Q380 Ref. angle in ref. axis?

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

Input: **0...360**

Example

11 TCH PROBE 462 CALIBRATION OF A TS IN A RING ~	
Q407=+5	;RING RADIUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q423=+8	;NO. OF PROBE POINTS ~
Q380=+0	;REFERENCE ANGLE

7.1.6 Cycle 463 TS CALIBRATION ON STUD (#17 / #1-05-1)

ISO programming

G463

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 stud (approximate measurement) and positions the touch probe in the center. Then, during the actual calibration process (fine measurement), the radius of the ball tip is determined. If the touch probe permits a reversal measurement, 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 reversal measurement determines now only the radius but also the center offset (CAL_OF in the touch-probe table).
- 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.

Whether or how your touch probe can be oriented is predefined for HEIDENHAIN touch probes. Other touch probes are configured by the machine manufacturer.

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

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

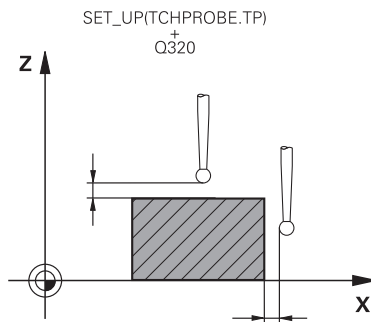
- This cycle can be executed only in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The center offset can be determined only with a suitable touch probe.
- A measuring log is created automatically during calibration. The log file is named TCHPRAUTO.html.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q407 Radius of calibr. stud?

Diameter of the Calibration ring

Input: **0.0001...99.9999**

Q320 Set-up clearance?

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

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

Q301 Move to clearance height (0/1)?

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

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Q423 Number of probes?

Number of measuring points on the diameter. This value has an absolute effect.

Input: **3...8**

Q380 Ref. angle in ref. axis?

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

Input: **0...360**

Example

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

7.2 Calibrating a tool touch probe (#17 / #1-05-1)

7.2.1 Overview

Cycle	Call	Further information
480 CALIBRATE TT (#17 / #1-05-1) ■ Calibrating the tool touch probe	DEF-active	Page 118
484 CALIBRATE IR TT (#17 / #1-05-1) ■ Calibrating the tool touch probe (e.g., infrared tool touch probe)	DEF-active	Page 120

7.2.2 Fundamentals

Application

The following cycles can be used to calibrate the tool touch probe or the infrared tool touch probe.

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 calibration pin or cylindrical pin. The control saves all determined calibration values and automatically considers them during all subsequent tool measurements.



As the calibration tool, HEIDENHAIN recommends using the calibration pin (**CAL_PIN**) tool type.

7.2.3 Cycle 480 CALIBRATE TT (#17 / #1-05-1)

ISO programming

G480

Application



Refer to your machine manual!

You calibrate the TT with touch probe cycle **480**. The calibration process runs automatically. The control also measures the center offset of the calibration tool automatically by rotating the spindle by 180° after the first half of the calibration cycle.

You calibrate the TT with touch probe cycle **480**.

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 be executed only in the **FUNCTION MODE MILL** machining mode.
- Before calibrating the touch probe, you must enter the exact length and radius of the calibration tool into the TOOL.T tool table.

Notes about machine parameters

- Use the machine parameter **CfgTTRoundStylus** (no. 114200) or **CfgTTRectStylus** (no. 114300) to define the functionality of the calibration cycle. Refer to your machine manual.
 - Use the machine parameter **centerPos** to define the position of the TT within the machine's working space.
- The TT needs to be recalibrated if you change the position of the TT on the table and/or a **centerPos** machine parameter.
- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual tooth measurements.

Cycle parameters

Help graphic	Parameter
	Q260 Clearance height? Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height 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)). Input: -99999.9999...+99999.9999

Example

11 TOOL CALL 12 Z	
12 TCH PROBE 480 CALIBRATE TT ~	
Q260=+100	;CLEARANCE HEIGHT

7.2.4 Cycle 484 CALIBRATE IR TT (#17 / #1-05-1)

ISO programming

G484

Application

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

- **With manual intervention:** If you define **Q536** = 0, then the control will stop before the calibration process. You then need to position the calibration tool manually above the center of the tool touch probe.
- **Without manual intervention:** If you define **Q536** = 1, then the control will automatically execute the cycle. You may have to program a 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.

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 run
- > If you have programmed **Q523 = 2**, the control writes the calibrated position to the machine parameter **centerPos** (no. 114200)

Q536 = 1: Without manual intervention before calibration

Proceed as follows:

- ▶ Insert the calibrating tool
- ▶ Position the calibration tool above the center of the tool touch probe before the start of the cycle.



- Ensure that the calibration tool is located above the measuring surface of the probe contact.
- For a calibration process without manual intervention, you do not need to position the calibration tool above the center of the tool touch probe. The cycle adopts the position from the machine parameters and automatically moves the tool to this position.

- ▶ Start the calibration cycle
- ▶ The calibration cycle is executed without stopping.
- ▶ If you have programmed **Q523 = 2**, then the control writes the calibrated position to the machine parameter **centerPos** (no. 114200).

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

If you program **Q536=1**, the tool must be pre-positioned before calling the cycle. The control also measures the center misalignment of the calibrating tool by rotating the spindle by 180° after the first half of the calibration cycle. There is a danger of collision!

- ▶ Specify whether to stop before cycle start or run the cycle automatically without stopping.

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The calibration tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck. If you use a cylinder pin of these dimensions, the resulting deformation will only be 0.1 µm per 1 N of probing force. Major inaccuracies may occur if you use a calibration tool whose diameter is too small and/or that protrudes too far from the chuck.
- Before calibrating the touch probe, you must enter the exact length and radius of the calibration tool into the TOOL.T tool table.
- The TT needs to be recalibrated if you change its position on the table.

Note regarding machine parameters

- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual tooth measurements.

Cycle parameters

Help graphic	Parameter
	Q536 Stop before running (0=Stop)? Define whether the control will stop before the calibration process or whether the cycle will automatically be executed without a stop: 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. 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. Input: 0, 1
	Q523 Position of tool probe (0-2)? Position of the tool touch probe: 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. 1: Configured position of the tool touch probe. The control adopts the position from the machine parameter centerPos (no. 114201). You do not need to pre-position the tool. The calibration tool approaches the position automatically. 2: Current position of the calibration tool. See Q523 = 0. 0: The control additionally writes the determined position (where applicable) to the machine parameter centerPos (no. 114201) after calibration. Input: 0, 1, 2

Example

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

8

**Touch-probe cycles
for workpieces
(#17 / #1-05-1)**

8.1 Overview

Determining workpiece misalignment

Cycle	Call	Further information
400 BASIC ROTATION (#17 / #1-05-1) - Automatic measurement using two points - Compensation via basic rotation	DEF-active	Page 144
401 ROT OF 2 HOLES (#17 / #1-05-1) - Automatic measurement using two holes - Compensation via basic rotation	DEF-active	Page 148
402 ROT OF 2 STUDS (#17 / #1-05-1) - Automatic measurement using two studs - Compensation via basic rotation	DEF-active	Page 153
403 ROT IN ROTARY AXIS (#17 / #1-05-1) - Automatic measurement using two points - Compensation via rotary table rotation	DEF-active	Page 158
404 SET BASIC ROTATION (#17 / #1-05-1) Setting any basic rotation	DEF-active	Page 163
405 ROT IN C AXIS (#17 / #1-05-1) - Automatic alignment of an angular offset between a hole center and the positive Y axis - Compensation via rotary table rotation	DEF-active	Page 165
1410 PROBING ON EDGE (#17 / #1-05-1) - Automatic measurement using two points - Compensation via basic rotation or rotary table rotation	DEF-active	Page 170
1411 PROBING TWO CIRCLES (#17 / #1-05-1) - Automatic measurement using two holes or studs - Compensation via basic rotation or rotary table rotation	DEF-active	Page 177
1412 INCLINED EDGE PROBING (#17 / #1-05-1) - Automatic measurement using two points on an inclined edge - Compensation via basic rotation or rotary table rotation	DEF-active	Page 185
1416 INTERSECTION PROBING (#17 / #1-05-1) - Automatically determines the intersection with four touch points on two straight lines - Compensation via basic rotation or rotary table rotation	DEF-active	Page 192

Cycle	Call	Further information
1420 PROBING IN PLANE (#17 / #1-05-1) ■ Automatic measurement using three points ■ Compensation via basic rotation or rotary table rotation	DEF -active	Page 200

Determining a preset

Cycle	Call	Further information
408 SLOT CENTER PRESET (#17 / #1-05-1) ■ Measuring the width of an inside slot ■ Setting the slot center as preset	DEF -active	Page 214
409 RIDGE CENTER PRESET (#17 / #1-05-1) ■ Measuring the outside width of a ridge ■ Definition of ridge center as preset	DEF -active	Page 219
410 PRESET INSIDE RECTAN (#17 / #1-05-1) ■ Measuring the inside length and width of a rectangle ■ Setting the rectangle center as preset	DEF -active	Page 224
411 PRESET OUTS. RECTAN (#17 / #1-05-1) ■ Measuring the outside length and width of a rectangle ■ Setting the rectangle center as preset	DEF -active	Page 229
412 PRESET INSIDE CIRCLE (#17 / #1-05-1) ■ Measuring any four points on the inside of a circle ■ Setting the circle center as preset	DEF -active	Page 234
413 PRESET OUTS. CIRCLE (#17 / #1-05-1) ■ Measuring any four points on the outside of a circle ■ Setting the circle center as preset	DEF -active	Page 240
414 PRESET OUTS. CORNER (#17 / #1-05-1) ■ Measuring two straight lines on the outside ■ Setting the intersection of the lines as preset	DEF -active	Page 246
415 PRESET INSIDE CORNER (#17 / #1-05-1) ■ Measuring two straight lines on the inside ■ Setting the intersection of the lines as preset	DEF -active	Page 253
416 PRESET CIRCLE CENTER (#17 / #1-05-1) ■ Measuring any three holes of a bolt hole circle ■ Definition of the circle center as preset	DEF -active	Page 259
417 PRESET IN TS AXIS (#17 / #1-05-1) ■ Measuring the any position in the tool axis ■ Setting any position as preset	DEF -active	Page 265

Cycle	Call	Further information
418 PRESET FROM 4 HOLES (#17 / #1-05-1) ■ Measuring two holes on each line crosswise ■ Setting the intersection of the connecting lines as preset	DEF -active	Page 269
419 PRESET IN ONE AXIS (#17 / #1-05-1) ■ Measuring any position in a selectable axis ■ Definition of any position in a selectable axis as preset	DEF -active	Page 274
1400 POSITION PROBING (#17 / #1-05-1) ■ Measuring the single position ■ Setting as preset, if applicable	DEF -active	Page 278
1401 CIRCLE PROBING (#17 / #1-05-1) ■ Measuring points on the inside or outside of a circle ■ Setting the circle center as preset, if applicable	DEF -active	Page 282
1402 SPHERE PROBING (#17 / #1-05-1) ■ Measuring points on a sphere ■ Definition of sphere center as preset, if necessary	DEF -active	Page 287
1403 RECTANGLE PROBING (#17 / #1-05-1) ■ Determining the center and lengths of a rectangle	DEF -active	Page 292
1404 PROBE SLOT/RIDGE (#17 / #1-05-1) ■ Determine the center of a slot width or ridge width ■ Set the center as a preset if needed	DEF -active	Page 298
1430 PROBE POSITION OF UNDERCUT (#17 / #1-05-1) ■ Measuring the undercut ■ Measure individual position with L-shaped stylus ■ Set the preset if needed	DEF -active	Page 304
1434 PROBE SLOT/RIDGE UNDERCUT (#17 / #1-05-1) ■ Measuring the undercut ■ Measuring the center of the slot width or ridge width with an L-shaped stylus ■ Set the center as a preset if needed	DEF -active	Page 310

Inspecting the workpiece

Cycle	Call	Further information
0 REF. PLANE (#17 / #1-05-1) ■ Measuring a coordinate in a selectable axis	DEF -active	Page 323
1 POLAR PRESET (#17 / #1-05-1) ■ Measuring a point ■ Probing direction via angle	DEF -active	Page 325

Cycle	Call	Further information
420 MEASURE ANGLE (#17 / #1-05-1) ■ Measuring an angle in the working plane	DEF-active	Page 327
421 MEASURE HOLE (#17 / #1-05-1) ■ Measuring the position of a hole ■ Measuring the diameter of a hole ■ Nominal-to-actual value comparison, if applicable	DEF-active	Page 330
422 MEAS. CIRCLE OUTSIDE (#17 / #1-05-1) ■ Measuring the position of a circular stud ■ Measuring the diameter of a circular stud ■ Nominal-to-actual value comparison, if applicable	DEF-active	Page 335
423 MEAS. RECTAN. INSIDE (#17 / #1-05-1) ■ 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 340
424 MEAS. RECTAN. OUTS. (#17 / #1-05-1) ■ 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 345
425 MEASURE INSIDE WIDTH (#17 / #1-05-1) ■ Measuring the position of a slot ■ Measuring the width of a slot ■ Nominal-to-actual value comparison, if applicable	DEF-active	Page 350
426 MEASURE RIDGE WIDTH (#17 / #1-05-1) ■ Measuring the position of a ridge ■ Measuring the width of a ridge ■ Nominal-to-actual value comparison, if applicable	DEF-active	Page 354
427 MEASURE COORDINATE (#17 / #1-05-1) ■ Measuring any coordinate in a selectable axis ■ Nominal-to-actual value comparison, if applicable	DEF-active	Page 358
430 MEAS. BOLT HOLE CIRC (#17 / #1-05-1) ■ 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 363

Cycle	Call	Further information
431 MEASURE PLANE (#17 / #1-05-1) ■ Finding the angle of a plane by measuring three points	DEF -active	Page 367

Probing the position in the plane or in space

Cycle	Call	Further information
3 MEASURING (#17 / #1-05-1) ■ Touch probe cycle for defining OEM cycles	DEF-active	Page 374
4 MEASURING IN 3-D (#17 / #1-05-1) ■ Measuring any position	DEF-active	Page 377
444 PROBING IN 3-D (#17 / #1-05-1) ■ Measuring any position ■ Determining the deviation from the nominal coordinates	DEF-active	Page 379

Influencing cycle runs

Cycle	Call	Further information
441 FAST PROBING (#17 / #1-05-1) ■ Touch probe cycle for defining various touch probe parameters	DEF-active	Page 385
1493 EXTRUSION PROBING (#17 / #1-05-1) ■ Touch probe cycle for defining an extrusion ■ Extrusion direction, length, and number of extrusion points can be programmed	DEF-active	Page 388

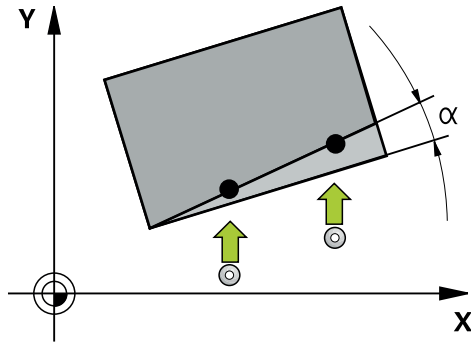
8.2 Conditional stops in touch probe cycles

If your machine has an override controller, you can activate conditional stops during program run. If you activate conditional stops with the **In cycle call** selection, the control does not interrupt program run during any touch probe cycles.

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

8.3 Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)

8.3.1 Application



The touch probe cycles contain the following:

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

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

8.3.2 Evaluation

Measurement results in Q parameters

The control saves the measurement results of the respective touch probe 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.

Notes on presets

- If you want to probe objects in a consistent machining plane or probe simple 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.

8.3.3 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.

8.3.4 Notes

- The probing positions are based on the programmed nominal positions in the **I-CS** input coordinate system.
- See your drawing for the nominal positions.
- Before defining a cycle, you must program a tool call in order to define the touch-probe axis.
- The **14xx** touch probe cycles support **SIMPLE** and **L-TYPE** styli.
- In order to achieve optimal accuracy results with a touch probe, HEIDENHAIN recommends that you perform probing and calibration at the same speed. Note the setting of the feed override if it is active during probing.
- If the workpiece touch probe does not deflect exactly horizontally or vertically, measuring results may deviate.
- If you want to use not only the measured rotation, but also a measured position, make sure to probe perpendicularly to the surface, if possible. The larger the angular error and the larger the ball-tip radius, the larger the positioning error. If the angular errors in the initial angular position are too large, corresponding position errors might be the result.
- If you use the touch probe cycles to correct the offset of a rotary axis, the control adds the values to the current value. Corrections can lead to values outside of the modulo range -360° to $+360^\circ$. If a rotary axis already has an offset outside of the modulo range, you can reduce the value with **PRESET CORR** and the entry **0** in the modulo range.

8.3.5 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.

You activate semi-automatic mode by entering a question mark ("?") before the nominal position in the cycle parameter. 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:

- **Further information:** "Alignment using two holes", Page 135
- **Further information:** "Alignment through an edge", Page 136
- **Further information:** "Alignment via the plane", Page 137

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 by 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**, then the control opens a message in the **FN 16** tab, **Status** workspace. This message indicates that the mode for retraction to the clearance height is not possible.



- ▶ Move the touch probe 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, manually traverse to a clearance height after every probing operation.



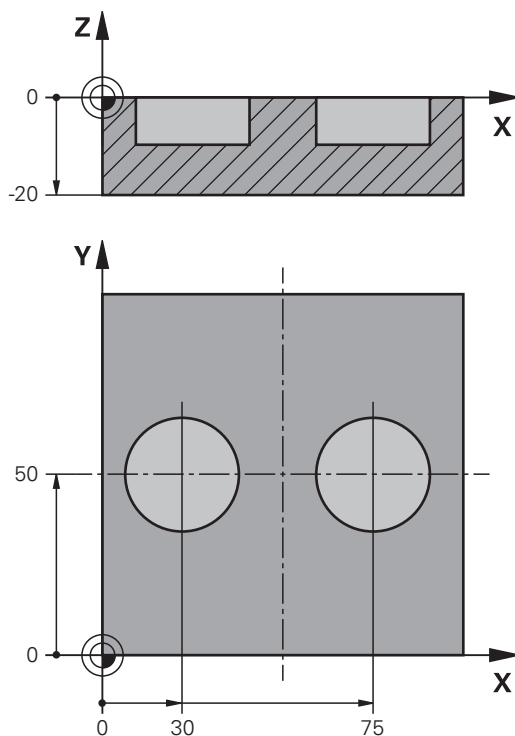
Programming and operating notes:

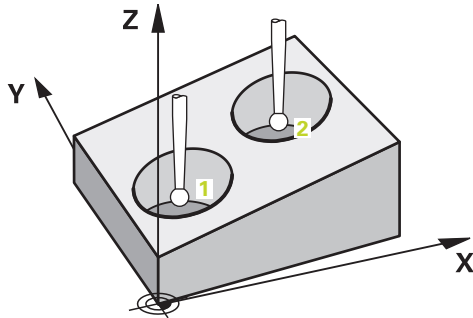
- See the drawing for these nominal positions.
- Semi-automatic mode is only executed in the machine operating modes, not in the simulation.
- If you did not define a nominal position for a touch point in any direction, the control generates an error message.
- If you did not define a nominal position 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.

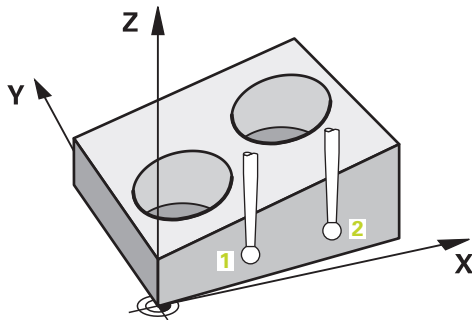


Alignment using two holes

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 of the main axis is provided, but the workpiece position is not known
- **QS1101** = Nominal Position 1 of the secondary axis is provided, but the workpiece position is not known
- **QS1102** = Nominal Position 1 in tool axis is unknown
- **QS1103** = Nominal Position 2 of the main axis is provided, but the workpiece position is not known
- **QS1104** = Nominal Position 2 of the secondary axis is provided, but the workpiece position is not known
- **QS1105** = Nominal Position 2 in tool axis is unknown

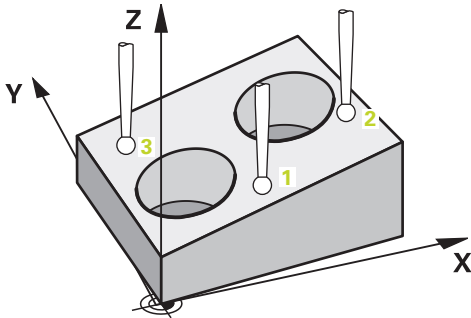
11 TCH PROBE 1411 PROBING TWO CIRCLES ~	
QS1100= "?30"	;1ST POINT REF AXIS ~
QS1101= "?50"	;1ST POINT MINOR AXIS ~
QS1102= "?"	;1ST POINT TOOL AXIS ~
Q1116=+10	;DIAMETER 1 ~
QS1103= "?75"	;2ND POINT REF AXIS ~
QS1104= "?50"	;2ND POINT MINOR AXIS ~
QS1105= "?"	;2ND POINT TOOL AXIS ~
Q1117=+10	;DIAMETER 2 ~
Q1115=+0	;1=HOLE - 2=HOLE ~
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	;BEFORE AND AFTER PROBING PROCESS ~
Q309=+0	;NO ERROR REACTION ~
Q1126=+0	;DO NOT ALIGN ~
Q1120=+0	;TRANSFER POSITION ~
Q1121=+0	;DO NOT APPLY 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 of the secondary axis is provided, but the workpiece position is not known
- **QS1102** = Nominal Position 1 in tool axis is unknown
- **QS1103** = Nominal Position 2 in main axis is unknown
- **QS1104** = Nominal Position 2 of the secondary axis is provided, but the workpiece position is not known
- **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	;MINOR AXIS PLUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q1125=+2	;BEFORE AND AFTER PROBING PROCESS ~
Q309=+0	;NO ERROR REACTION ~
Q1126=+0	;DO NOT ALIGN ~
Q1120=+0	;TRANSFER POSITION ~
Q1121=+0	;DO NOT APPLY 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 of the main axis is provided, but the workpiece position is not known
- **QS1101** = Nominal Position 1 of the secondary axis is provided, but the workpiece position is not known
- **QS1102** = Nominal Position 1 of the tool axis is provided, but the workpiece position is not known
- **QS1103** = Nominal Position 2 of the main axis is provided, but the workpiece position is not known
- **QS1104** = Nominal Position 2 of the secondary axis is provided, but the workpiece position is not known
- **QS1105** = Nominal Position 2 of the tool axis is provided, but the workpiece position is not known
- **QS1106** = Nominal Position 3 of the main axis is provided, but the workpiece position is not known
- **QS1107** = Nominal Position 3 of the secondary axis is provided, but the workpiece position is not known
- **QS1108** = Nominal Position 3 of the tool axis is provided, but the workpiece position is not known

11 TCH PROBE 1420 PROBING IN PLANE ~	
QS1100= "?50"	;1ST POINT REF AXIS ~
QS1101= "?10"	;1ST POINT MINOR AXIS ~
QS1102= "?0"	;1ST POINT TOOL AXIS ~
QS1103= "?80"	;2ND POINT REF AXIS ~
QS1104= "?50"	;2ND POINT MINOR AXIS ~
QS1105= "?0"	;2ND POINT TOOL AXIS ~
QS1106= "?20"	;3RD POINT REF AXIS ~
QS1107= "?80"	;3RD POINT MINOR AXIS ~
QS1108= "?0"	;3RD POINT TOOL AXIS ~
Q372=-3	;TOOL AXIS MINUS ~
Q320=+2	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q1125=+2	;BEFORE AND AFTER PROBING PROCESS ~
Q309=+0	;NO ERROR REACTION ~
Q1126=+0	;DO NOT ALIGN ~
Q1120=+0	;TRANSFER POSITION ~
Q1121=+0	;DO NOT APPLY ROTATION

8.3.6 Evaluation of tolerances

Cycles **14xx** also allow you to evaluate tolerance ranges. That way, you can evaluate the position and size of an object. Evaluation of the tolerance ranges enables you, for example, to increase the degree of automation by disabling the subsequent machining operation if the workpiece status is scrap.

You can define the following tolerances:

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

You can enter nominal dimensions with the following tolerances:

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

If you enter a value with a tolerance in your program, the control will monitor the tolerance band. The control writes the following statuses to the return parameter **Q183**: Pass, rework, or scrap. If a compensation for the preset is programmed, the control compensates for 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 a tolerance that does not comply with the DIN standard or if you indicate tolerances incorrectly when programming nominal dimensions (e.g., by entering blanks), the control aborts execution and displays an error message.
- Ensure correct upper and lower case when entering the DIN EN ISO and DIN ISO tolerances. Entering space characters is not allowed.

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 the NC program
- or
- Press **NC Start** to resume NC program run



Please note that the deviations returned by the touch probe cycles are based on the mean tolerance in **Q98x** and **Q99x**. If **Q1120** and **Q1121** are defined, then the values are equivalent to the values used for the compensation. If no automatic evaluation is active, then the control saves the values (based on the mean tolerance) in the intended Q parameter, allowing you to process these values.

Example

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

11 TCH PROBE 1411 PROBING 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 ~
Q1105=-5	;2ND POINT TOOL AXIS ~
QS1117="+8-2-1"	;DIAMETER 2 ~
Q1115=+0	;1=HOLE - 2=HOLE ~
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	;BEFORE AND AFTER PROBING PROCESS ~
Q309=2	;NO ERROR REACTION ~
Q1126=+0	;DO NOT ALIGN ~
Q1120=+0	;TRANSFER POSITION ~
Q1121=+0	;DO NOT APPLY ROTATION

8.3.7 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 the nominal position with tolerance monitoring as needed
- ▶ 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 based on its center and the position of a tooth

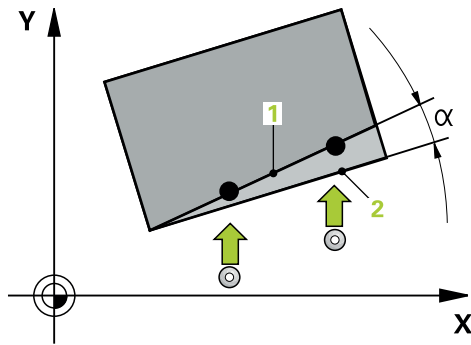
The nominal positions are defined here with tolerance monitoring and actual position.

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	;MINOR AXIS PLUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q1125=+2	;BEFORE AND AFTER PROBING PROCESS ~
Q309=+0	;NO ERROR REACTION ~
Q1126=+0	;DO NOT ALIGN ~
Q1120=+0	;TRANSFER POSITION ~
Q1121=+0	;DO NOT APPLY ROTATION

8.4 Determining workpiece misalignment (#17 / #1-05-1)

8.4.1 Fundamentals of touch probe cycles 400 to 405

Characteristics common to all touch probe cycles for measuring workpiece misalignment



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



These cycles do not work with 3D ROT! In such a case, use cycles **14xx**. **Further information:** "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 131

Note

- If you use the touch probe cycles to correct the offset of a rotary axis, the control adds the values to the current value. Corrections can lead to values outside of the modulo range -360° to $+360^\circ$. If a rotary axis already has an offset outside of the modulo range, you can reduce the value with **PRESET CORR** and the entry **0** in the modulo range.

8.4.2 Cycle 400 BASIC ROTATION (#17 / #1-05-1)

ISO programming

G400

Application

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



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

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

Related topics

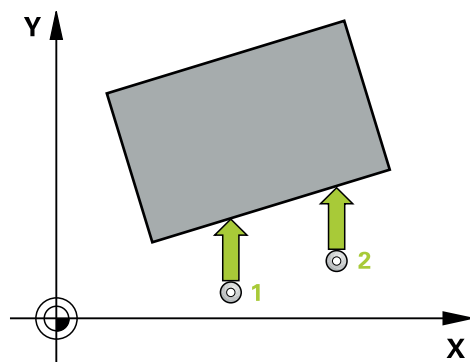
- Cycle **1410 PROBING ON EDGE**

Further information: "Cycle 1410 PROBING ON EDGE (#17 / #1-05-1)",
Page 170

- Cycle **1412 INCLINED EDGE PROBING**

Further information: "Cycle 1412 INCLINED EDGE PROBING (#17 / #1-05-1)",
Page 185

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 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 functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

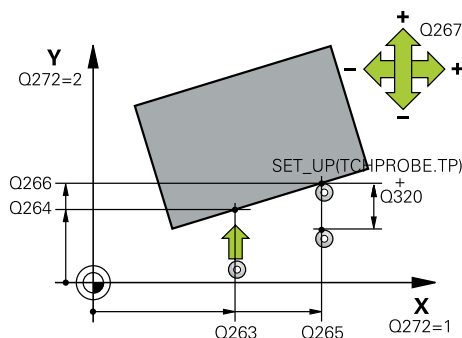
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

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

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

Q264 1st measuring point in 2nd axis?

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

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

Q265 2nd measuring point in 1st axis?

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

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

Q266 2nd measuring point in 2nd axis?

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

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

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

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

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

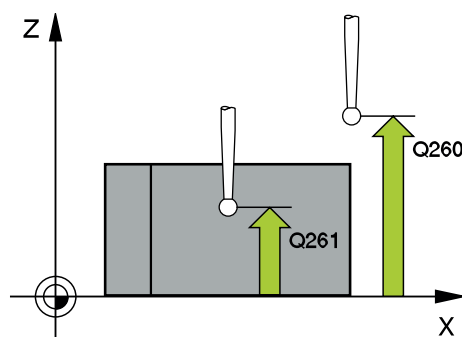
Input: **1, 2**

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

Direction in which the touch probe will approach the workpiece:

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

Input: **-1, +1**



Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

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

Example

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

8.4.3 Cycle 401 ROT OF 2 HOLES (#17 / #1-05-1)

ISO programming

G401

Application

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



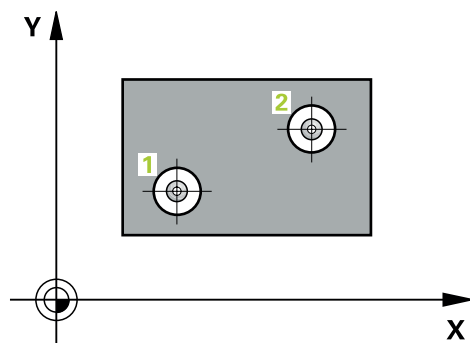
Instead of Cycle **401 ROT OF 2 HOLES**, HEIDENHAIN recommends using the more powerful Cycle **1411 PROBING TWO CIRCLES**.

Related topics

■ Cycle **1411 PROBING TWO CIRCLES**

Further information: "Cycle 1411 PROBING TWO CIRCLES (#17 / #1-05-1)", Page 177

Cycle sequence



- 1 The control positions the touch probe at the entered center of the first hole **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 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 calculated basic rotation.

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

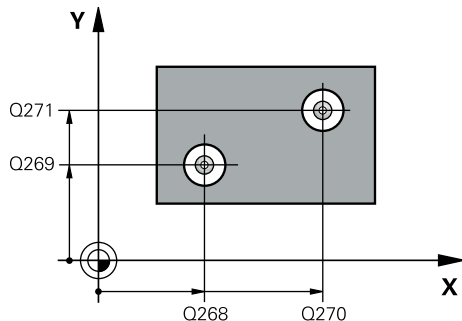
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.
- If you want to compensate for the misalignment by rotating the rotary table, the control will automatically use the following rotary axes: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

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q268 1st hole: center in 1st axis?

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

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

Q269 1st hole: center in 2nd axis?

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

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

Q270 2nd hole: center in 1st axis?

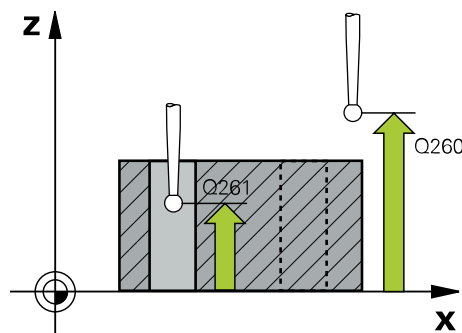
Center of the second hole in the main axis of the working plane. This value has an absolute effect.

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

Q271 2nd hole: center in 2nd axis?

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

Input: **-99999.9999...+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. This 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. This value has an absolute effect.

Input: **-99999.9999...+9999.9999** (or **PREDEF**)

Q307 Preset value for rotation angle

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

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

Help graphic

Parameter

Q305 Number in table? (optional)

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

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

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

Q305 depends on the following parameters: (optional)

- **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) (optional)

Define whether the control will set the determined misalignment as a basic rotation or will compensate for 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? (optional)

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	;SET BASIC ROTATION ~
Q337=+0	;DO NOT SET A PRESET

8.4.4 Cycle 402 ROT OF 2 STUDS (#17 / #1-05-1)

ISO programming

G402

Application

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

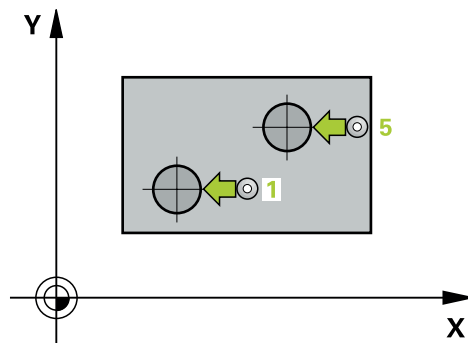
i Instead of Cycle **402 ROT OF 2 STUDS**, HEIDENHAIN recommends using the more powerful Cycle **1411 PROBING TWO CIRCLES**.

Related topics

■ Cycle **1411 PROBING TWO CIRCLES**

Further information: "Cycle 1411 PROBING TWO CIRCLES (#17 / #1-05-1)", Page 177

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 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 functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

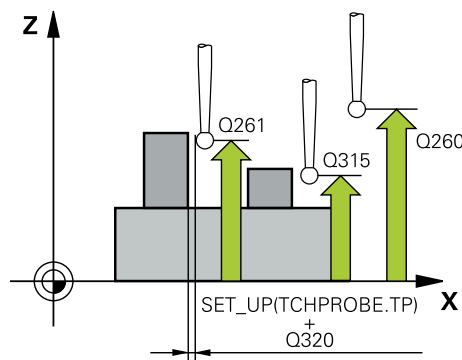
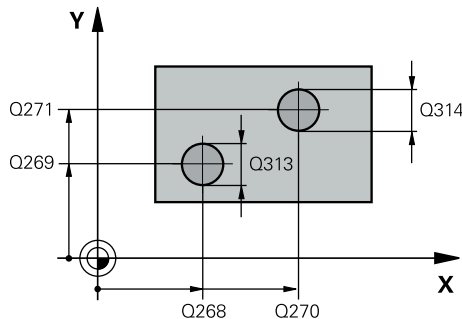
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.
- If you want to compensate for the misalignment by rotating the rotary table, the control will automatically use the following rotary axes: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

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q268 1st stud: center in 1st axis?

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

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

Q269 1st stud: center in 2nd axis?

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

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

Q313 Diameter of stud 1?

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

Input: **0...99999.9999**

Q261 Meas. height stud 1 in TS axis?

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

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

Q270 2nd stud: center in 1st axis?

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

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

Q271 2nd stud: center in 2nd axis?

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

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

Q314 Diameter of stud 2?

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

Input: **0...99999.9999**

Q315 Meas. height stud 2 in TS axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Help graphic

Parameter

Q301 Move to clearance height (0/1)?

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

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Q307 Preset value for rotation angle

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

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

Q305 Number in table? (optional)

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

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

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

Q305 depends on the following parameters: (optional)

- **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)** (optional)

Define whether the control will set the determined misalignment as a basic rotation or will compensate for 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? (optional)

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	;TO MEASURING HEIGHT ~
Q307=+0	;PRESET ROTATION ANG. ~
Q305=+0	;NUMBER IN TABLE ~
Q402=+0	;SET BASIC ROTATION ~
Q337=+0	;DO NOT SET A PRESET

8.4.5 Cycle 403 ROT IN ROTARY AXIS (#17 / #1-05-1)

ISO programming

G403

Application

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

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

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

Related topics

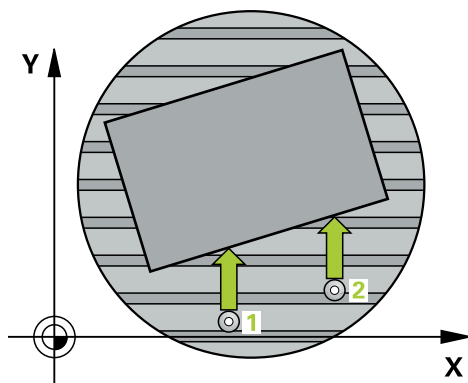
- Cycle **1410 PROBING ON EDGE**

Further information: "Cycle 1410 PROBING ON EDGE (#17 / #1-05-1)",
Page 170

- Cycle **1412 INCLINED EDGE PROBING**

Further information: "Cycle 1412 INCLINED EDGE PROBING (#17 / #1-05-1)",
Page 185

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 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.

- ▶ Pay attention to possible collisions between the tool and any elements attached to the table
- ▶ Use a sufficient clearance height so that no collisions can occur

NOTICE**Danger of collision!**

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

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

NOTICE**Danger of collision!**

When running touch probe cycles **400** to **499**, no functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

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

Help graphic



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

Q264 1st measuring point in 2nd axis?

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

Q265 2nd measuring point in 1st axis?

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

Q266 2nd measuring point in 2nd axis?

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

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

1: Main axis = measuring axis

2: Secondary axis = measuring axis

3: Touch probe axis = measuring axis

Input: **1, 2, 3**

Direction in which the touch probe will approach the workpiece:

-1: Negative traverse direction

+1: Positive traverse direction

Input: **-1, +1**



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

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

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

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

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

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

Help graphic

Parameter

Q301 Move to clearance height (0/1)?

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

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Q312 Axis for compensating movement?

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

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

4: Compensate for misalignment with rotary axis A

5: Compensate for misalignment with rotary axis B

6: Compensate for misalignment with rotary axis C

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

Q337 Set to zero after alignment? (optional)

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

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

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

Input: **0, 1**

Q305 Number in table? (optional)

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

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

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

Q305 depends on the following parameters:

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

Input: **0...99999**

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

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

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

1: Write the calculated preset to the preset table.

Input: **0, 1**

Q380 Ref. angle in ref. axis? (optional)

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

Input: **0...360**

Example

11 TCH PROBE 403 ROT IN ROTARY AXIS ~	
Q263=+0	;1ST POINT 1ST AXIS ~
Q264=+0	;1ST POINT 2ND AXIS ~
Q265=+20	;2ND PNT IN 1ST AXIS ~
Q266=+30	;2ND POINT 2ND AXIS ~
Q272=+1	;MAIN AXIS ~
Q267=-1	;NEGATIVE TRAVERSE DIRECTION ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;TO CLEARANCE HEIGHT ~
Q312=+0	;ROTARY-TABLE AXIS ~
Q337=+0	;DO NOT SET A PRESET ~
Q305=+1	;NUMBER IN TABLE ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q380=+90	;REFERENCE ANGLE

8.4.6 Cycle 404 SET BASIC ROTATION (#17 / #1-05-1)

ISO programming

G404

Application

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

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

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

Cycle parameters

Help graphic	Parameter
	Q307 Preset value for rotation angle Angle value at which the basic rotation will be set. Input: -360.000...+360.000
	Q305 Preset number in table? (optional) 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

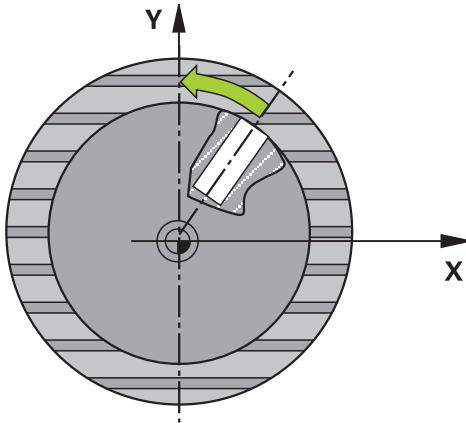
11 TCH PROBE 404 SET BASIC ROTATION ~	
Q307=+0	;PRESET ROTATION ANG. ~
Q305=-1	;NUMBER IN TABLE

8.4.7 Cycle 405 ROT IN C AXIS (#17 / #1-05-1)

ISO programming

G405

Application



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

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

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

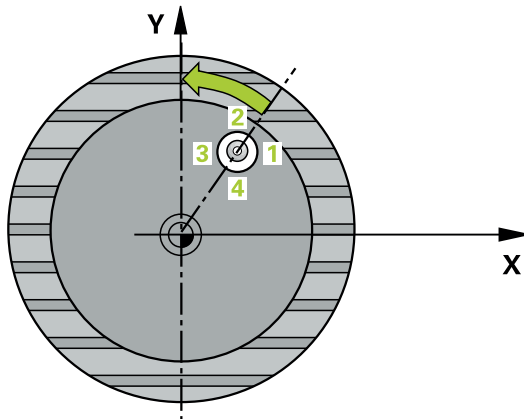


Instead of Cycle **405 ROT IN C AXIS**, HEIDENHAIN recommends using the more powerful Cycle **1411 PROBING TWO CIRCLES**.

Related topics

- Cycle **1411 PROBING TWO CIRCLES**

Further information: "Cycle 1411 PROBING TWO CIRCLES (#17 / #1-05-1)", Page 177

Cycle sequence

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the programmed starting angle.
- 3 Then, the touch probe moves along a circular arc, either at measuring height or at clearance height, to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times and then positions the touch probe on the calculated hole center.
- 5 Finally, the control returns the touch probe to the clearance height and aligns the workpiece by rotating the rotary table. The control rotates the rotary table in such a way that the hole center, after compensation, lies in the direction of the positive Y axis or at the nominal position of the hole center point—both with a vertical and a horizontal touch probe axis. The measured angular offset is also available in the parameter **Q150**.

Notes

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

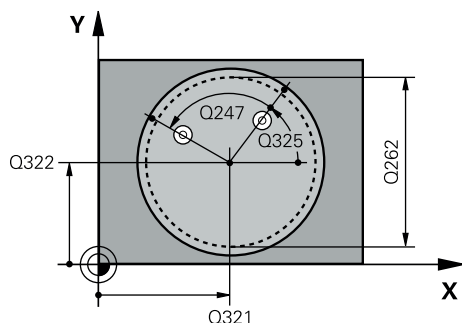
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

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

Cycle parameters

Help graphic



Parameter

Q321 Center in 1st axis?

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

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

Q322 Center in 2nd axis?

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

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

Q262 Nominal diameter?

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

Input: **0...99999.9999**

Q325 Starting angle?

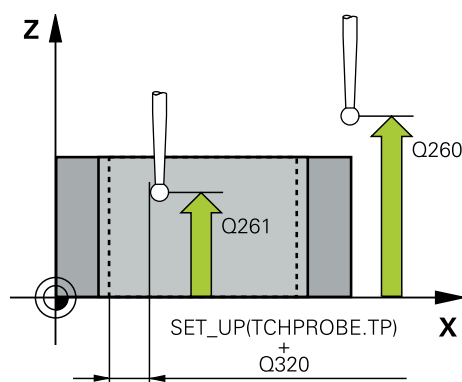
Angle between the main axis of the working plane and the first touch point. This value has an absolute effect.

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

Q247 Intermediate stepping angle?

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

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



Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

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

Example

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

8.4.8 Cycle 1410 PROBING ON EDGE (#17 / #1-05-1)

ISO programming

G1410

Application

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

Cycle **1493 EXTRUSION PROBING** allows you to repeat the touch points in a chosen direction and for a defined distance along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)", Page 388

The cycle also offers the following possibilities:

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

Further information: "Semi-automatic mode", Page 133

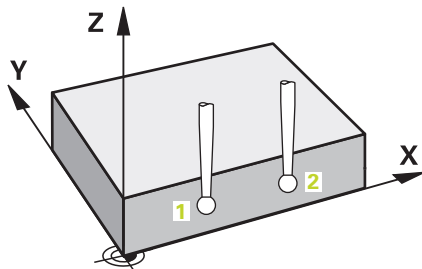
- Optionally, the cycle can monitor the tolerances. That way you can monitor the position and size of an object.

Further information: "Evaluation of tolerances", Page 139

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

Further information: "Transferring the actual position", Page 142

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 2 The touch probe then moves to the entered measuring height **Q1102** and performs the first probing procedure at probing speed **F** from the touch probe table.
- 3 The control offsets the touch probe by the amount of the set-up clearance in the direction opposite to the direction of probing.
- 4 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
- 5 The touch probe then moves to the next touch point **2** and probes again.
- 6 The control then positions the touch probe back to the clearance height (depending on **Q1125**) and stores the determined values in the following Q parameters:

Q parameter number	Meaning
Q950 to Q952	Measured position 1 in the main axis, secondary axis, and tool axis
Q953 to Q955	Measured position 2 in the main axis, secondary axis, and tool axis
Q964	Measured basic rotation
Q965	Measured table rotation
Q980 to Q982	Measured deviation from the first touch point
Q983 to Q985	Measured deviation from the second touch point
Q994	Measured angle deviation of basic rotation
Q995	Measured angle deviation of table rotation
Q183	Workpiece status ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not deflected The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle. Further information: "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 385
Q970	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation starting from the first touch point
Q971	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation starting from the second touch point

Notes

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

When running touch probe cycles **444** and **14xx**, no NC functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following simple transformations before using the touch probe cycle:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations before the cycle call.

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

- Observe the fundamentals of touch probe cycles **14xx**.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 131

Note about rotary axes:

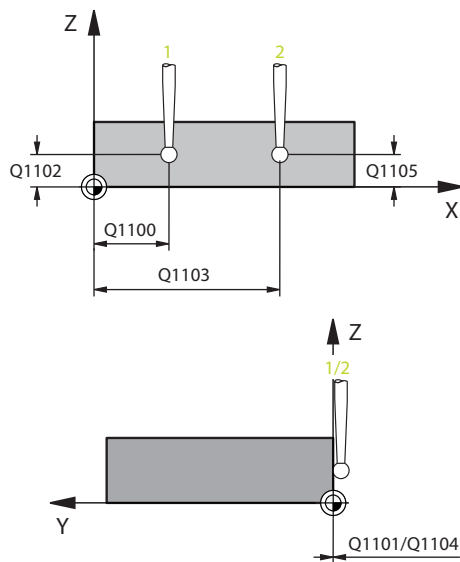
- If you determine the basic rotation in a tilted machining plane, then note the following:
 - If the current coordinates of the rotary axes and the defined tilting angle (3D-ROT menu) match, then the working plane is consistent. The control calculates the basic rotation in the input coordinate system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

Aligning the rotary table axes:

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

Further information: "Example: Determining a basic rotation from a plane and two holes", Page 208

Further information: "Example: Aligning the rotary table from two holes", Page 210

Cycle parameters**Help graphic****Parameter****Q1100 1st noml. position of ref. axis?**

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

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

- ? : Semi-automatic mode, see Page 133
- -, + : Evaluation of the tolerance, see Page 139
- @ : Transfer of an actual position, see Page 142

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 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 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

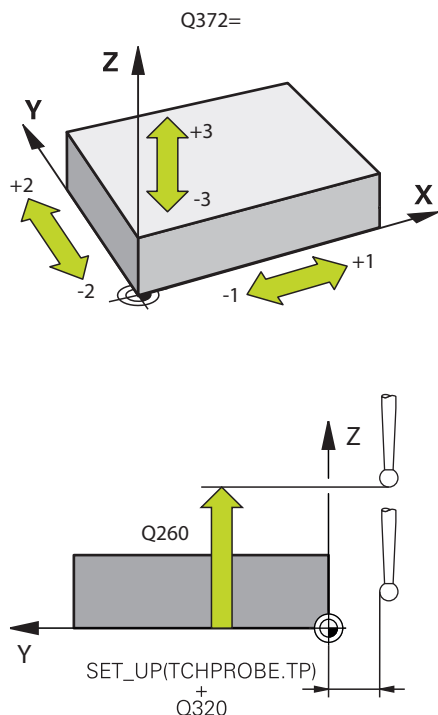
Input: -99999.9999...+9999.9999 Optional input (see **Q1100**)

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 Optional input (see **Q1100**)

Help graphic



Parameter

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** Optional input (see **Q1100**)

Q372 Probe direction (-3 to +3)?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

-1: Do not move to the clearance height.

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

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

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

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

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

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

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

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

Input: **0, 1, 2**

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

Example

11 TCH PROBE 1410 PROBING ON EDGE ~	
Q1100=+0	;1ST POINT REF AXIS ~
Q1101=+0	;1ST POINT MINOR AXIS ~
Q1102=+0	;1ST POINT TOOL AXIS ~
Q1103=+0	;2ND POINT REF AXIS ~
Q1104=+0	;2ND POINT MINOR AXIS ~
Q1105=+0	;2ND POINT TOOL AXIS ~
Q372=+1	;MAIN AXIS PLUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q1125=+2	;BEFORE AND AFTER PROBING PROCESS ~
Q309=+0	;NO ERROR REACTION ~
Q1126=+0	;DO NOT ALIGN ~
Q1120=+0	;TRANSFER POSITION ~
Q1121=+0	;DO NOT APPLY ROTATION

8.4.9 Cycle 1411 PROBING TWO CIRCLES (#17 / #1-05-1)

ISO programming

G1411

Application

Touch probe cycle **1411** captures the centers of two holes or cylindrical studs and calculates a straight line connecting these centers. The control calculates the rotation based on the nominal positions of the circle center points and the measured circle center points.

Cycle **1493 EXTRUSION PROBING** allows you to repeat the touch points in a chosen direction and for a defined distance along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)", Page 388

The cycle also offers the following possibilities:

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

Further information: "Semi-automatic mode", Page 133

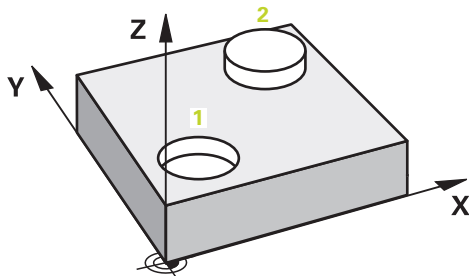
- Optionally, the cycle can monitor the tolerances. That way you can monitor the position and size of an object.

Further information: "Evaluation of tolerances", Page 139

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

Further information: "Transferring the actual position", Page 142

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch object **1** at **FMAX** (from the touch probe table), using positioning logic.

Further information: "Positioning logic", Page 88

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

Q parameter number	Meaning
Q950 to Q952	Measured circle center point 1 in the main axis, secondary axis, and tool axis
Q953 to Q955	Measured circle center point 2 in the main axis, secondary axis, and tool axis
Q964	Measured basic rotation
Q965	Measured table rotation
Q966 to Q967	Measured first and second diameters
Q980 to Q982	Measured deviation of the first circle center
Q983 to Q985	Measured deviation of the second center
Q994	Measured angle deviation of basic rotation
Q995	Measured angle deviation of table rotation
Q996 to Q997	Measured deviation of the diameters
Q183	Workpiece status ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not deflected The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle. Further information: "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 385
Q970	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation starting from the first circle center
Q971	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation starting from the second circle center
Q973	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation starting from Diameter 1
Q974	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation starting from Diameter 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, press **NC Start** to execute the cycle with the values from the dialog. The active set-up clearance is reduced to the displayed value only for this object.
 - You can cancel the cycle by pressing Cancel.

Notes

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

When running touch probe cycles **444** and **14xx**, no NC functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following simple transformations before using the touch probe cycle:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations before the cycle call.

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- Observe the fundamentals of touch probe cycles **14xx**.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 131

Note about rotary axes:

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

Aligning the rotary table axes:

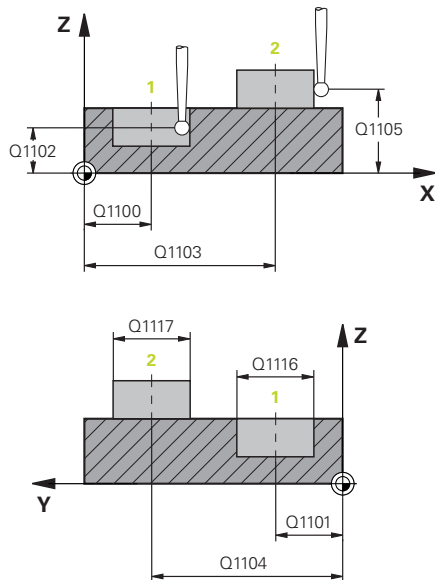
- The control can align the rotary table only if the measured rotation can be compensated for using a rotary table axis. This axis must be the first rotary table axis (as viewed from the workpiece).
- To align the rotary table axes (**Q1126** not equal to 0), you must apply the rotation (**Q1121** not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

Further information: "Example: Determining a basic rotation from a plane and two holes", Page 208

Further information: "Example: Aligning the rotary table from two holes", Page 210

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

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

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

- **"?...":** Semi-automatic mode, see Page 133
- **"...-...+...":** Evaluation of the tolerance, see Page 139
- **"...@...":** Transfer of an actual position, see Page 142

Q1101 1st noml. position of minor axis?

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

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

Q1116 Diameter of 1st position?

Diameter of the first hole or the first stud

Input: **0...9999.9999** Optional input:

- **"...-...+...":** Evaluation of the tolerance, see Page 139

Q1103 2nd noml. position of ref axis?

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

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

Q1104 2nd noml. position of minor axis?

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

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

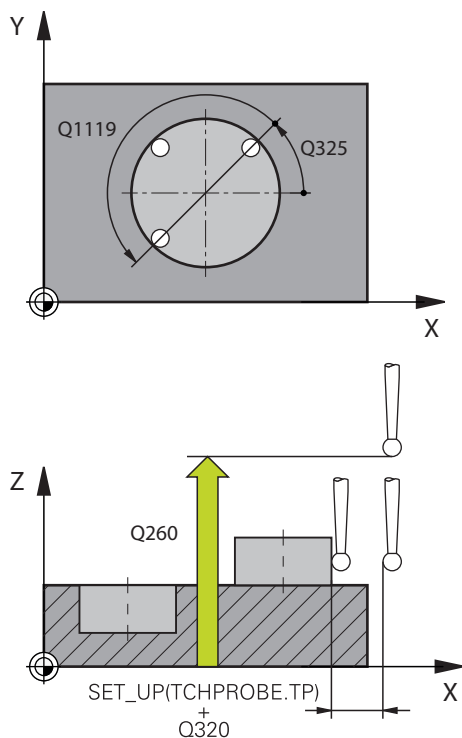
Q1117 Diameter of 2nd position?

Diameter of the second hole or the second stud

Input: **0...9999.9999** Optional input:

- **"...-...+...":** Evaluation of the tolerance, see Page 139

Help graphic



Parameter

Q1115 Geometry type (0-3)?

Type of object to be probed:

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

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

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

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

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

Q423 Number of probes?

Number of touch points on the diameter

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

Q325 Starting angle?

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

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

Q1119 Arc angular length?

Angular range in which the touch points are distributed.

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

-1: Do not move to the clearance height.

0: Move to the clearance height before and after the cycle.

Pre-positioning occurs at **FMAX_PROBE**.

1: Move to the clearance height before and after each object.

Pre-positioning occurs at **FMAX_PROBE**.

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

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

Help graphic	Parameter
	Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: 0: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. 1: Interrupt program run when tolerance is exceeded. The control opens a window with the results. 2: The control does not open a window if rework is necessary. The control opens a window with results and interrupts the program if the actual position is at scrap level. Input: 0, 1, 2
	Q1126 Align rotary axes? Position the rotary axes for inclined machining: 0: Retain the current position of the rotary axis. 1: Automatically position the rotary axis, and orient the tool tip (MOVE). The relative position between the workpiece and touch probe remains unchanged. The control performs a compensating movement with the linear axes. 2: Automatically position the rotary axis without orienting the tool tip (TURN). Input: 0, 1, 2
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: 0: No correction 1: Correction based on the 1st touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 1st touch point. 2: Correction based on the 2nd touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the 2nd touch point. 3: Correction based on the mean touch point. The control corrects the active preset by the amount of deviation between the nominal and actual position of the mean touch point. Input: 0, 1, 2, 3
	Q1121 CONFIRM ROTATION? Define whether the control should use the determined misalignment: 0: No basic rotation 1: Set the basic rotation: The control transfers the misalignment to the preset table as a basic transformation. 2: Rotate the rotary table: The control transfers the misalignment to the preset table as an offset. Input: 0, 1, 2

Example

11 TCH PROBE 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	;1=HOLE - 2=HOLE ~
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	;BEFORE AND AFTER PROBING PROCESS ~
Q309=+0	;NO ERROR REACTION ~
Q1126=+0	;DO NOT ALIGN ~
Q1120=+0	;TRANSFER POSITION ~
Q1121=+0	;CONFIRM ROTATION

8.4.10 Cycle 1412 INCLINED EDGE PROBING (#17 / #1-05-1)

ISO programming

G1412

Application

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

Cycle **1493 EXTRUSION PROBING** allows you to repeat the touch points in a chosen direction and for a defined distance along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)", Page 388

The cycle also offers the following possibilities:

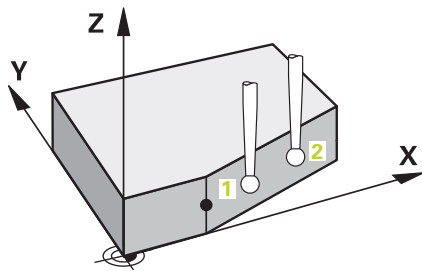
- If the coordinates of the touch points are not known, then you can execute the cycle in semi-automatic mode.

Further information: "Semi-automatic mode", Page 133

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

Further information: "Transferring the actual position", Page 142

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 2 The control then moves the touch probe to the entered measuring height **Q1102** and performs the first probing procedure at probing speed **F** from the touch probe table.
- 3 The control retracts the touch probe by the amount of the set-up clearance in the direction opposite to the direction of probing.
- 4 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
- 5 The touch probe then moves to the touch point **2** and probes again.
- 6 The control then positions the touch probe back to the clearance height (depending on **Q1125**) and stores the determined values in the following Q parameters:

Q parameter number	Meaning
Q950 to Q952	Measured position 1 in the main axis, secondary axis, and tool axis
Q953 to Q955	Measured position 2 in the main axis, secondary axis, and tool axis
Q964	Measured basic rotation
Q965	Measured table rotation
Q980 to Q982	Measured deviation from the first touch point
Q983 to Q985	Measured deviation from the second touch point
Q994	Measured angle deviation of basic rotation
Q995	Measured angle deviation of table rotation
Q183	Workpiece status ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not deflected The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle. Further information: "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 385
Q970	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation starting from the first touch point
Q971	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation starting from the second touch point

Notes

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

When running touch probe cycles **444** and **14xx**, no NC functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following simple transformations before using the touch probe cycle:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations before the cycle call.

- This cycle can be executed only 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 edges. Use the **TOLERANCE QS400** parameter to program a tolerance for the surface normal along the inclined edge.
- Observe the fundamentals of touch probe cycles **14xx**.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 131

Note about rotary axes:

- When you determine the basic rotation in a tilted working plane, keep the following in mind:
 - If the current coordinates of the rotary axes and the defined tilt angles (3D ROT menu) match, the working plane is consistent. The control calculates the basic rotation in the input coordinate system **I-CS**.
 - If the current coordinates of the rotary axes and the defined tilt angles (3D ROT menu) do not match, the working plane is inconsistent. The control calculates the basic rotation in the workpiece coordinate system **W-CS** in dependence on the tool axis.

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

Aligning the rotary table axes:

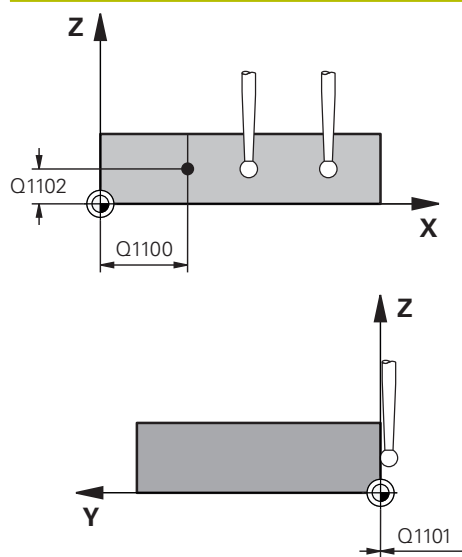
- The control can align the rotary table only if the measured rotation can be compensated for using a rotary table axis. This axis must be the first rotary table axis (as viewed from the workpiece).
- To align the rotary table axes (**Q1126** not equal to 0), you must apply the rotation (**Q1121** not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

Further information: "Example: Determining a basic rotation from a plane and two holes", Page 208

Further information: "Example: Aligning the rotary table from two holes", Page 210

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

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

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

- **?**: Semi-automatic mode, see Page 133
- **-, +**: Evaluation of the tolerance, see Page 139
- **@**: Transfer of an actual position, see Page 142

Q1101 1st noml. position of minor axis?

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

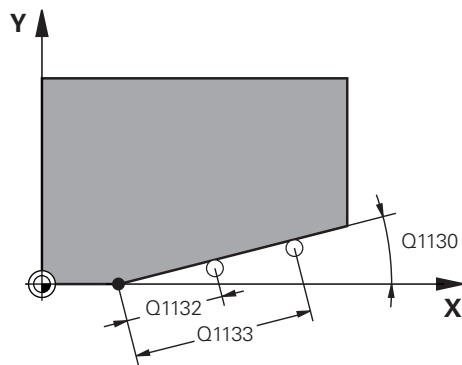
Input: **-99999.9999...+99999.9999** Optional input (see **Q1100**)

Q1102 1st nominal position tool axis?

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

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

Help graphic



Parameter

QS400 Tolerance value?

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

Examples:

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

Input: Max. **255** characters

Q1130 Nominal angle for 1st line?

Nominal angle of the first straight line

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

Q1131 Probing direction for 1st line?

Probing direction for the first edge:

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

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

Input: **-1, +1**

Q1132 First distance on 1st line?

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

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

Q1133 Second distance on 1st line?

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

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

Q1139 Plane for object (1-3)?

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

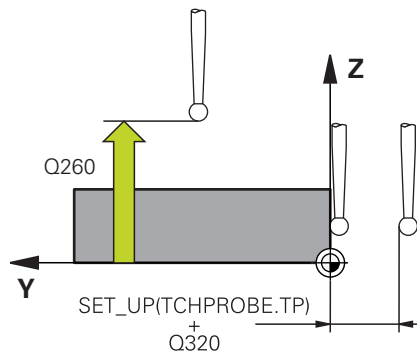
1: YZ plane

2: ZX plane

3: XY plane

Input: **1, 2, 3**

Help graphic



Parameter

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

-1: Do not move to the clearance height.

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

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

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

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

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

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

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

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

Input: **0, 1, 2**

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**

Help graphic**Parameter****Q1120 Transfer position?**

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

0: No correction

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

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

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

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

Q1121 CONFIRM ROTATION?

Define whether the control should use the determined misalignment:

0: No basic rotation

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

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

Input: **0, 1, 2**

Example

11 TCH PROBE 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	;+90° TO THE STRAIGHT LINE ~
Q1132=+10	;FIRST DISTANCE, 1ST LINE ~
Q1133=+20	;SECOND DISTANCE, 1ST LINE ~
Q1139=+3	;TOOL AXIS ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q1125=+2	;BEFORE AND AFTER PROBING PROCESS ~
Q309=+0	;NO ERROR REACTION ~
Q1126=+0	;DO NOT ALIGN ~
Q1120=+0	;TRANSFER POSITION ~
Q1121=+0	;DO NOT APPLY ROTATION

8.4.11 Cycle 1416 INTERSECTION PROBING (#17 / #1-05-1)

ISO programming

G1416

Application

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

Cycle **1493 EXTRUSION PROBING** allows you to repeat the touch points in a chosen direction and for a defined distance along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)", Page 388

The cycle also offers the following possibilities:

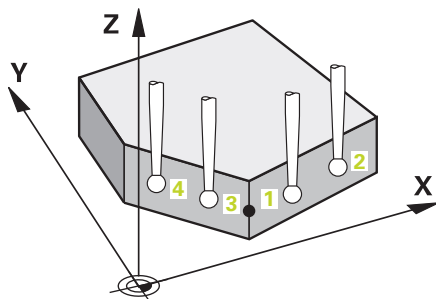
- If the coordinates of the touch points are not known, then you can execute the cycle in semi-automatic mode.

Further information: "Semi-automatic mode", Page 133

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

Further information: "Transferring the actual position", Page 142

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 2 The control then moves the touch probe to the entered measuring height **Q1102** and performs the first probing procedure at probing speed **F** from the touch probe table.
- 3 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
- 4 The control positions the touch probe to the next touch point.
- 5 The control positions the touch probe to the entered measuring height **Q1102** and measures the next touch point.
- 6 The control repeats Steps 3 to 5 until all four touch points are measured.
- 7 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSFER POSITION** is defined with the value **1**, then the control corrects the ascertained deviations in the active row of the preset table.

Q parameter number	Meaning
Q950 to Q952	Measured position 1 in the main axis, secondary axis and tool axis
Q953 to Q955	Measured position 2 in the main axis, secondary axis and tool axis
Q956 to Q958	Measured position 3 in the main axis, secondary axis and tool axis
Q959 to Q960	Measured intersection in the main axis and secondary axis
Q964	Measured basic rotation
Q965	Measured table rotation
Q980 to Q982	Measured deviation of the first touch point in the main axis, auxiliary axis and tool axis
Q983 to Q985	Measured deviation of the second touch point in the main axis, auxiliary axis and tool axis
Q986 to Q988	Measured deviation of the third touch point in the main axis, auxiliary axis and tool axis
Q989 to Q990	Measured deviations of the intersection in the main axis and secondary axis
Q994	Measured angle deviation of basic rotation
Q995	Measured angle deviation of table rotation
Q183	Workpiece status ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not deflected The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle. Further information: "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 385
Q970	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation from the 1st touch point
Q971	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation from the 2nd touch point
Q972	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation from the 3rd touch point

Notes

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

When running touch probe cycles **444** and **14xx**, no NC functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following simple transformations before using the touch probe cycle:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations before the cycle call.

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- Observe the fundamentals of touch probe cycles **14xx**.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 131

Note about rotary axes:

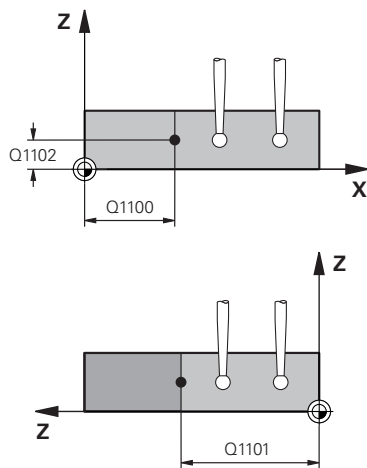
- If you determine the basic rotation in a tilted machining plane, then note the following:
 - If the current coordinates of the rotary axes and the defined tilting angle (3D-ROT menu) match, then the machining plane is consistent. The control calculates the basic rotation in the input coordinate system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

Aligning the rotary table axes:

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

Further information: "Example: Determining a basic rotation from a plane and two holes", Page 208

Further information: "Example: Aligning the rotary table from two holes", Page 210

Cycle parameters**Help graphic****Parameter****Q1100 1st noml. position of ref. axis?**

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

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

- **?**: Semi-automatic mode, see Page 133
- **@**: Transfer of an actual position, see Page 142

Q1101 1st noml. position of minor axis?

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

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

Q1102 1st nominal position tool axis?

Absolute nominal position of the touch points in the tool axis

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

QS400 Tolerance value?

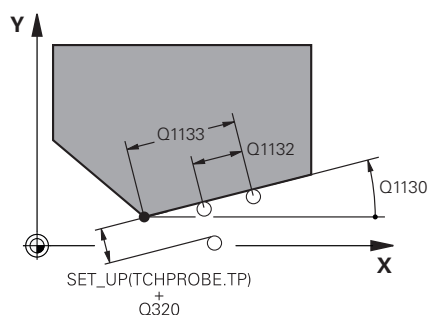
Tolerance band monitored by the cycle. The tolerance defines the permissible deviation of the surface normal along the first edge. The control determines the deviation using the nominal coordinates and the actual coordinates of the part.

Examples:

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

Input: Max. **255** characters

Help graphic



Parameter

Q1130 Nominal angle for 1st line?

Nominal angle of the first straight line

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

Q1131 Probing direction for 1st line?

Probing direction for the first edge:

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

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

Input: **-1, +1**

Q1132 First distance on 1st line?

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

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

Q1133 Second distance on 1st line?

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

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

QS401 Tolerance value 2?

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

Input: Max. **255** characters

Q1134 Nominal angle for 2nd line?

Nominal angle of the first straight line

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

Q1135 Probing direction for 2nd line?

Probing direction for the second edge:

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

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

Input: **-1, +1**

Q1136 First distance on 2nd line?

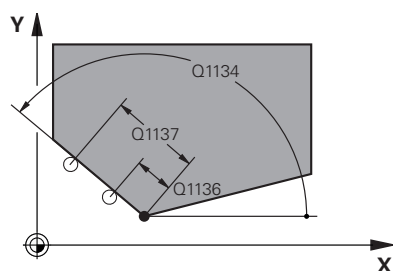
Distance between the intersection and the first touch point on the second edge. This value has an incremental effect.

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

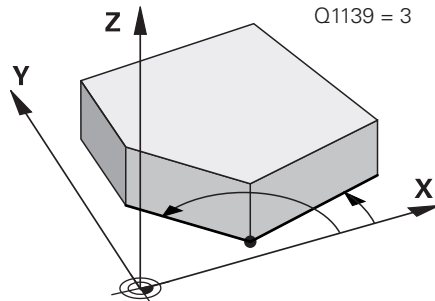
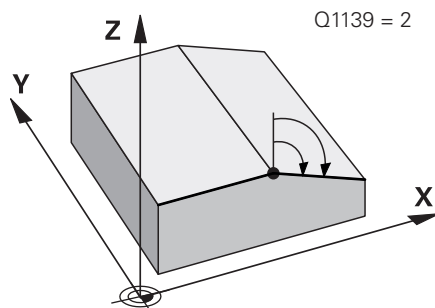
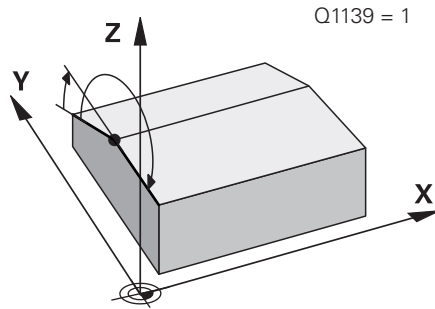
Q1137 Second distance on 2nd line?

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

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



Help graphic



Parameter

Q1139 Plane for object (1-3)?

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

- 1: YZ plane
- 2: ZX plane
- 3: XY plane

Input: **1, 2, 3**

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

- 1: Do not move to the clearance height.
- 0: Move to the clearance height before and after the cycle. Pre-positioning occurs at **FMAX_PROBE**.
- 1: Move to the clearance height before and after each object. Pre-positioning occurs at **FMAX_PROBE**.
- 2: Move to the clearance height before and after each touch point. Pre-positioning occurs at **FMAX_PROBE**.

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

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

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

Input: **0, 1, 2**

Help graphic

Parameter

Q1126 Align rotary axes?

Position the rotary axes for inclined machining:

0: Retain the current position of the rotary axis.

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

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

Input: **0, 1, 2**

Q1120 Transfer position?

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

0: No correction

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

Input: **0, 1**

Q1121 CONFIRM ROTATION?

Define whether the control should use the determined misalignment:

0: No basic rotation

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

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

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

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

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

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

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

Example

11 TCH PROBE 1416 INTERSECTION PROBING ~	
Q1100=+50	;1ST POINT REF AXIS ~
Q1101=+10	;1ST POINT MINOR AXIS ~
Q1102=-5	;1ST POINT TOOL AXIS ~
QS400="0"	;TOLERANCE ~
Q1130=+45	;NOMINAL ANGLE, 1ST LINE ~
Q1131=+1	;+90° TO THE STRAIGHT LINE ~
Q1132=+10	;FIRST DISTANCE, 1ST LINE ~
Q1133=+25	;SECOND DISTANCE, 1ST LINE ~
QS401="0"	;TOLERANCE 2 ~
Q1134=+135	;NOMINAL ANGLE, 2ND LINE ~
Q1135=-1	; -90° TO THE STRAIGHT LINE ~
Q1136=+10	;FIRST DISTANCE, 2ND LINE ~
Q1137=+25	;SECOND DISTANCE, 2ND LINE ~
Q1139=+3	;TOOL AXIS ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q1125=+2	;BEFORE AND AFTER PROBING PROCESS ~
Q309=+0	;NO ERROR REACTION ~
Q1126=+0	;DO NOT ALIGN ~
Q1120=+0	;TRANSFER POSITION ~
Q1121=+0	;CONFIRM ROTATION

8.4.12 Cycle 1420 PROBING IN PLANE (#17 / #1-05-1)

ISO programming

G1420

Application

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

Cycle **1493 EXTRUSION PROBING** allows you to repeat the touch points in a chosen direction and for a defined distance along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)", Page 388

The cycle also offers the following possibilities:

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

Further information: "Semi-automatic mode", Page 133

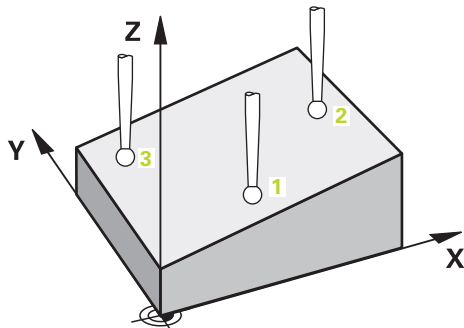
- Optionally, the cycle can monitor the tolerances. That way you can monitor the position and size of an object.

Further information: "Evaluation of tolerances", Page 139

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

Further information: "Transferring the actual position", Page 142

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 2 The touch probe then moves to the entered measuring height **Q1102** and performs the first probing procedure at probing speed **F** from the touch probe table.
- 3 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
- 4 It then moves in the working plane to touch point **2** to measure the actual value of the second touch point in the plane.
- 5 The touch probe returns to the clearance height (depending on **Q1125**), then moves in the working plane to touch point **3** and measures the actual position of the third point of the plane.
- 6 The control then positions the touch probe back to the clearance height (depending on **Q1125**) and stores the determined values in the following Q parameters:

Q parameter number	Meaning
Q950 to Q952	Measured position 1 in the main axis, secondary axis, and tool axis
Q953 to Q955	Measured position 2 in the main axis, secondary axis, and tool axis
Q956 to Q958	Measured position 3 in the main axis, secondary axis, and tool axis
Q961 to Q963	Measured spatial angle SPA, SPB, and SPC in the W-CS
Q980 to Q982	Measured deviation from the first touch point
Q983 to Q985	Measured deviation from the second touch point
Q986 to Q988	Third measured deviation of the positions
Q183	Workpiece status ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not deflected The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle. Further information: "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 385
Q970	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation starting from the first touch point
Q971	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation starting from the second touch point
Q972	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation starting from the third touch point

Notes

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

When running touch probe cycles **444** and **14xx**, no NC functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following simple transformations before using the touch probe cycle:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations before the cycle call.

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control can calculate the angle values only if the three touch points are not positioned on a straight line.
- The nominal spatial angle results from the defined nominal positions. The cycle saves the measured spatial angle in parameters **Q961** to **Q963**. For the transfer to the 3D basic rotation, the control uses the difference between the measured spatial angle and the nominal spatial angle.
- Observe the fundamentals of touch probe cycles **14xx**.
Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 131



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

Aligning the rotary table axes:

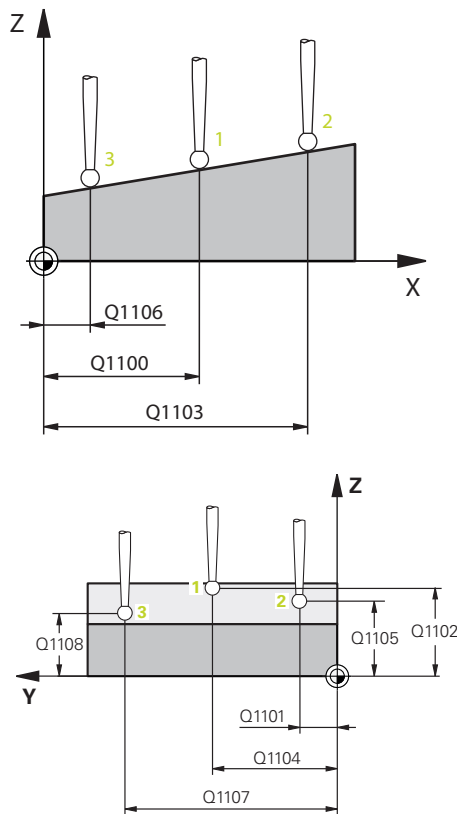
- Alignment of rotary axes is only possible if two rotary axes are available in the kinematics.
- To align the rotary axes (**Q1126** unequal to 0), the rotation must be accepted (**Q1121** unequal to 0). Otherwise, the control will display an error message.

Further information: "Example: Determining a basic rotation from a plane and two holes", Page 208

Further information: "Example: Aligning the rotary table from two holes", Page 210

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

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

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

- **?**: Semi-automatic mode, see Page 133
- **-, +**: Evaluation of the tolerance, see Page 139
- **@**: Transfer of an actual position, see Page 142

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** 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** 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** 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** 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** 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** 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** 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** Optional input (see **Q1100**)

Q372 Probe direction (-3 to +3)?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

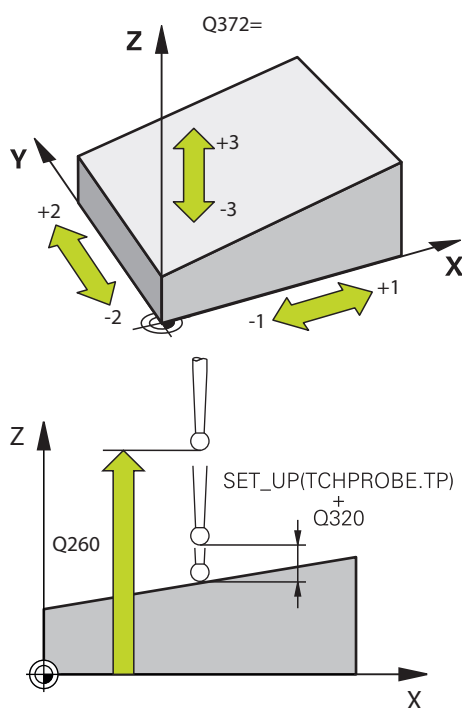
-1: Do not move to the clearance height.

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

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

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

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

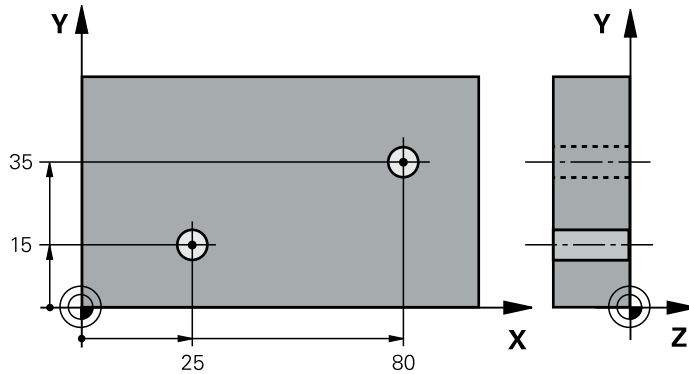


Help graphic	Parameter
	Q309 Reaction to tolerance error? Reaction when tolerance is exceeded: 0: Do not interrupt program run when tolerance is exceeded. The control does not open a window with the results. 1: Interrupt program run when tolerance is exceeded. The control opens a window with the results.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

Example

11 TCH PROBE 1420 PROBING IN PLANE ~	
Q1100=+0	;1ST POINT REF AXIS ~
Q1101=+0	;1ST POINT MINOR AXIS ~
Q1102=+0	;1ST POINT TOOL AXIS ~
Q1103=+0	;2ND POINT REF AXIS ~
Q1104=+0	;2ND POINT MINOR AXIS ~
Q1105=+0	;2ND POINT TOOL AXIS ~
Q1106=+0	;3RD POINT REF AXIS ~
Q1107=+0	;3RD POINT MINOR AXIS ~
Q1108=+0	;3RD POINT TOOL AXIS ~
Q372=+1	;MAIN AXIS PLUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+100	;CLEARANCE HEIGHT ~
Q1125=+2	;BEFORE AND AFTER PROBING PROCESS ~
Q309=+0	;NO ERROR REACTION ~
Q1126=+0	;DO NOT ALIGN ~
Q1120=+0	;TRANSFER POSITION ~
Q1121=+0	;DO NOT APPLY ROTATION

8.4.13 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

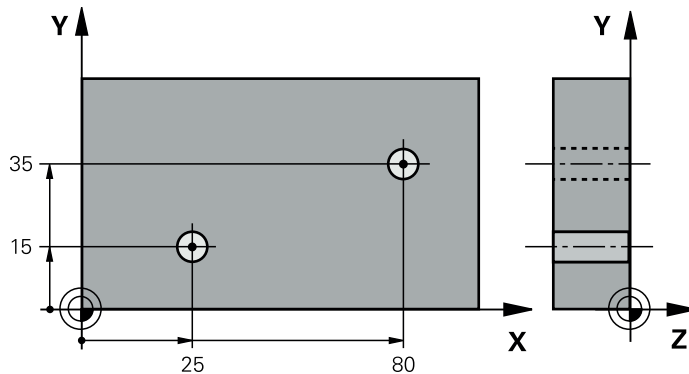
0 BEGIN PGM TOUCHPROBE MM	
1 TOOL CALL 600 Z	
2 TCH PROBE 401 ROT OF 2 HOLES ~	
Q268=+25 ;1ST CENTER 1ST AXIS ~	
Q269=+15 ;1ST CENTER 2ND AXIS ~	
Q270=+80 ;2ND CENTER 1ST AXIS ~	
Q271=+35 ;2ND CENTER 2ND AXIS ~	
Q261=-5 ;MEASURING HEIGHT ~	
Q260=+20 ;CLEARANCE HEIGHT ~	
Q307=+0 ;PRESET ROTATION ANG.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

8.4.14 Example: Determining a basic rotation from a plane and two holes

When setting a basic rotation with cycles **14xx**, this must be defined by parameters **Q1120 TRANSFER POSITION** and **Q1121 CONFIRM ROTATION**.

Program sequence

- Cycle **1420 PROBING IN PLANE**
 - **Q1120=+4**: Compensation to the mean touch point
 - **Q1121=+1**: Set basic rotation
- Cycle **1411 PROBING TWO CIRCLES**
 - **Q1120=+3**: Compensation to the mean touch point
 - **Q1121=+1**: Set basic rotation



0 BEGIN PGM TOUCHPROBE MM	
1 TOOL CALL 600 Z	
2 TCH PROBE 1420 PROBING IN PLANE ~	
Q1100=+20 ;1ST POINT REF AXIS ~	
Q1101=+20 ;1ST POINT MINOR AXIS ~	
Q1102=+0 ;1ST POINT TOOL AXIS ~	
Q1103=+80 ;2ND POINT REF AXIS ~	
Q1104=+50 ;2ND POINT MINOR AXIS ~	
Q1105=+0 ;2ND POINT TOOL AXIS ~	
Q1106=+10 ;3RD POINT REF AXIS ~	
Q1107=+60 ;3RD POINT MINOR AXIS ~	
Q1108=+0 ;3RD POINT TOOL AXIS ~	
Q372=-3 ;TOOL AXIS MINUS ~	
Q320=+2 ;SET-UP CLEARANCE ~	
Q260=+50 ;CLEARANCE HEIGHT ~	
Q1125=+2 ;BEFORE AND AFTER PROBING PROCESS ~	
Q309=+0 ;NO ERROR REACTION ~	
Q1126=+1 ;ALIGN WITH MOVE ~	
Q1120=+4 ;TRANSFER POSITION ~	
Q1121=+1 ;ROTATION AS BASIC ROTATION	
3 TCH PROBE 1411 PROBING TWO CIRCLES ~	
Q1100=+25 ;1ST POINT REF AXIS ~	
Q1101=+15 ;1ST POINT MINOR AXIS ~	

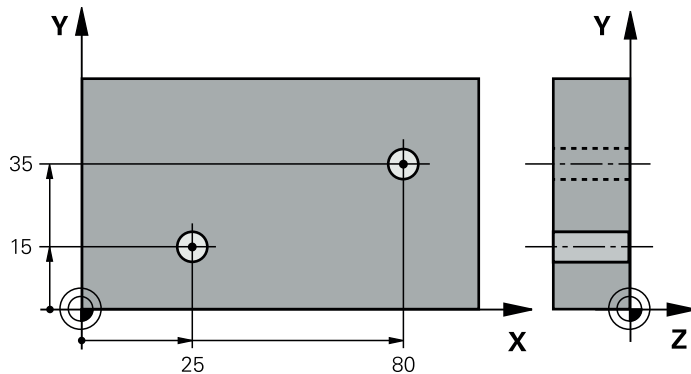
Q1102=-10	;1ST POINT TOOL AXIS ~	
Q1116=+8	;DIAMETER 1 ~	
Q1103=+80	;2ND POINT REF AXIS ~	
Q1104=+35	;2ND POINT MINOR AXIS ~	
Q1105=-10	;2ND POINT TOOL AXIS ~	
Q1117=+8	;DIAMETER 2 ~	
Q1115=+0	;1=HOLE - 2=HOLE ~	
Q423=+4	;NO. OF PROBE POINTS ~	
Q325=+0	;STARTING ANGLE ~	
Q1119=+360	;ANGULAR LENGTH ~	
Q320=+0	;SET-UP CLEARANCE ~	
Q260=+50	;CLEARANCE HEIGHT ~	
Q1125=+2	;BEFORE AND AFTER PROBING PROCESS ~	
Q309=+0	;NO ERROR REACTION ~	
Q1126=+0	;DO NOT ALIGN ~	
Q1120=+3	;TRANSFER POSITION ~	
Q1121=+1	;ROTATION AS BASIC ROTATION	
4 CALL PGM 35		; Call the part program
5 END PGM TOUCHPROBE MM		

8.4.15 Example: Aligning the rotary table from two holes

When aligning a rotary table with cycles **14xx** this must be defined by parameters **Q1126 ALIGN ROTARY AXIS**, **Q1120 TRANSFER POSITION** and **Q1121 CONFIRM ROTATION**.

Program sequence

- Cycle **1411 PROBING TWO CIRCLES**
 - **Q1126=+2**: Position rotary axes with motion control **TURN**
 - **Q1120=+3**: Compensation to the mean touch point
 - **Q1121=+2**: Execute rotary table alignment and accept offset



0	BEGIN PGM TOUCHPROBE MM	
1	TOOL CALL 600 Z	
2	TCH PROBE 1411 PROBING TWO CIRCLES ~	
	Q1100=+25 ;1ST POINT REF AXIS ~	
	Q1101=+15 ;1ST POINT MINOR AXIS ~	
	Q1102=-10 ;1ST POINT TOOL AXIS ~	
	Q1116=+8 ;DIAMETER 1 ~	
	Q1103=+80 ;2ND POINT REF AXIS ~	
	Q1104=+35 ;2ND POINT MINOR AXIS ~	
	Q1105=-10 ;2ND POINT TOOL AXIS ~	
	Q1117=+8 ;DIAMETER 2 ~	
	Q1115=+0 ;1=HOLE - 2=HOLE ~	
	Q423=+4 ;NO. OF PROBE POINTS ~	
	Q325=+0 ;STARTING ANGLE ~	
	Q1119=+360 ;ANGULAR LENGTH ~	
	Q320=+0 ;SET-UP CLEARANCE ~	
	Q260=+50 ;CLEARANCE HEIGHT ~	
	Q1125=+2 ;BEFORE AND AFTER PROBING PROCESS ~	
	Q309=+0 ;NO ERROR REACTION ~	
	Q1126=+2 ;ALIGN WITH TURN ~	
	Q1120=+3 ;TRANSFER POSITION ~	
	Q1121=+2 ;ROTATION AS ROT.-TABLE ROTATION	
3	CALL PGM 35	; Call the part program

4 END PGM TOUCHPROBE MM

8.5 Determining the preset (#17 / #1-05-1)

8.5.1 Fundamentals of touch probe cycles 408 to 419 for preset setting

Application



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

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

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

Preset and touch probe axis

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

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

Saving the calculated preset

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

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



This combination can only occur if you

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

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

Note

If you use the touch probe cycles to correct the offset of a rotary axis, the control adds the values to the current value. Corrections can lead to values outside of the modulo range -360° to $+360^\circ$. If a rotary axis already has an offset outside of the modulo range, you can reduce the value with **PRESET CORR** and the entry **0** in the modulo range.

Measurement results in Q parameters

The control saves the measurement results of the respective touch probe 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.

8.5.2 Cycle 408 SLOT CENTER PRESET (#17 / #1-05-1)

ISO programming

G408

Application

Touch probe cycle **408** finds the center of a slot and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.



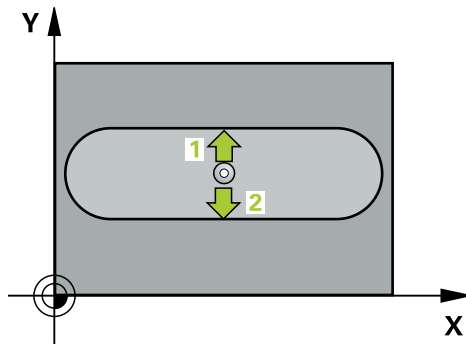
Instead of Cycle **408 SLOT CENTER PRESET**, HEIDENHAIN recommends using the more powerful Cycle **1404 PROBE SLOT/RIDGE**.

Related topics

- Cycle **1404 PROBE SLOT/RIDGE**

Further information: "Cycle 1404 PROBE SLOT/RIDGE (#17 / #1-05-1)", Page 298

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control returns the touch probe to the clearance height.
- 5 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset, (see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 212).
- 6 Then the control saves the actual values in the Q parameters listed below.
- 7 If desired, the control subsequently determines 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 functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

NOTICE

Danger of collision!

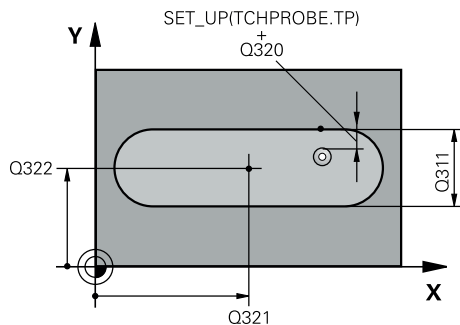
If the slot width and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the slot. In this case, the touch probe does not return to the clearance height between the two measuring points. There is a risk of collision!

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

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

Cycle parameters

Help graphic



Parameter

Q321 Center in 1st axis?

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

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

Q322 Center in 2nd axis?

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

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

Q311 Width of slot?

Width of the slot, regardless of its position in the working plane. This value has an incremental effect.

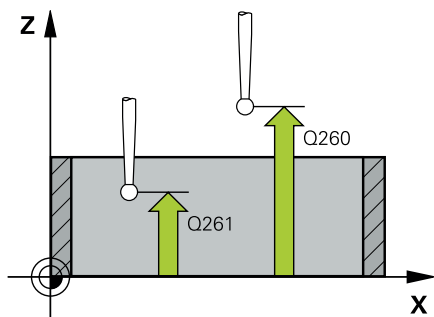
Input: **0...99999.9999**

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

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

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

Input: **1, 2**



Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q301 Move to clearance height (0/1)?

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

- 0: Move to measuring height between measuring points
- 1: Move to clearance height between measuring points

Input: **0, 1**

Help graphic	Parameter
	Q305 Number in table? Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on Q303, the control writes the entry to the preset table or datum table. If Q303=1, the control will write the data to the preset table. If Q303=0, then the control describes the datum table. The datum is not automatically activated. Further information: "Saving the calculated preset", Page 213 Input: 0...99999
	Q405 New preset? Coordinate in the measuring axis at which the control will set the calculated slot center. Default setting = 0. This value has an absolute effect. Input: -99999.9999...+9999.9999
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: 0: Write the calculated preset to the active datum table as a datum shift. The reference system is the active workpiece coordinate system. 1: Write the calculated preset to the preset table. Input: 0, 1
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: 0: Do not set the preset in the touch probe axis 1: Set the preset in the touch probe axis Input: 0, 1
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. This value has an absolute effect. Input: -99999.9999...+99999.9999

Help graphic

Parameter

Q383 Probe TS axis: Coord. 2nd axis?

Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

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

Q384 Probe TS axis: Coord. 3rd axis?

Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if **Q381** = 1. This value has an absolute effect.

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

Q333 New preset in TS axis?

Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect.

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

Example

11 TCH PROBE 408 SLOT CENTER PRESET ~	
Q321=+50	;CENTER IN 1ST AXIS ~
Q322=+50	;CENTER IN 2ND AXIS ~
Q311=+25	;SLOT WIDTH ~
Q272=+1	;MAIN AXIS ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;TO MEASURING HEIGHT ~
Q305=+10	;NUMBER IN TABLE ~
Q405=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE THE TOUCH-PROBE 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

8.5.3 Cycle 409 RIDGE CENTER PRESET (#17 / #1-05-1)

ISO programming

G409

Application

Touch probe cycle **409** finds the center of a ridge and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.



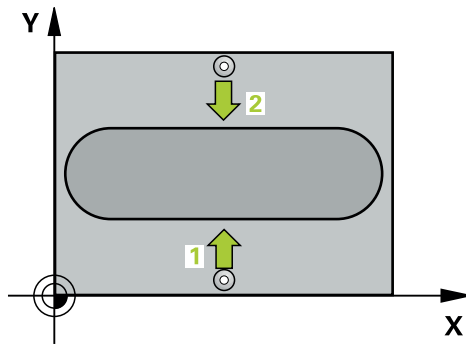
Instead of Cycle **409 RIDGE CENTER PRESET**, HEIDENHAIN recommends using the more powerful Cycle **1404 PROBE SLOT/RIDGE**.

Related topics

■ Cycle 1404 PROBE SLOT/RIDGE

Further information: "Cycle 1404 PROBE SLOT/RIDGE (#17 / #1-05-1)", Page 298

Cycle sequence



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

Further information: "Positioning logic", Page 88

- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves at clearance height to the next touch point **2** and probes it.
- 4 The control returns the touch probe to the clearance height.
- 5 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset,
- 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 functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

NOTICE

Danger of collision!

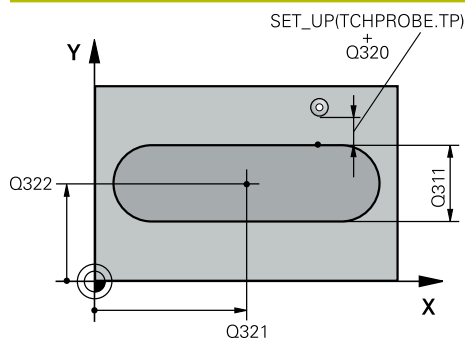
To prevent a collision between touch probe and workpiece, enter a **high** estimate for the ridge width.

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

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

Cycle parameters

Help graphic



Parameter

Q321 Center in 1st axis?

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

Input: -99999.9999...+99999.9999

Q322 Center in 2nd axis?

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

Input: -99999.9999...+99999.9999

Q311 Ridge width?

Width of the ridge, regardless of its position in the working plane. This value has an incremental effect.

Input: 0...99999.9999

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

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

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

Input: 1, 2

Q261 Measuring height in probe axis?

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

Input: -99999.9999...+99999.9999

Q320 Set-up clearance?

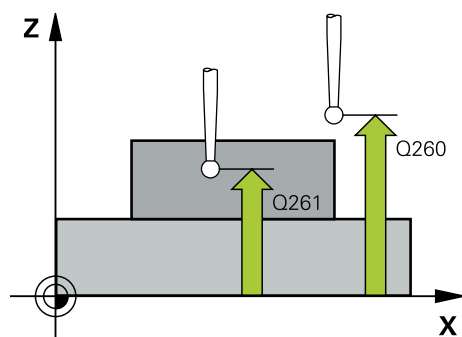
Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: 0...99999.9999 (or PREDEF)

Q260 Clearance height?

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

Input: -99999.9999...+99999.9999 (or PREDEF)



Help graphic	Parameter
	Q305 Number in table? Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on Q303, the control writes the entry to the preset table or datum table. If Q303=1, the control will write the data to the preset table. If Q303=0, then the control describes the datum table. The datum is not automatically activated. Further information: "Saving the calculated preset", Page 213 Input: 0...99999
	Q405 New preset? Coordinate in the measuring axis at which the control will set the calculated ridge center. Default setting = 0. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: 0: Write the calculated preset to the active datum table as a datum shift. The reference system is the active workpiece coordinate system. 1: Write the calculated preset to the preset table. Input: 0, 1
	Q381 Probe in TS axis? (0/1) Define whether the control will also set the preset in the touch probe axis: 0: Do not set the preset in the touch probe axis 1: Set the preset in the touch probe axis Input: 0, 1
	Q382 Probe TS axis: Coord. 1st axis? Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q383 Probe TS axis: Coord. 2nd axis? Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q384 Probe TS axis: Coord. 3rd axis? Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. This value has an absolute effect. Input: -99999.9999...+99999.9999

Help graphic**Parameter****Q333 New preset in TS axis?**

Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect.

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

Example

11 TCH PROBE 409 RIDGE CENTER PRESET ~	
Q321=+50	;CENTER IN 1ST AXIS ~
Q322=+50	;CENTER IN 2ND AXIS ~
Q311=+25	;RIDGE WIDTH ~
Q272=+1	;MAIN 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 THE TOUCH-PROBE 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

8.5.4 Cycle 410 PRESET INSIDE RECTAN (#17 / #1-05-1)

ISO programming

G410

Application

Touch probe cycle **410** finds the center of a rectangular pocket and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.



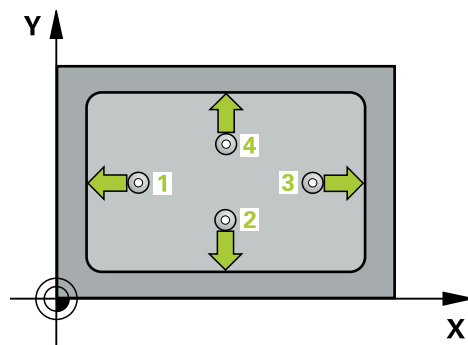
Instead of Cycle **410 PRESET INSIDE RECTAN**, HEIDENHAIN recommends using the more powerful Cycle **1403 RECTANGLE PROBING**.

Related topics

■ Cycle 1403 RECTANGLE PROBING

Further information: "Cycle 1403 RECTANGLE PROBING (#17 / #1-05-1)", Page 292

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 The control returns the touch probe to the clearance height.
- 6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset, (see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 212).
- 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 functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

NOTICE

Danger of collision!

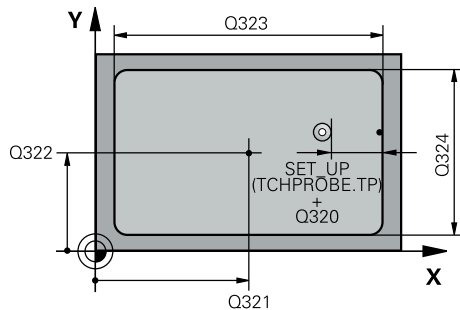
If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points. There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

- ▶ To prevent a collision between touch probe and workpiece, enter **low** estimates for the lengths of the first and second sides.
- ▶ Before the cycle definition, you must have programmed a tool call to define the touch probe axis.

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

Cycle parameters

Help graphic



Parameter

Q321 Center in 1st axis?

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

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

Q322 Center in 2nd axis?

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

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

Q323 First side length?

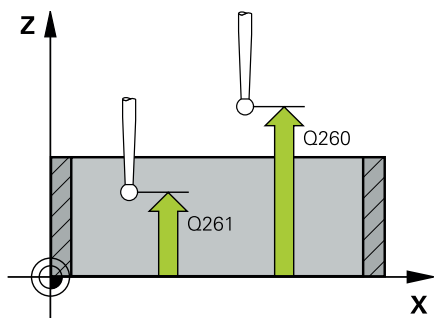
Pocket length, parallel to the main axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q324 Second side length?

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

Input: **0...99999.9999**



Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q301 Move to clearance height (0/1)?

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

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Help graphic	Parameter
	Q305 Number in table? Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on Q303, the control writes the entry to the preset table or datum table. If Q303=1, the control will write the data to the preset table. If Q303=0, then the control describes the datum table. The datum is not automatically activated. Further information: "Saving the calculated preset", Page 213 Input: 0...99999
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated pocket center. Default setting = 0. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated pocket center. Default setting = 0. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: -1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 212 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) (optional) 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? (optional) Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q383 Probe TS axis: Coord. 2nd axis? (optional) Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q384 Probe TS axis: Coord. 3rd axis? (optional) Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q333 New preset in TS axis? (optional) Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect. Input: -99999.9999...+99999.9999

Example

11 TCH PROBE 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	;TO MEASURING HEIGHT ~
Q305=+10	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE THE TOUCH-PROBE 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

8.5.5 Cycle 411 PRESET OUTS. RECTAN (#17 / #1-05-1)

ISO programming

G411

Application

Touch probe cycle **411** finds the center of a rectangular stud and defines this position as the datum. If desired, the control can also write the center point coordinates to a datum table or the preset table.



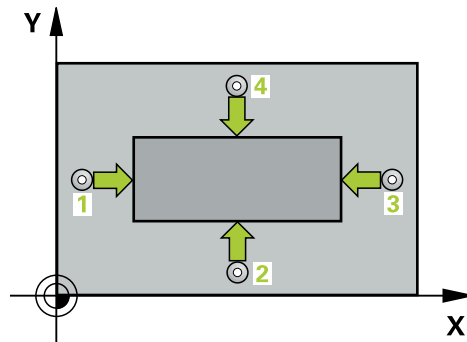
Instead of Cycle **411 PRESET OUTS. RECTAN**, HEIDENHAIN recommends using the more powerful Cycle **1403 RECTANGLE PROBING**.

Related topics

■ Cycle 1403 RECTANGLE PROBING

Further information: "Cycle 1403 RECTANGLE PROBING (#17 / #1-05-1)", Page 292

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 Then the touch probe moves either paraxially at measuring height or at clearance height to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 The control returns the touch probe to the clearance height.
- 6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset, (see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 212).
- 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 functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

NOTICE

Danger of collision!

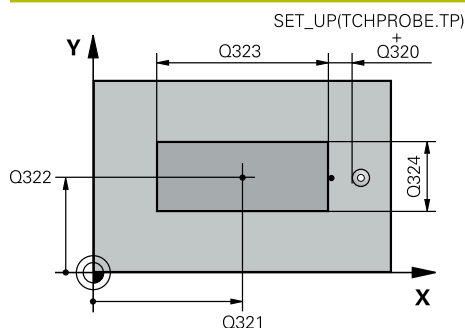
To prevent a collision between touch probe and workpiece, enter **high** estimates for the lengths of the 1st and 2nd sides.

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

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

Cycle parameters

Help graphic



Parameter

Q321 Center in 1st axis?

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

Input: -99999.9999...+9999.9999

Q322 Center in 2nd axis?

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

Input: -99999.9999...+99999.9999

Q323 First side length?

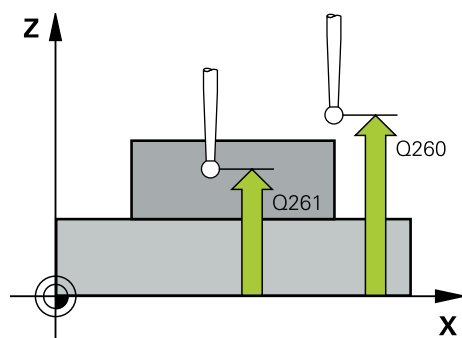
Length of stud parallel to the main axis of the working plane. This value has an incremental effect.

Input: 0...99999.9999

Q324 Second side length?

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

Input: 0...99999.9999



Q261 Measuring height in probe axis?

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

Input: -99999.9999...+99999.9999

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q301 Move to clearance height (0/1)?

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

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: 0, 1

Help graphic	Parameter
	Q305 Number in table? Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on Q303, the control writes the entry to the preset table or datum table. If Q303=1, the control will write the data to the preset table. If Q303=0, then the control describes the datum table. The datum is not automatically activated. Further information: "Saving the calculated preset", Page 213 Input: 0...99999
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated stud center. Default setting = 0. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated stud center. Default setting = 0. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q303 Meas. value transfer (0,1)? Define whether the calculated preset will be saved in the datum table or in the preset table: -1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 212 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) (optional) 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? (optional) Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. This value has an absolute effect. Input: -99999.9999...+99999.9999

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

Example

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

8.5.6 Cycle 412 PRESET INSIDE CIRCLE (#17 / #1-05-1)

ISO programming

G412

Application

Touch probe cycle **412** finds the center of a circular pocket (hole) and defines this position as the preset. If desired, the control can also write the center point coordinates to a datum table or the preset table.



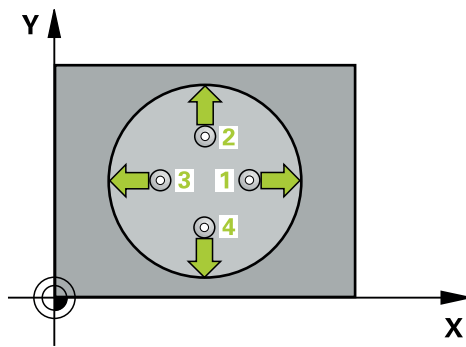
Instead of Cycle **412 PRESET INSIDE CIRCLE**, HEIDENHAIN recommends using the more powerful Cycle **1401 CIRCLE PROBING**.

Related topics

■ Cycle **1401 CIRCLE PROBING**

Further information: "Cycle 1401 CIRCLE PROBING (#17 / #1-05-1)", Page 282

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 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.
- 5 The control returns the touch probe to the clearance height.
- 6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset, (see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 212).
- 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
Q153	Actual value of diameter

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

NOTICE

Danger of collision!

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

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

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

Note on programming

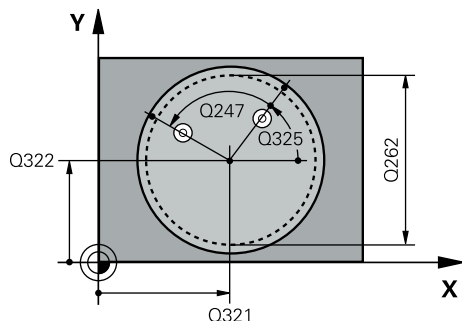
- The smaller the stepping angle **Q247**, the less accurately the control can calculate the preset. Minimum input value: 5°



Program the stepping angle to be less than 90°.

Cycle parameters

Help graphic



Parameter

Q321 Center in 1st axis?

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

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

Q322 Center in 2nd axis?

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

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

Q262 Nominal diameter?

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

Input: **0...99999.9999**

Q325 Starting angle?

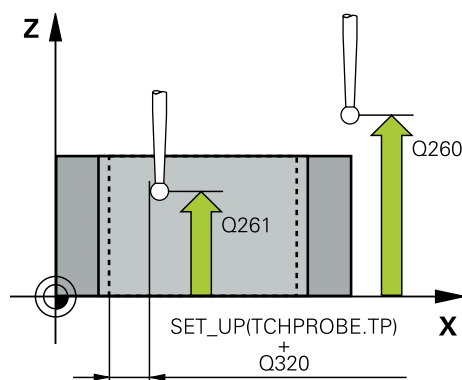
Angle between the main axis of the working plane and the first touch point. This value has an absolute effect.

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

Q247 Intermediate stepping angle?

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

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



Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Help graphic	Parameter
	Q301 Move to clearance height (0/1)? Define how the touch probe will move between the measuring points: 0: Move to measuring height between measuring points 1: Move to clearance height between measuring points Input: 0, 1
	Q305 Number in table? Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on Q303, the control writes the entry to the preset table or datum table. If Q303=1, the control will write the data to the preset table. If Q303=0, then the control describes the datum table. The datum is not automatically activated. Further information: "Saving the calculated preset", Page 213 Input: 0...99999
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated pocket center. Default setting = 0. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated pocket center. Default setting = 0. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q303 Meas. value transfer (0,1)? (optional) Define whether the calculated preset will be saved in the datum table or in the preset table: -1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 212 0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. 1: Write the calculated preset to the preset table. Input: -1, 0, +1

Help graphic	Parameter
	Q381 Probe in TS axis? (0/1) (optional) 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? (optional) Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q383 Probe TS axis: Coord. 2nd axis? (optional) Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q384 Probe TS axis: Coord. 3rd axis? (optional) Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q333 New preset in TS axis? (optional) Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q423 No. probe points in plane (4/3)? (optional) 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 (optional) Specify the path function to be used by the tool for moving between the measuring points if "traverse to clearance height" (Q301 = 1) is active. 0: Move in a straight line between machining operations 1: Move along a circular arc on the pitch circle diameter between machining operations Input: 0, 1

Example

11 TCH PROBE 412 PRESET INSIDE CIRCLE ~	
Q321=+50	;CENTER IN 1ST AXIS ~
Q322=+50	;CENTER IN 2ND AXIS ~
Q262=+75	;NOMINAL DIAMETER ~
Q325=+0	;STARTING ANGLE ~
Q247=+60	;STEPPING ANGLE ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;TO MEASURING HEIGHT ~
Q305=+12	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE THE TOUCH-PROBE 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	;CIRCLE

8.5.7 Cycle 413 PRESET OUTS. CIRCLE (#17 / #1-05-1)

ISO programming

G413

Application

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



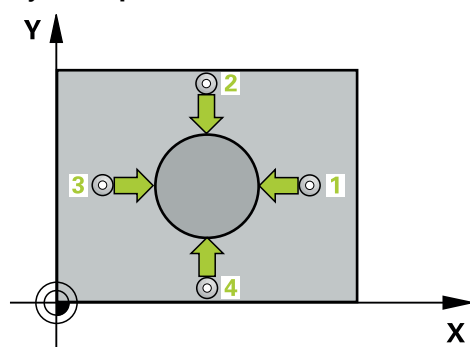
Instead of Cycle **413 PRESET OUTS. CIRCLE**, HEIDENHAIN recommends using the more powerful Cycle **1401 CIRCLE PROBING**.

Related topics

■ Cycle **1401 CIRCLE PROBING**

Further information: "Cycle 1401 CIRCLE PROBING (#17 / #1-05-1)", Page 282

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 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.
- 5 The control returns the touch probe to the clearance height.
- 6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset, see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 212.
- 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 main axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - 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.

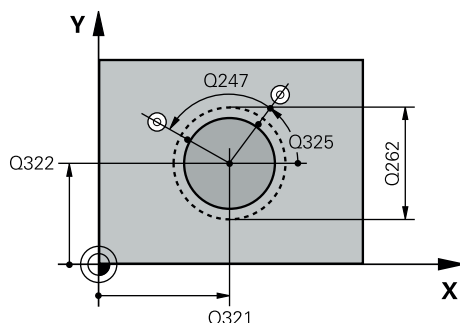
- The control will reset an active basic rotation at the beginning of the cycle.
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The smaller the stepping angle **Q247**, the less accurately the control can calculate the preset. Minimum input value: 5°



Program the stepping angle to be less than 90°.

Cycle parameters

Help graphic



Parameter

Q321 Center in 1st axis?

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

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

Q322 Center in 2nd axis?

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

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

Q262 Nominal diameter?

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

Input: **0...99999.9999**

Q325 Starting angle?

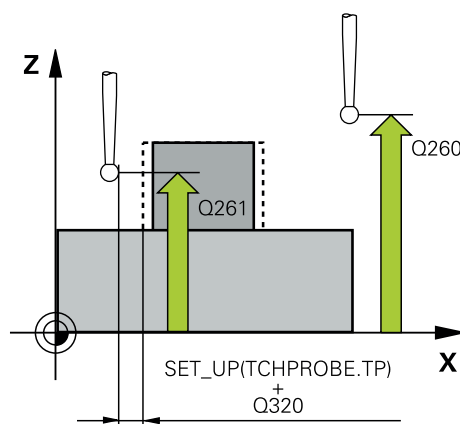
Angle between the main axis of the working plane and the first touch point. This value has an absolute effect.

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

Q247 Intermediate stepping angle?

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

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



Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Help graphic	Parameter
	Q301 Move to clearance height (0/1)? Define how the touch probe will move between the measuring points: 0: Move to measuring height between measuring points 1: Move to clearance height between measuring points Input: 0, 1
	Q305 Number in table? Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on Q303, the control writes the entry to the preset table or datum table. If Q303=1, the control will write the data to the preset table. If Q303=0, then the control describes the datum table. The datum is not automatically activated. Further information: "Saving the calculated preset", Page 213 Input: 0...99999
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated stud center. Default setting = 0. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated stud center. Default setting = 0. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q303 Meas. value transfer (0,1)? (optional) Define whether the calculated preset will be saved in the datum table or in the preset table: -1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 212 0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. 1: Write the calculated preset to the preset table. Input: -1, 0, +1

Help graphic	Parameter
	Q381 Probe in TS axis? (0/1) (optional) 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? (optional) Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q383 Probe TS axis: Coord. 2nd axis? (optional) Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q384 Probe TS axis: Coord. 3rd axis? (optional) Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q333 New preset in TS axis? (optional) Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q423 No. probe points in plane (4/3)? (optional) 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 (optional) 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	;TO MEASURING HEIGHT ~
Q305=+15	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE THE TOUCH-PROBE 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	;CIRCLE

8.5.8 Cycle 414 PRESET OUTS. CORNER (#17 / #1-05-1)

ISO programming

G414

Application

Touch probe cycle **414** finds the intersection of two lines and defines it as the preset. If desired, the control can also write the point of intersection coordinates to a datum table or the preset table.

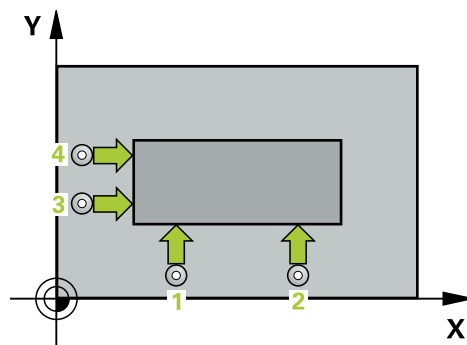
i Instead of Cycle **414 PRESET OUTS. CORNER**, HEIDENHAIN recommends using the more powerful Cycle **1416 INTERSECTION PROBING**.

Related topics

■ Cycle **1416 INTERSECTION PROBING**

Further information: "Cycle 1416 INTERSECTION PROBING (#17 / #1-05-1)", Page 192

Cycle sequence



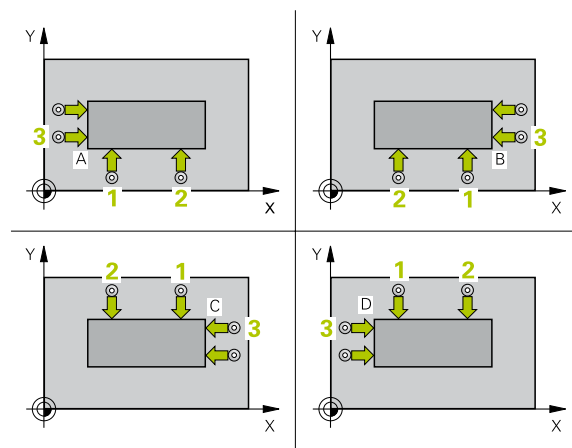
- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column). The control derives the probing direction automatically from the 3rd measuring point.
- 3 The touch probe then moves to the next touch point **2** and probes again.
- 4 The control positions the touch probe to touch point **3** and then to touch point **4** to probe two more times.
- 5 The control returns the touch probe to the clearance height.
- 6 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset, (see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 212).
- 7 Then the control saves the coordinates of the calculated corner in the Q parameters listed below.
- 8 If desired, the control subsequently determines the preset in the touch probe axis in a separate probing operation.

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

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

Definition of the corner

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



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

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

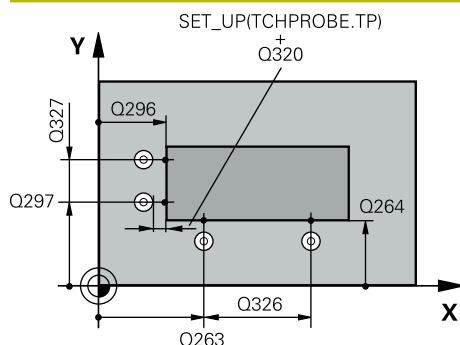
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

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

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

Q264 1st measuring point in 2nd axis?

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

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

Q326 Spacing in 1st axis?

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

Input: **0...99999.9999**

Q296 3rd measuring point in 1st axis?

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

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

Q297 3rd measuring point in 2nd axis?

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

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

Q327 Spacing in 2nd axis?

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

Input: **0...99999.9999**

Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

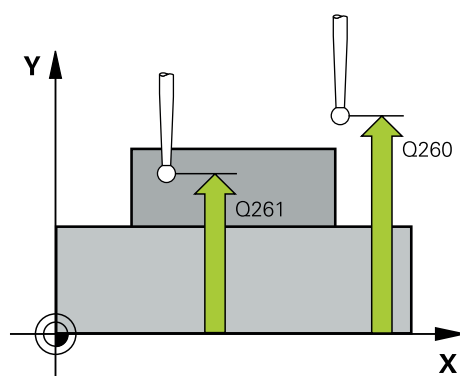
Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

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

Q260 Clearance height?

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

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



Help graphic

Parameter

Q301 Move to clearance height (0/1)?

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

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Q304 Execute basic rotation (0/1)?

Define whether the control will compensate for workpiece misalignment with a basic rotation:

0: No basic rotation

1: Basic rotation

Input: **0, 1**

Q305 Number in table?

Indicate the number of the row of the preset table or datum table, in which the control saves the corner coordinates.

Depending on **Q303**, the control writes the entry to the preset table or datum table:

If **Q303 = 1**, the control will write the data to the preset table.

If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated.

Further information: "Saving the calculated preset", Page 213

Input: **0...99999**

Q331 New preset in reference axis?

Coordinate in the main axis at which the control will set the calculated corner. Default setting = 0. This value has an absolute effect.

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

Q332 New preset in minor axis?

Coordinate in the secondary axis at which the control will set the calculated corner. Default setting = 0. This value has an absolute effect.

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

Q303 Meas. value transfer (0,1)? (optional)

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

-1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 212

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

1: Write the calculated preset to the preset table.

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

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

Example

11 TCH PROBE 414 PRESET OUTS. CORNER ~	
Q263=+37	;1ST POINT 1ST AXIS ~
Q264=+7	;1ST POINT 2ND AXIS ~
Q326=+50	;SPACING IN 1ST AXIS ~
Q296=+95	;3RD PNT IN 1ST AXIS ~
Q297=+25	;3RD PNT IN 2ND AXIS ~
Q327=+45	;SPACING IN 2ND AXIS ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;TO MEASURING HEIGHT ~
Q304=+0	;DO NOT PERFORM BASIC ROTATION ~
Q305=+7	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE THE TOUCH-PROBE 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

8.5.9 Cycle 415 PRESET INSIDE CORNER (#17 / #1-05-1)

ISO programming

G415

Application

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



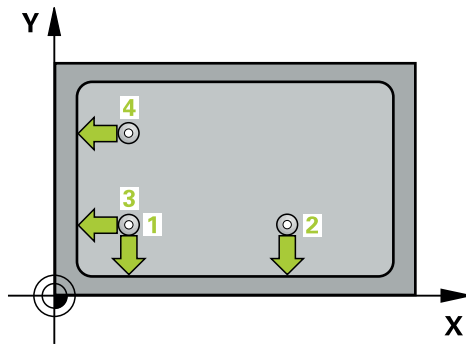
Instead of Cycle **415 PRESET INSIDE CORNER**, HEIDENHAIN recommends using the more powerful Cycle **1416 INTERSECTION PROBING**.

Related topics

■ Cycle **1416 INTERSECTION PROBING**

Further information: "Cycle 1416 INTERSECTION PROBING (#17 / #1-05-1)", Page 192

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 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 determined preset, (see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 212).
- 8 Then the control saves the coordinates of the calculated corner in the Q parameters listed below.
- 9 If desired, the control subsequently determines the preset in the touch probe axis in a separate probing operation.



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 functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

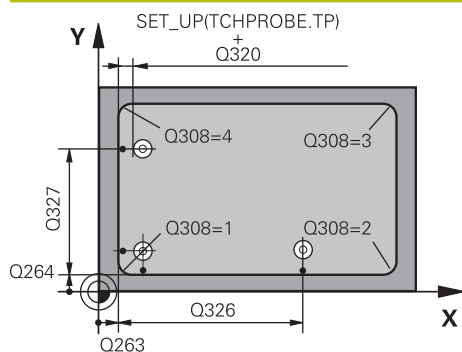
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

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

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

Q264 1st measuring point in 2nd axis?

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

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

Q326 Spacing in 1st axis?

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

Input: **0...99999.9999**

Q327 Spacing in 2nd axis?

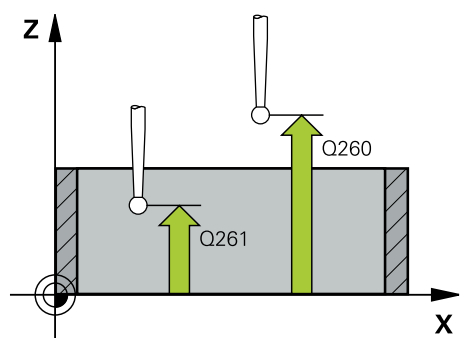
Distance between the corner and the fourth measuring point in the secondary axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

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

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

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



Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q301 Move to clearance height (0/1)?

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

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Help graphic	Parameter
	Q304 Execute basic rotation (0/1)? Define whether the control will compensate for workpiece misalignment with a basic rotation: 0: No basic rotation 1: Basic rotation Input: 0, 1
	Q305 Number in table? Indicate the number of the row of the preset table or datum table, in which the control saves the corner coordinates. Depending on Q303, the control writes the entry to the preset table or datum table: If Q303 = 1, the control will write the data to the preset table. If Q303 = 0, the control will write the data to the datum table. The datum is not automatically activated. Further information: "Saving the calculated preset", Page 213 Input: 0...99999
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated corner. Default setting = 0. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated corner. Default setting = 0. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q303 Meas. value transfer (0,1)? (optional) Define whether the calculated preset will be saved in the datum table or in the preset table: -1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 212 0: Write the calculated preset to the active datum table. The reference system is the active workpiece coordinate system. 1: Write the calculated preset to the preset table. Input: -1, 0, +1

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

Example

11 TCH PROBE 415 PRESET INSIDE CORNER ~	
Q263=+37	;1ST POINT 1ST AXIS ~
Q264=+7	;1ST POINT 2ND AXIS ~
Q326=+50	;SPACING IN 1ST AXIS ~
Q327=+45	;SPACING IN 2ND AXIS ~
Q308=+1	;CORNER ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+0	;TO MEASURING HEIGHT ~
Q304=+0	;DO NOT PERFORM BASIC ROTATION ~
Q305=+7	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE THE TOUCH-PROBE 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

8.5.10 Cycle 416 PRESET CIRCLE CENTER (#17 / #1-05-1)

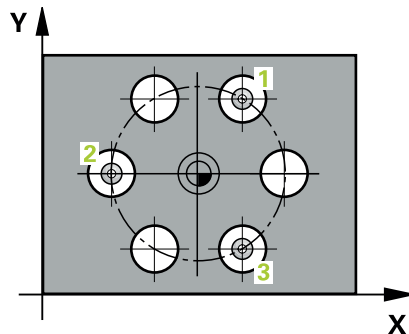
ISO programming

G416

Application

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

Cycle sequence



- 1 The control positions the touch probe at the entered center of the first hole **1**, using positioning logic.
- Further information:** "Positioning logic", Page 88
- 2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
- 3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
- 4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
- 5 The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
- 6 The control moves the touch probe to the entered measuring height and probes four points to determine the third hole center point.
- 7 The control returns the touch probe to the clearance height.
- 8 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset, (see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 212).
- 9 Then the control saves the actual values in the Q parameters listed below.
- 10 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
Q153	Actual value of bolt hole circle diameter

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

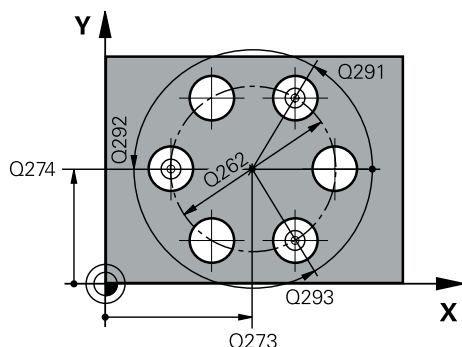
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q273 Center in 1st axis (nom. value)?

Bolt hole circle center (nominal value) in the main axis of the working plane. This value has an absolute effect.

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

Q274 Center in 2nd axis (nom. value)?

Bolt hole circle center (nominal value) in the secondary axis of the working plane. This value has an absolute effect.

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

Q262 Nominal diameter?

Enter the approximate bolt hole circle diameter. The smaller the hole diameter, the more exact the nominal diameter must be.

Input: **0...99999.9999**

Q291 Polar coord. angle of 1st hole?

Polar coordinate angle of the first hole center in the working plane. This value has an absolute effect.

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

Q292 Polar coord. angle of 2nd hole?

Polar coordinate angle of the second hole center in the working plane. This value has an absolute effect.

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

Q293 Polar coord. angle of 3rd hole?

Polar coordinate angle of the third hole center in the working plane. This value has an absolute effect.

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

Q261 Measuring height in probe axis?

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

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

Q260 Clearance height?

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

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

Help graphic	Parameter
	Q305 Number in table? Enter the row number from the preset table / datum table in which the control saves the center coordinates. Depending on Q303, the control writes the entry to the preset table or datum table. If Q303=1, the control will write the data to the preset table. If Q303=0, then the control describes the datum table. The datum is not automatically activated. Further information: "Saving the calculated preset", Page 213 Input: 0...99999
	Q331 New preset in reference axis? Coordinate in the main axis at which the control will set the calculated bolt-hole center. Default setting = 0. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q332 New preset in minor axis? Coordinate in the secondary axis at which the control will set the calculated bolt-hole circle center. Default setting = 0. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q303 Meas. value transfer (0,1)? (optional) Define whether the calculated preset will be saved in the datum table or in the preset table: -1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 212 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) (optional) 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? (optional) Coordinate of the touch point in the main axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q383 Probe TS axis: Coord. 2nd axis? (optional) Coordinate of the touch point in the secondary axis of the working plane; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q384 Probe TS axis: Coord. 3rd axis? (optional) Coordinate of the touch point in the touch probe axis; the preset will be set at this point in the touch probe axis. Only effective if Q381 = 1. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q333 New preset in TS axis? (optional) Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q320 Set-up clearance? (optional) Additional distance between touch point and ball tip. Q320 is added to SET_UP (touch probe table), and is only active when the preset is probed in the touch probe axis. This value has an incremental effect. Input: 0...99999.9999 (or PREDEF)

Example

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

8.5.11 Cycle 417 PRESET IN TS AXIS (#17 / #1-05-1)

ISO programming

G417

Application

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

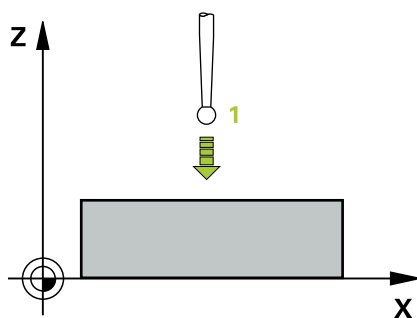
i Instead of Cycle **417 PRESET IN TS AXIS**, HEIDENHAIN recommends using the more powerful Cycle **1400 POSITION PROBING**.

Related topics

- Cycle **1400 POSITION PROBING**

Further information: "Cycle 1400 POSITION PROBING (#17 / #1-05-1)",
Page 278

Cycle sequence



- 1 Following the positioning logic, the control positions the touch probe to the programmed touch point **1**. In this process, the control offsets the touch probe by the set-up clearance in the direction of the positive touch probe axis.

Further information: "Positioning logic", Page 88

- 2 Then the touch probe moves in its own axis to the coordinate entered as touch point **1** and measures the actual position with a simple probing movement.
- 3 The control returns the touch probe to the clearance height.
- 4 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset, (see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 212).
- 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 functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

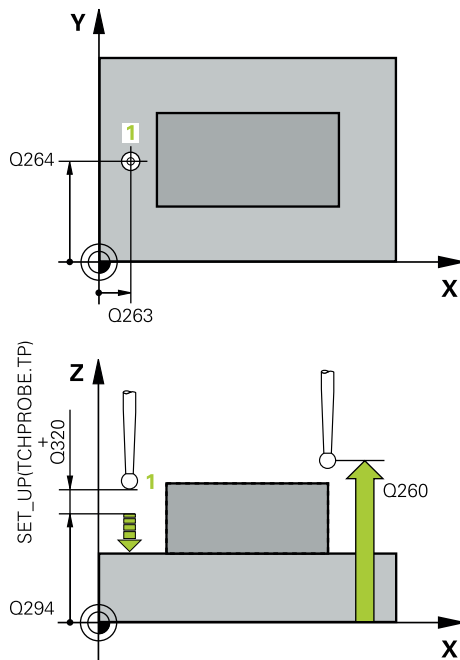
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control sets the preset in this axis.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

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

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

Q264 1st measuring point in 2nd axis?

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

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

Q294 1st measuring point in 3rd axis?

Coordinate of the first touch point in the touch probe axis. This value has an absolute effect.

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q305 Number in table?

Indicate the number of the row of the preset table or datum table, in which the control saves the coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table.

If **Q303 = 1**, the control will write the data to the preset table.

If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated.

Further information: "Saving the calculated preset", Page 213

Input: **0...99999**

Q333 New preset in TS axis?

Coordinate in the touch probe axis at which the control will set the preset. Default setting = 0. This value has an absolute effect.

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

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

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

-1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 212

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

1: Write the calculated preset to the preset table.

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

Example

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

8.5.12 Cycle 418 PRESET FROM 4 HOLES (#17 / #1-05-1)

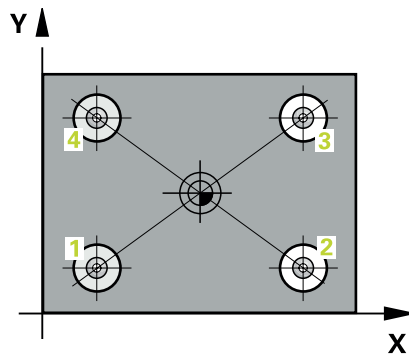
ISO programming

G418

Application

Touch probe cycle **418** calculates the intersection of the lines connecting two opposite hole center points and sets the preset at the point of intersection. If desired, the control can also write the point of intersection coordinates to a datum table or the preset table.

Cycle sequence



- 1 The control positions the touch probe at the center of the first hole **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 2 Then the probe moves to the entered measuring height and probes four points to determine the first hole center point.
- 3 The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
- 4 The control moves the touch probe to the entered measuring height and probes four points to determine the second hole center point.
- 5 The control repeats this step for holes **3** and **4**.
- 6 The control returns the touch probe to the clearance height.
- 7 Depending on the cycle parameters **Q303** and **Q305**, the control processes the determined preset, (see "Fundamentals of touch probe cycles 408 to 419 for preset setting", Page 212).
- 8 The control calculates the preset as the intersection of the lines connecting the centers of holes **1/3** and **2/4** and saves the actual values in the Q parameters listed below.
- 9 If desired, the control subsequently determines 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 functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

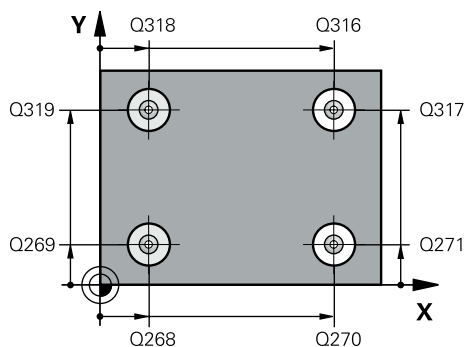
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q268 1st hole: center in 1st axis?

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

Input: -99999.9999...+9999.9999

Q269 1st hole: center in 2nd axis?

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

Input: -99999.9999...+99999.9999

Q270 2nd hole: center in 1st axis?

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

Input: -99999.9999...+99999.9999

Q271 2nd hole: center in 2nd axis?

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

Input: -99999.9999...+99999.9999

Q316 3rd hole: Center in 1st axis?

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

Input: -99999.9999...+99999.9999

Q317 3rd hole: Center in 2nd axis?

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

Input: -99999.9999...+99999.9999

Q318 4th hole: Center in 1st axis?

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

Input: -99999.9999...+99999.9999

Q319 4th hole: Center in 2nd axis?

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

Input: -99999.9999...+99999.9999

Q261 Measuring height in probe axis?

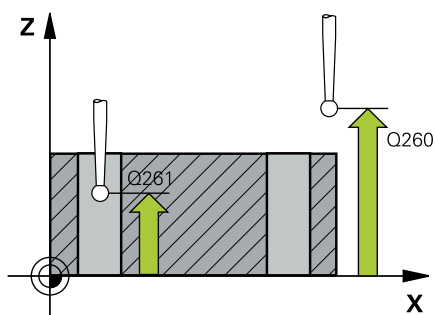
Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: -99999.9999...+99999.9999

Q260 Clearance height?

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

Input: -99999.9999...+99999.9999 (or PREDEF)



Help graphic

Parameter

Q305 Number in table?

Indicate the number of the row in the preset table or datum table in which the control saves the coordinates of the point of intersection of the connecting lines. Depending on **Q303**, the control writes the entry to the preset table or datum table.

If **Q303 = 1**, the control will write the data to the preset table.

If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated.

Further information: "Saving the calculated preset", Page 213

Input: **0...99999**

Q331 New preset in reference axis?

Coordinate in the main axis at which the control will set the calculated intersection of the connecting lines. Default setting = 0. This value has an absolute effect.

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

Q332 New preset in minor axis?

Coordinate in the secondary axis at which the control will set the calculated intersection of the connecting lines. Default setting = 0. This value has an absolute effect.

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

Q303 Meas. value transfer (0,1)? (optional)

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

-1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 212

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) (optional)

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

Example

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

8.5.13 Cycle 419 PRESET IN ONE AXIS (#17 / #1-05-1)

ISO programming

G419

Application

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



Instead of Cycle **419 PRESET IN ONE AXIS**, HEIDENHAIN recommends using the more powerful Cycle **1400 POSITION PROBING**.

Related topics

■ Cycle **1400 POSITION PROBING**

Further information: "Cycle 1400 POSITION PROBING (#17 / #1-05-1)", Page 278

Cycle sequence

- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 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 408 to 419 for preset setting", Page 212

Notes

NOTICE

Danger of collision!

When running touch probe cycles **400** to **499**, no functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations beforehand.

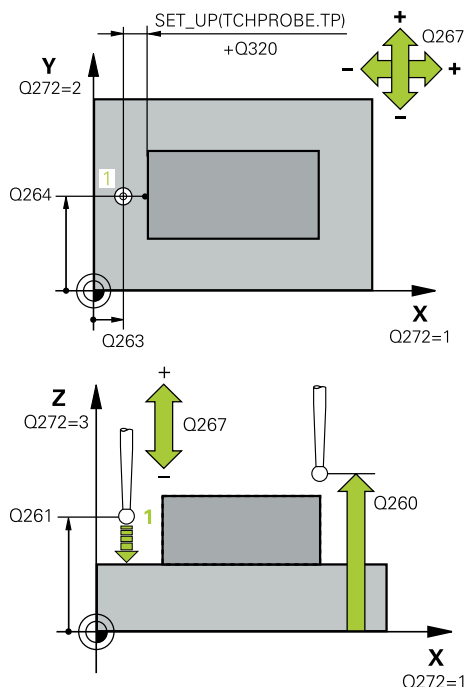
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- If you want to save the preset in several axes in the preset table, you can use Cycle **419** several times in a row. However, you also have to reactivate the preset number after every run of Cycle **419**. If you work with preset 0 as active preset, this process is not required.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

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

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

Q264 1st measuring point in 2nd axis?

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

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

Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

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

Axis in which the measurement will be made:

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

Axis assignment

Active touch probe axis: Q272 = 3	Corresponding main axis: Q272 = 1	Corresponding secondary axis: Q272 = 2
Z	X	Y
Y	Z	X
X	Y	Z

Input: **1, 2, 3**

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

Direction in which the touch probe will approach the workpiece:

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

Input: **-1, +1**

Help graphic**Parameter****Q305 Number in table?**

Indicate the number of the row of the preset table or datum table, in which the control saves the coordinates. Depending on **Q303**, the control writes the entry to the preset table or datum table.

If **Q303 = 1**, the control will write the data to the preset table.

If **Q303 = 0**, the control will write the data to the datum table. The datum is not automatically activated.

Further information: "Saving the calculated preset", Page 213

Input: **0...99999**

Q333 New preset?

Coordinate at which the control will set the preset. Default setting = 0. This value has an absolute effect.

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

Q303 Meas. value transfer (0,1)? (optional)

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

-1: Do not use. Is entered by the control when old NC programs are loaded see "Application", Page 212

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	;MAIN AXIS ~
Q267=+1	;POSITIVE TRAVERSE DIRECTION ~
Q305=+0	;NUMBER IN TABLE ~
Q333=+0	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER

8.5.14 Cycle 1400 POSITION PROBING (#17 / #1-05-1)

ISO programming

G1400

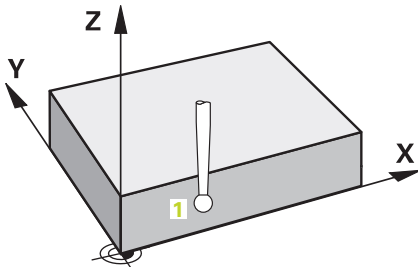
Application

Touch probe cycle **1400** measures any position in a selectable axis. You can transfer the deviations as correction values to the active row of the preset table.

Cycle **1493 EXTRUSION PROBING** allows you to repeat the touch points in a chosen direction and for a defined distance along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)", Page 388

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 2 The control then positions the touch probe to the entered measuring height **Q1102** and performs the first probing procedure with the probing feed rate **F** from the touch probe table.
- 3 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
- 4 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSFER POSITION** is defined with the value **1**, then the control corrects the ascertained deviations in the active row of the preset table.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 131

Q parameter number	Meaning
Q950 to Q952	Measured position 1 in the main axis, secondary axis, and tool axis
Q980 to Q982	Measured deviation from the first touch point
Q183	Workpiece status ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not deflected The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle. Further information: "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 385
Q970	If you have programmed Cycle 1493 EXTRUSION PROBING: Maximum deviation starting from the first touch point

Notes

NOTICE

Danger of collision!

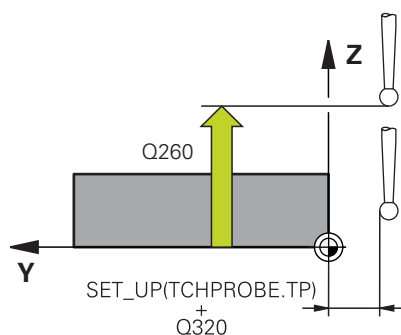
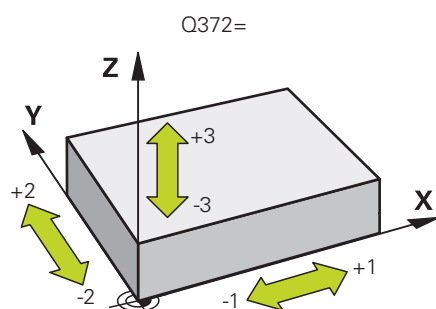
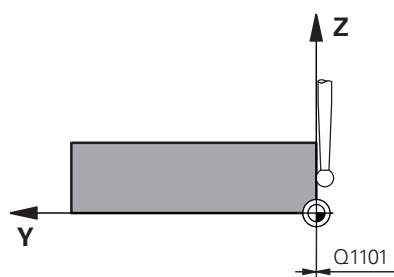
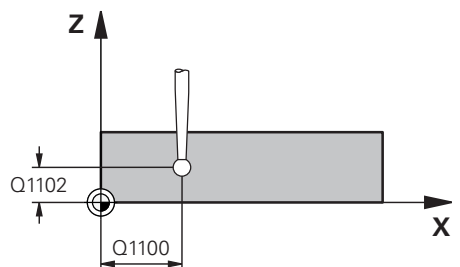
When running touch probe cycles **444** and **14xx**, no NC functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following simple transformations before using the touch probe cycle:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations before the cycle call.

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
 - Observe the fundamentals of touch probe cycles **14xx**.
- Further information:** "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 131

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

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

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

- **?**: Semi-automatic mode, see Page 133
- **-, +**: Evaluation of the tolerance, see Page 139
- **@**: Transfer of an actual position, see Page 142

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** 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** Optional input (see **Q1100**)

Q372 Probe direction (-3 to +3)?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Help graphic

Parameter

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

-1: Do not move to the clearance height.

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

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



This cycle performs only one touch point operation. The behavior is therefore identical for the values **0, 1** and **2**. Differences between these values occur only with cycles involving various probing objects or probing points (e.g., circle, slot/ridge).

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

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

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

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

Input: **0, 1, 2**

Q1120 Transfer position?

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

0: No correction

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

Input: **0, 1**

Example

11 TCH PROBE 1400 POSITION PROBING ~	
Q1100=+25	;1ST POINT REF AXIS ~
Q1101=+25	;1ST POINT MINOR AXIS ~
Q1102=-5	;1ST POINT TOOL AXIS ~
Q372=+0	;MAIN AXIS PLUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q1125=+1	;BEFORE AND AFTER PROBE OBJECT ~
Q309=+0	;NO ERROR REACTION ~
Q1120=+0	;TRANSFER POSITION

8.5.15 Cycle 1401 CIRCLE PROBING (#17 / #1-05-1)

ISO programming

G1401

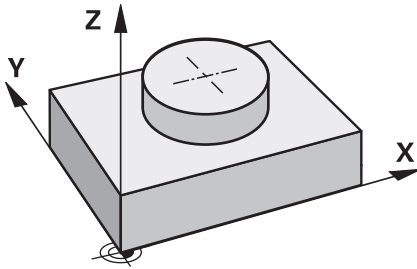
Application

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

Cycle **1493 EXTRUSION PROBING** allows you to repeat the touch points in a chosen direction and for a defined distance along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)",
Page 388

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point, using positioning logic.
Further information: "Positioning logic", Page 88
- 2 The control then positions the touch probe to the entered measuring height **Q1102** and performs the first probing procedure with the probing feed rate **F** from the touch probe table.
- 3 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
- 4 The control positions the touch probe to the next touch point.
- 5 The control moves the touch probe to the entered measuring height **Q1102** and measures the next touch point.
- 6 Depending on the definition of **Q423 NO. OF PROBE POINTS**, steps 3 to 5 repeat themselves.
- 7 The control returns the touch probe to the clearance height **Q260**.
- 8 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSFER POSITION** is defined with the value **1**, then the control corrects the ascertained deviations in the active row of the preset table.
Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)",
Page 131

Q parameter number	Meaning
Q950 to Q952	Measured circle center point in the main axis, secondary axis, and tool axis
Q966	Measured diameter
Q980 to Q982	Measured deviation of the circle center
Q996	Measured deviation of the diameter
Q183	Workpiece status ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not deflected The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle. Further information: "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 385
Q970	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation starting from the first circle center
Q973	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation starting from Diameter 1

Notes

NOTICE

Danger of collision!

When running touch probe cycles **444** and **14xx**, no NC functions for coordinate transformation must be active. Risk of collision!

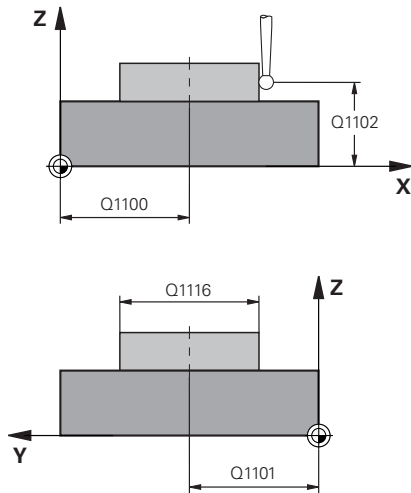
- ▶ Do not activate the following simple transformations before using the touch probe cycle:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations before the cycle call.

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- Observe the fundamentals of touch probe cycles **14xx**.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 131

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

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

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

- **"?...":** Semi-automatic mode, see Page 133
- **"...-...+...":** Evaluation of the tolerance, see Page 139
- **"...@...":** Transfer of an actual position, see Page 142

Q1101 1st noml. position of minor axis?

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

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

Q1116 Diameter of 1st position?

Diameter of the first hole or the first stud

Input: **0...9999.9999** Optional input:

- **"...-...+...":** Evaluation of the tolerance, see Page 139

Q1115 Geometry type (0/1)?

Type of object to be probed:

0: Hole

1: Stud

Input: **0, 1**

Q423 Number of probes?

Number of touch points on the diameter

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

Q325 Starting angle?

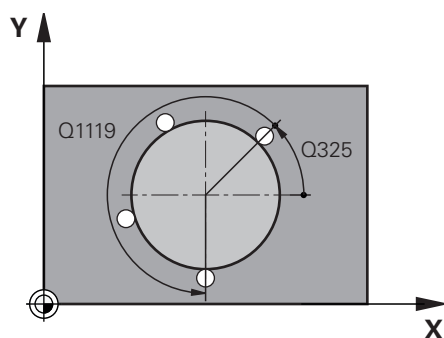
Angle between the main axis of the working plane and the first touch point. This value has an absolute effect.

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

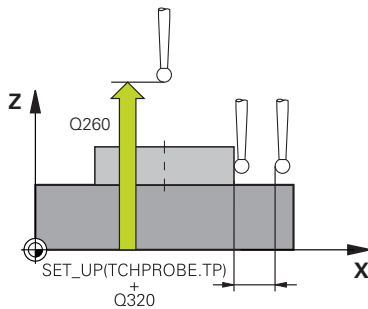
Q1119 Arc angular length?

Angular range in which the touch points are distributed.

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



Help graphic



Parameter

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q1125 Traverse to clearance height?

Positioning behavior between the touch points

-1: Do not move to the clearance height.

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

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

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



This cycle performs only one touch probing object operation. The behavior is therefore identical for the values **0** and **1**. Differences between these values occur only with cycles involving various probing objects or (e.g., two circles).

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

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

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

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

Input: **0, 1, 2**

Q1120 Transfer position?

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

0: No correction

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

Input: **0, 1**

Example

11 TCH PROBE 1401 CIRCLE PROBING ~	
Q1100=+25	;1ST POINT REF AXIS ~
Q1101=+25	;1ST POINT MINOR AXIS ~
Q1102=-5	;1ST POINT TOOL AXIS ~
Q1116=+10	;DIAMETER 1 ~
Q1115=+0	;HOLE ~
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	;BEFORE AND AFTER PROBE OBJECT ~
Q309=+0	;NO ERROR REACTION ~
Q1120=+0	;TRANSFER POSITION

8.5.16 Cycle 1402 SPHERE PROBING (#17 / #1-05-1)

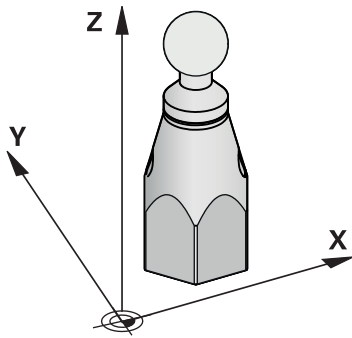
ISO programming

G1402

Application

Touch probe cycle **1402** determines the center point of a sphere. You can transfer the deviations as correction values to the active row of the preset table.

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point, using positioning logic.
- Further information:** "Positioning logic", Page 88
- 2 The control then moves the touch probe to the entered measuring height **Q1102** and performs the first probing procedure at probing speed **F** from the touch probe table.
- 3 If you program **CLEAR. HEIGHT MODE Q1125**, then the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.
- 4 The control positions the touch probe to the next touch point.
- 5 The control moves the touch probe to the entered measuring height **Q1102** and measures the next touch point.
- 6 Depending on the definition of **Q423** "Number of probe measurements", steps 3 to 5 repeat themselves.
- 7 The control moves the touch probe in the tool axis by the set-up clearance to a position above the sphere.
- 8 The touch probe moves to the center of the sphere and probes another touch point.
- 9 The touch probe returns to the clearance height **Q260**.
- 10 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSFER POSITION** is defined with the value **1**, then the control corrects the ascertained deviations in the active row of the preset table.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 131

Q parameter number	Meaning
Q950 to Q952	Measured circle center in the main axis, secondary axis, and tool axis
Q966	Measured diameter
Q980 to Q982	Measured deviation of the circle center
Q996	Measured deviation of the diameter
Q183	Workpiece status ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not deflected The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle. Further information: "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 385

Notes

NOTICE

Danger of collision!

When running touch probe cycles **444** and **14xx**, no NC functions for coordinate transformation must be active. Risk of collision!

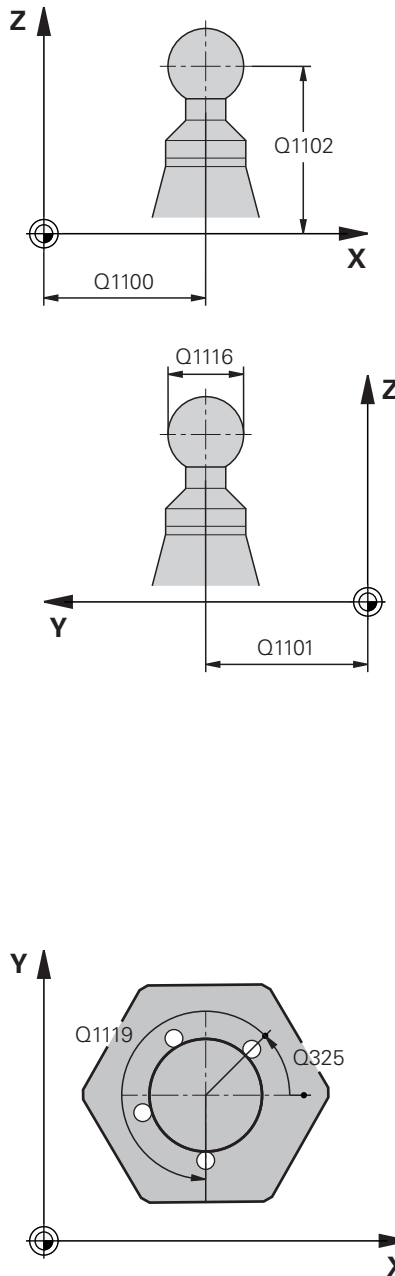
- ▶ Do not activate the following simple transformations before using the touch probe cycle:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations before the cycle call.

- This cycle can be executed only 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**.
- Observe the fundamentals of touch probe cycles **14xx**.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 131

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

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

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

- **"?...":** Semi-automatic mode, see Page 133
- **"...-...+...":** Evaluation of the tolerance, see Page 139
- **"...@...":** Transfer of an actual position, see Page 142

Q1101 1st noml. position of minor axis?

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

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

Q1116 Diameter of 1st position?

Diameter of the sphere

Input: **0...9999.9999** Optional input (see **Q1100**)

- **"...-...+...":** Evaluation of the tolerance, see Page 139

Q423 Number of probes?

Number of touch points on the diameter

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

Q325 Starting angle?

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

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

Q1119 Arc angular length?

Angular range in which the touch points are distributed.

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

Q320 Set-up clearance?

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

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

Help graphic

Parameter

Q260 Clearance height?

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

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

Q1125 Traverse to clearance height?

Positioning behavior between the touch points

-1: Do not move to the clearance height.

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

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

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



This cycle performs only one touch probing object operation. The behavior is therefore identical for the values **0** and **1**. Differences between these values occur only with cycles involving various probing objects or (e.g., two circles).

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

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

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

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

Input: **0, 1, 2**

Q1120 Transfer position?

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

0: No correction

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

Input: **0, 1**

Example

11 TCH PROBE 1402 SPHERE PROBING ~	
Q1100=+25	;1ST POINT REF AXIS ~
Q1101=+25	;1ST POINT MINOR AXIS ~
Q1102=-5	;1ST POINT TOOL AXIS ~
Q1116=+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	;BEFORE AND AFTER PROBE OBJECT ~
Q309=+0	;ERROR REACTION ~
Q1120=+0	;TRANSFER POSITION

8.5.17 Cycle 1403 RECTANGLE PROBING (#17 / #1-05-1)

ISO programming

G1403

Application

Touch probe cycle **1403** determines the center and both lengths of a rectangle.

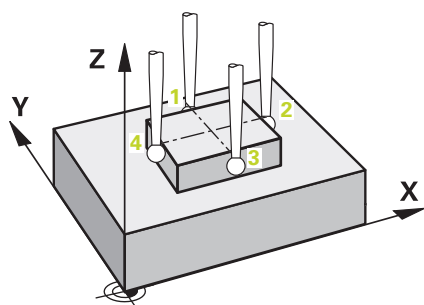
The control probes opposing touch points, one on each side. The control probes perpendicularly to the side surface of the object to be probed, even if the object to be probed is rotated.

You can transfer the deviations from the nominal position as correction values to the active row of the preset table.

Cycle **1493 EXTRUSION PROBING** allows you to repeat the touch points in a chosen direction and for a defined distance along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)",
Page 388

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 2 The control then positions the touch probe to the entered measuring height **Q1102** and performs the first probing procedure with the probing feed rate **F** from the touch probe table.
- 3 Depending on the selected type of geometry in the parameter **Q1115**, the control proceeds as follows:
 - For a pocket (**Q1115=0**) and depending on the definition of **Q1125 CLEAR. HEIGHT MODE**, the control retracts the touch probe at **FMAX_PROBE** to the clearance height **Q260**.
 - For an island (**Q1115=1**) the control retracts the touch probe at **FMAX_PROBE** back to the clearance height **Q260** after each probe point, regardless of **Q1125**.
- 4 The touch probe moves to the next touch point **2** and performs the second probing procedure at the probing rate **F**.
- 5 The control repeats Steps 1 to 4 until all four touch points have been probed.
- 6 The control saves the measured results in the following Q parameters. If **Q1120 TRANSFER POSITION** is defined with the value **1**, then the control corrects the ascertained deviations in the active row of the preset table.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)",
Page 131

Q parameter number	Meaning
Q950 to Q952	Measured center point of the rectangle in the main axis, auxiliary axis and tool axis
Q968	Measured width in the minor axis
Q969	Measured length in the major axis
Q998	Measured width deviation
Q999	Measured length deviation
Q183	Workpiece status ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not deflected The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle. Further information: "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 385
Q970	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation from the center of the rectangle
Q975	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation based on the width
Q976	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation based on the length

Notes

NOTICE

Danger of collision!

When running touch probe cycles **444** and **14xx**, no NC functions for coordinate transformation must be active. Risk of collision!

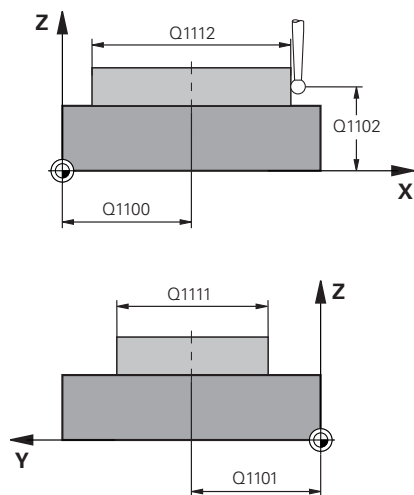
- ▶ Do not activate the following simple transformations before using the touch probe cycle:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations before the cycle call.

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- When performing an extrusion with Cycle **1493 EXTRUSION PROBING**, the only possible extrusion direction is the tool axis.
- Observe the fundamentals of touch probe cycles **14xx**.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 131

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

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

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

- **"?...":** Semi-automatic mode, see Page 133
- **"...-...+...":** Evaluation of the tolerance, see Page 139
- **"...@...":** Transfer of an actual position, see Page 142

Q1101 1st noml. position of minor axis?

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

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

Q1102 1st nominal position tool axis?

Absolute nominal position of the touch points in the tool axis

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

Q1111 First side length?

Length of rectangle, parallel to the main axis of the working plane

Input: **0...9999.9999** Or - or +:

- **"...-...+...":** Evaluation of the tolerance, see Page 139

Q1112 Second side length?

Width of rectangle, parallel to the secondary axis of the working plane.

Input: **0...9999.9999** Or - or +:

- **"...-...+...":** Evaluation of the tolerance, see Page 139

Q1115 Geometry type (0/1)?

Type of object to be probed:

0: Pocket

1: Island

Input: **0, 1**

Q1114 Angle of rotation?

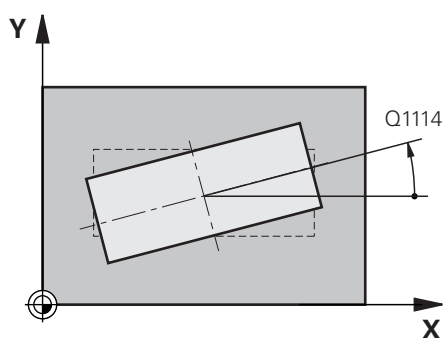
Angle by which the rectangle is rotated. The center of rotation is in **Q1100** and **Q1101**. This value has an absolute effect.

Input: **0...359999**

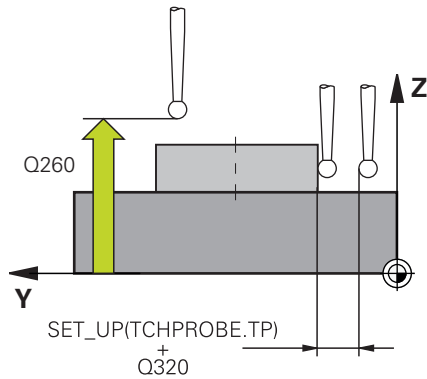
Q320 Set-up clearance?

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

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



Help graphic



Parameter

Q260 Clearance height?

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

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

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

-1: Do not move to the clearance height.

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

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

This parameter is in effect only for **Q1115=+1** (pocket).

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



This cycle performs only one touch probing object operation. The behavior is therefore identical for the values **0** and **1**. Differences between these values occur only with cycles involving various probing objects or (e.g., two circles).

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

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

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

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

Input: **0, 1, 2**

Q1120 Transfer position?

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

0: No correction

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

Input: **0, 1**

Example

11 TCH PROBE 1403 RECTANGLE PROBING ~	
Q1100=+25	;1ST POINT REF AXIS ~
Q1101=+25	;1ST POINT MINOR AXIS ~
Q1102=-5	;1ST POINT TOOL AXIS ~
Q1111=+60	;FIRST SIDE LENGTH ~
Q1112=+30	;2ND SIDE LENGTH ~
Q1115=+0	;POCKET ~
Q1114=+0	;ANGLE OF ROTATION ~
Q320=+2	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q1125=+1	;BEFORE AND AFTER PROBE OBJECT ~
Q309=+0	;NO ERROR REACTION ~
Q1120=+0	;TRANSFER POSITION

8.5.18 Cycle 1404 PROBE SLOT/RIDGE (#17 / #1-05-1)

ISO programming

G1404

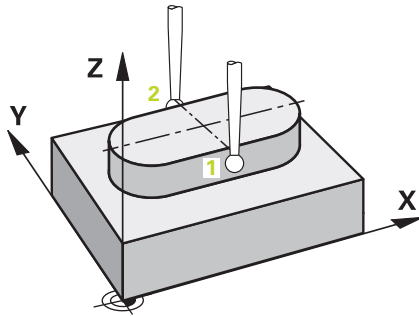
Application

Touch probe cycle **1404** determines the center of the width of a slot or ridge. The control probes the two opposing touch points. The control probes perpendicularly to the side surface of the object to be probed, even if the object to be probed is rotated. You can transfer the deviations as correction values to the active row of the preset table.

Cycle **1493 EXTRUSION PROBING** allows you to repeat the touch points in a chosen direction and for a defined distance along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)",
Page 388

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 2 The control then positions the touch probe to the entered measuring height **Q1102** and performs the first probing procedure with the probing feed rate **F** from the touch probe table.
- 3 Depending on the selected type of geometry in the parameter **Q1115**, the control proceeds as follows:
Slot **Q1115=0**:
 - If you program **CLEAR. HEIGHT MODE Q1125** with the value **0, 1** or **2**, the control positions the touch probe at **FMAX_PROBE** back to **Q260 CLEARANCE HEIGHT**.
 Ridge **Q1115=1**:
 - Independently of **Q1125**, the control positions the touch probe at **FMAX_PROBE** after every touch point back to **Q260 CLEARANCE HEIGHT**.
- 4 The touch probe moves to the next touch point **2** and performs the second probing procedure at the probing rate **F**.
- 5 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSFER POSITION** is defined with the value **1**, then the control corrects the ascertained deviations in the active row of the preset table.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)",
Page 131

Q parameter number	Meaning
Q950 to Q952	Measured center of the slot or ridge in the main axis, auxiliary axis and tool axis
Q968	Measured slot or ridge width
Q980 to Q982	Measured deviation of the center of the slot or ridge
Q998	Measured deviation of the slot width or ridge width
Q183	Workpiece status ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not deflected The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle. Further information: "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 385
Q970	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation from the center of the slot or ridge
Q975	If you have programmed Cycle 1493 EXTRUSION PROBING : Maximum deviation based on the slot width or ridge width

Notes

NOTICE

Danger of collision!

When running touch probe cycles **444** and **14xx**, no NC functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following simple transformations before using the touch probe cycle:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations before the cycle call.

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- Observe the fundamentals of touch probe cycles **14xx**.

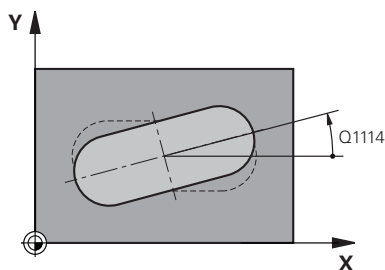
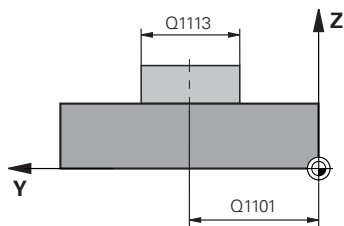
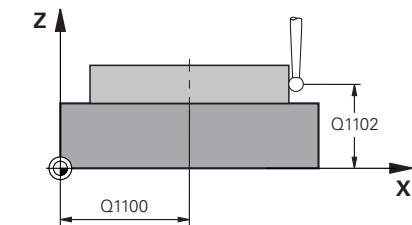
Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 131

Notes regarding probing of an extrusion

- When performing an extrusion with Cycle **1493 EXTRUSION PROBING**, the extrusion direction is possible only in the main axis or the tool axis.
- In order to measure an extrusion on a rotated slot or ridge in the main axis, you must define the rotational angle using **TRANS ROTATION** instead of parameter **Q1114**. The value 0 must be defined for **Q1114**.
- In order to measure an extrusion on a rotated slot or ridge in the tool axis, you must define the rotational angle using parameter **Q1114**.

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

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

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

- **"?...":** Semi-automatic mode, see Page 133
- **"...-...+...":** Evaluation of the tolerance, see Page 139
- **"...@...":** Transfer of an actual position, see Page 142

Q1101 1st noml. position of minor axis?

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

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

Q1102 1st nominal position tool axis?

Absolute nominal position of the touch points in the tool axis

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

Q1113 Width of slot/ridge?

Width of the slot or ridge parallel to the secondary axis of the machining plane. This value has an incremental effect.

Input: **0...9999.9999** Or **-** or **+**:

- **"...-...+...":** Evaluation of the tolerance, see Page 139

Q1115 Geometry type (0/1)?

Type of object to be probed:

0: Slot

1: Ridge

Input: **0, 1**

Q1114 Angle of rotation?

Angle about which the slot or the ridge is rotated. The center of rotation is in **Q1100** and **Q1101**. This value has an absolute effect.

Input: **0...359999**

Q320 Set-up clearance?

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

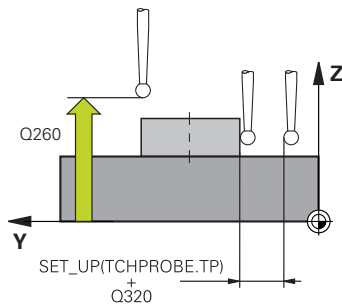
Input: **0...99999.9999** (or **PREDEF**)

Q260 Clearance height?

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

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

Help graphic



Parameter

Q1125 Traverse to clearance height?

Positioning behavior between the touch points with a slot:

-1: Do not move to the clearance height.

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

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

The parameter takes effect only with **Q1115=+1** (slot).

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



This cycle performs only one touch probing object operation. The behavior is therefore identical for the values **0** and **1**. Differences between these values occur only with cycles involving various probing objects or (e.g., two circles).

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

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

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

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

Input: **0, 1, 2**

Q1120 Transfer position?

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

0: No correction

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

Input: **0, 1**

Example

11 TCH PROBE 1404 PROBE SLOT/RIDGE ~	
Q1100=+25	;1ST POINT REF AXIS ~
Q1101=+25	;1ST POINT MINOR AXIS ~
Q1102=-5	;1ST POINT TOOL AXIS ~
Q1113=+20	;WIDTH OF SLOT/RIDGE ~
Q1115=+0	;SLOT ~
Q1114=+0	;ANGLE OF ROTATION ~
Q320=+2	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q1125=+1	;BEFORE AND AFTER PROBE OBJECT ~
Q309=+0	;NO ERROR REACTION ~
Q1120=+0	;TRANSFER POSITION

8.5.19 Cycle 1430 PROBE POSITION OF UNDERCUT (#17 / #1-05-1)

ISO programming

G1430

Application

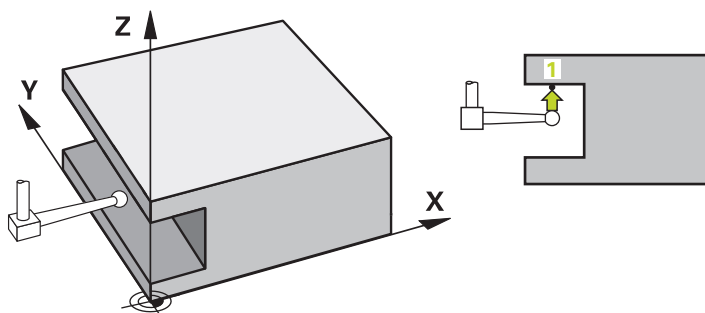
Touch probe cycle **1430** allows a position to be probed with an L-shaped stylus. The control can probe undercuts due to the shape of the stylus. You can transfer the deviations as correction values to the active row of the preset table.

In the main axis and secondary axis, the touch probe is oriented in accordance with the calibration angle. In the tool axis, the touch probe is oriented in accordance with the programmed spindle angle and the calibration angle.

Cycle **1493 EXTRUSION PROBING** allows you to repeat the touch points in a chosen direction and for a defined distance along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)", Page 388

Cycle sequence



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

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

- **Q372=+/-1**: The preposition in the main axis is at a distance of **Q1118 RADIAL APPROACH PATH** from the nominal position **Q1100**. The radial approach length takes effect in the direction opposite to the probing direction.
- **Q372=+/-2**: The pre-position in the secondary axis is at a distance of **Q1118 RADIAL APPROACH PATH** from the nominal position **Q1101**. The radial approach length takes effect in the direction opposite to the probing direction.
- **Q372=+/-3**: The pre-position of the main axis and secondary axis depends on the direction in which the stylus is oriented. The pre-position is at a distance of **Q1118 RADIAL APPROACH PATH** from the nominal position. The radial approach length takes effect in the direction opposite to the spindle angle **Q336**.

Further information: "Positioning logic", Page 88

- 2 The control then positions the touch probe to the entered measuring height **Q1102** and performs the first probing procedure with the probing feed rate **F** from the touch probe table. The probing feed rate must be identical to the calibration feed rate.
- 3 The control retracts the touch probe in the machining plane at **FMAX_PROBE** by the amount **Q1118 RADIAL APPROACH PATH**.
- 4 If you program **CLEAR. HEIGHT MODE Q1125** with the value **0, 1** or **2**, the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.

- 5 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSFER POSITION** is defined with the value **1**, then the control corrects the ascertained deviations in the active row of the preset table.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 131

Q parameter number	Meaning
Q950 to Q952	Measured position in the main axis, auxiliary axis and tool axis
Q980 to Q982	Measured deviation of the position in the main axis, auxiliary axis and tool axis
Q183	Workpiece status ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not deflected The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle. Further information: "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 385
Q970	If you have programmed Cycle 1493 EXTRUSION PROBING: Maximum deviation based on the nominal position of the first touch point

Notes

NOTICE**Danger of collision!**

When running touch probe cycles **444** and **14xx**, no NC functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following simple transformations before using the touch probe cycle:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations before the cycle call.

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- This cycle is not intended for L-shaped styli. For simple styli, HEIDENHAIN recommends Cycle **1400 POSITION PROBING**.

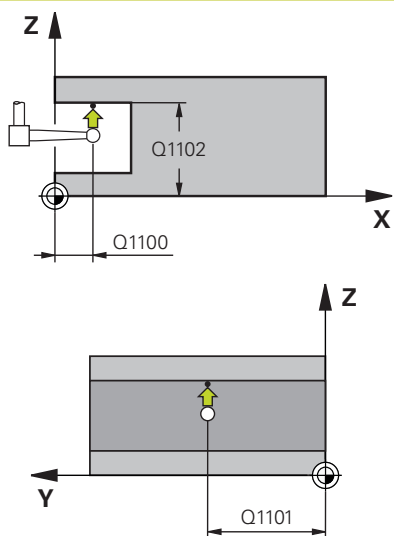
Further information: "Cycle 1400 POSITION PROBING (#17 / #1-05-1)", Page 278

- Observe the fundamentals of touch probe cycles **14xx**.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 131

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

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

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

- **?**: Semi-automatic mode, see Page 133
- **-, +**: Evaluation of the tolerance, see Page 139
- **@**: Transfer of an actual position, see Page 142

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** 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** Optional input (see **Q1100**)

Help graphic

Parameter

Q372 Probe direction (-3 to +3)?

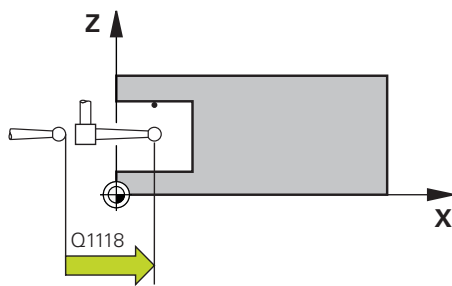
Axis defining the direction of probing. The algebraic sign lets you define whether the control moves in the positive or negative direction.

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

Q336 Angle for spindle orientation?

Angle at which the control orients the tool prior to the probing procedure. This angle takes effect only during probing in the tool axis (**Q372 = +/- 3**). This value has an absolute effect.

Input: **0...360**

**Q1118 Distance of radial approach?**

Distance to the nominal position at which the touch probe is pre-positioned in the machining plane and to which it retracts after probing.

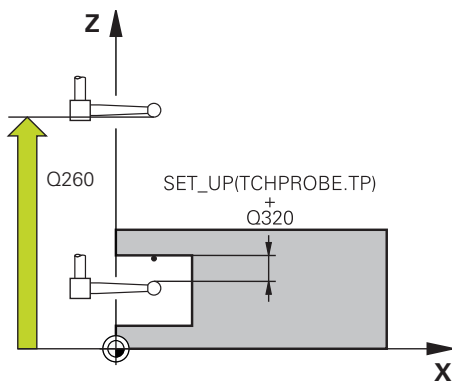
If **Q372 = +/- 1**: Distance is in the direction opposite to the probing direction.

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

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

This value has an incremental effect.

Input: **0...9999.9999**

**Q320 Set-up clearance?**

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

Input: **0...9999.9999** (or **PREDEF**)

Q260 Clearance height?

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

Input: **-9999.9999...+9999.9999** (or **PREDEF**)

Q1125 Traverse to clearance height?

Positioning behavior between the touch points:

-1: Do not move to the clearance height.

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

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



This cycle performs only one touch point operation. The behavior is therefore identical for the values **0, 1** and **2**. Differences between these values occur only with cycles involving various probing objects or probing points (e.g., circle, slot/ridge).

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

Reaction when tolerance is exceeded:

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

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

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

Input: **0, 1, 2**

Q1120 Transfer position?

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

0: No correction

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

Input: **0, 1**

Example

11 TCH PROBE 1430 PROBE POSITION OF UNDERCUT ~	
Q1100=+10	;1ST POINT REF AXIS ~
Q1101=+25	;1ST POINT MINOR AXIS ~
Q1102=-15	;1ST POINT TOOL AXIS ~
Q372=+1	;MAIN AXIS PLUS ~
Q336=+0	;ANGLE OF SPINDLE ~
Q1118=+20	;RADIAL APPROACH PATH ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q1125=+1	;BEFORE AND AFTER PROBE OBJECT ~
Q309=+0	;NO ERROR REACTION ~
Q1120=+0	;TRANSFER POSITION

8.5.20 Cycle 1434 PROBE SLOT/RIDGE UNDERCUT (#17 / #1-05-1)

ISO programming

G1434

Application

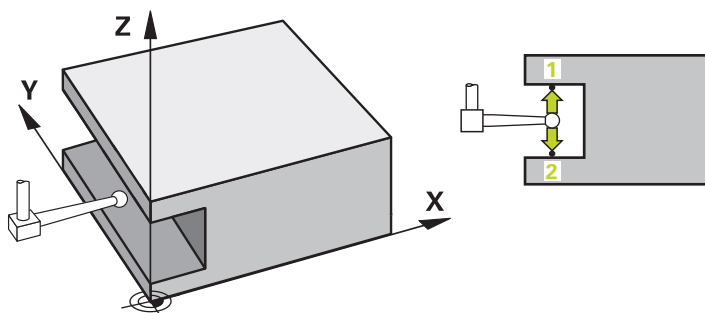
Touch probe cycle **1434** determines the center and width of a slot or a ridge using an L-shaped stylus. The control can probe undercuts due to the shape of the stylus. The control probes the two opposing touch points. You can transfer the deviations as correction values to the active row of the preset table.

The control orients the touch probe to the calibration angle from the touch probe table.

Cycle **1493 EXTRUSION PROBING** allows you to repeat the touch points in a chosen direction and for a defined distance along a straight line.

Further information: "Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)", Page 388

Cycle sequence



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

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

- **Q1139=+1**: The pre-position in the main axis is at a distance of **Q1118 RADIAL APPROACH PATH** from the nominal position **Q1100**. The direction of the radial approach length **Q1118** depends on the algebraic sign. The pre-position of the secondary axis is equivalent to the nominal position.
- **Q1139=+2**: The pre-position in the secondary axis is at a distance of **Q1118 RADIAL APPROACH PATH** from the nominal position **Q1101**. The direction of the radial approach length **Q1118** depends on the algebraic sign. The pre-position of the main axis is equivalent to the nominal position.

Further information: "Positioning logic", Page 88

- 2 The control then positions the touch probe at the entered measuring height **Q1102** and performs the first probing procedure **1** at probing feed rate **F** from the touch probe table. The probing feed rate must be identical to the calibration feed rate.
- 3 The control retracts the touch probe in the machining plane at **FMAX_PROBE** by the amount **Q1118 RADIAL APPROACH PATH**.
- 4 The control positions the touch probe to the next touch point **2** and performs the second probing procedure at probing feed rate **F**.
- 5 The control retracts the touch probe in the machining plane at **FMAX_PROBE** by the amount **Q1118 RADIAL APPROACH PATH**.
- 6 If you program the parameter **CLEAR. HEIGHT MODE Q1125** with the value **0** or **1**, the control positions the touch probe at **FMAX_PROBE** back to the clearance height **Q260**.

- 7 The control saves the measured positions in the following Q parameters. If **Q1120 TRANSFER POSITION** is defined with the value **1**, then the control corrects the ascertained deviations in the active row of the preset table.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 131

Q parameter number	Meaning
Q950 to Q952	Measured center of the slot or ridge in the main axis, auxiliary axis and tool axis
Q968	Measured slot or ridge width
Q980 to Q982	Measured deviation of the center of the slot or ridge
Q998	Measured deviation of the slot width or ridge width
Q183	Workpiece status ■ -1 = Not defined ■ 0 = Good ■ 1 = Rework ■ 2 = Scrap ■ 3 = Stylus not deflected The control displays the workpiece status 3 only in connection with the 441 FAST PROBING cycle. Further information: "Cycle 441 FAST PROBING (#17 / #1-05-1)", Page 385
Q970	If you have programmed Cycle 1493 EXTRUSION PROBING: Maximum deviation based on the center of the slot or the ridge
Q975	If you have programmed Cycle 1493 EXTRUSION PROBING: Maximum deviation based on the slot width or ridge width

Notes

NOTICE

Danger of collision!

When running touch probe cycles **444** and **14xx**, no NC functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following simple transformations before using the touch probe cycle:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - Cycle **26 AXIS-SPECIFIC SCALING**
- ▶ Reset any coordinate transformations before the cycle call.

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- If you program in the radial approach length **Q1118=-0**, then the algebraic sign has no effect. The behavior is identical to +0.
- This cycle is intended for an L-shaped stylus. For simple styli, HEIDENHAIN recommends Cycle **1404 PROBE SLOT/RIDGE**.

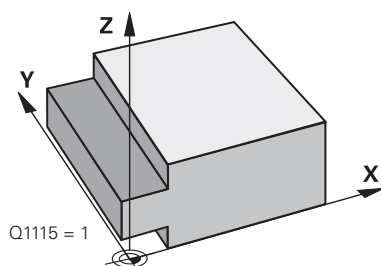
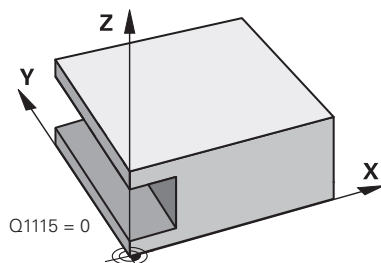
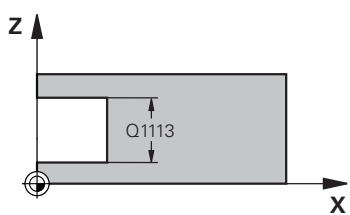
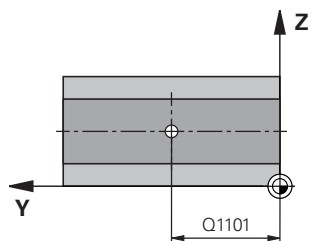
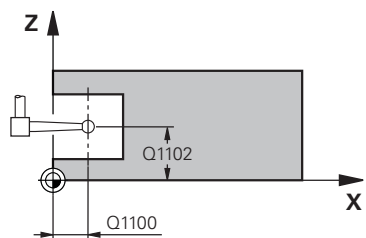
Further information: "Cycle 1404 PROBE SLOT/RIDGE (#17 / #1-05-1)", Page 298

- Observe the fundamentals of touch probe cycles **14xx**.

Further information: "Fundamentals of touch probe cycles 14xx (#17 / #1-05-1)", Page 131

Cycle parameters

Help graphic



Parameter

Q1100 1st noml. position of ref. axis?

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

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

- **"?..."**: Semi-automatic mode, see Page 133
- **"...-...+..."**: Evaluation of the tolerance, see Page 139
- **"...@..."**: Transfer of an actual position, see Page 142

Q1101 1st noml. position of minor axis?

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

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

Q1102 1st nominal position tool axis?

Absolute spindle position of the center in the tool axis

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

Q1113 Width of slot/ridge?

Width of the slot or ridge parallel to the secondary axis of the machining plane. This value has an incremental effect.

Input: **0...9999.9999** Or **-** or **+**:

"...-...+...": Evaluation of the tolerance, see Page 139

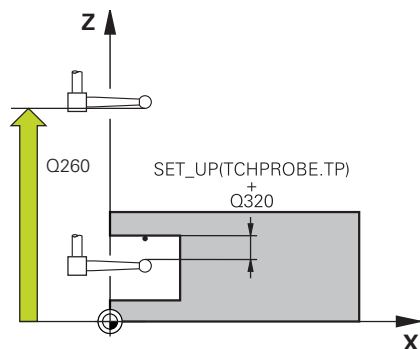
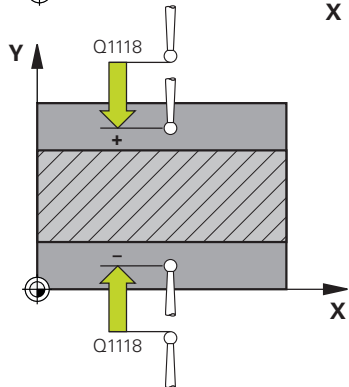
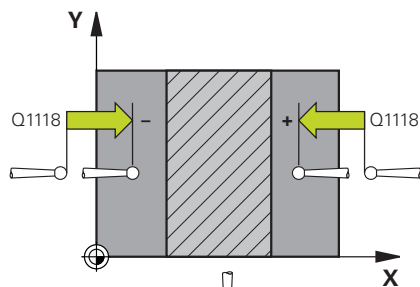
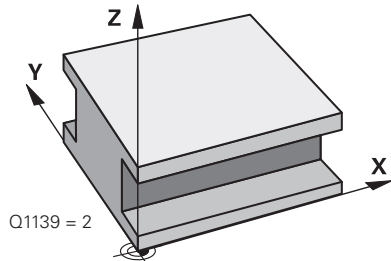
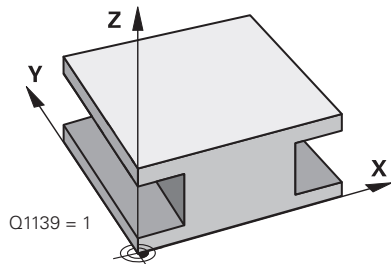
Q1115 Geometry type (0/1)?

Type of object to be probed:

- 0**: Slot
- 1**: Ridge

Input: **0, 1**

Help graphic



Parameter

Q1139 Object plane (1-2)?

Plane in which the control interprets the probing direction.

1: YZ plane

2: ZX plane

Input: **1, 2**

Q1118 Distance of radial approach?

Distance to the nominal position at which the touch probe is pre-positioned in the machining plane and to which it retracts after probing. The direction of **Q1118** is equivalent to the probing direction and is in the direction opposite to the algebraic sign. This value has an incremental effect.

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q1125 Traverse to clearance height?

Positioning behavior before and after the cycle:

-1: Do not move to the clearance height.

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

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



This cycle performs only one touch probing object operation. The behavior is therefore identical for the values **0** and **1**. Differences between these values occur only with cycles involving various probing objects or (e.g., two circles).

Q309 Reaction to tolerance error?

Reaction when tolerance is exceeded:

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

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

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

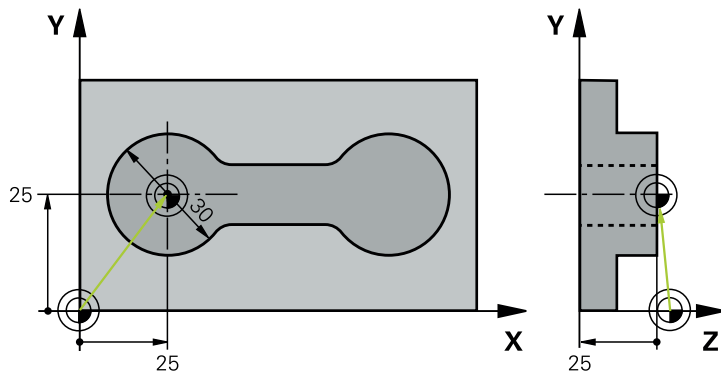
Input: **0, 1, 2**

Help graphic	Parameter
	Q1120 Transfer position? Define which touch point will be used to correct the active preset: 0: No correction 1: Correction of the active preset based on the center point. The control corrects the active preset by the amount of the deviation of the nominal and actual position of the center. Input: 0, 1

Example

11 TCH PROBE 1434 PROBE SLOT/RIDGE UNDERCUT ~	
Q1100=+25	;1ST POINT REF AXIS ~
Q1101=+25	;1ST POINT MINOR AXIS ~
Q1102=-5	;1ST POINT TOOL AXIS ~
Q1113=+20	;WIDTH OF SLOT/RIDGE ~
Q1115=+0	;SLOT ~
Q1139=+1	;MAIN AXIS ~
Q1118=-15	;RADIAL APPROACH PATH ~
Q320=+2	;SET-UP CLEARANCE ~
Q260=+50	;CLEARANCE HEIGHT ~
Q1125=+1	;BEFORE AND AFTER PROBE OBJECT ~
Q309=+0	;NO ERROR REACTION ~
Q1120=+0	;TRANSFER POSITION

8.5.21 Example: Presetting at center of a circular segment and on top surface of workpiece

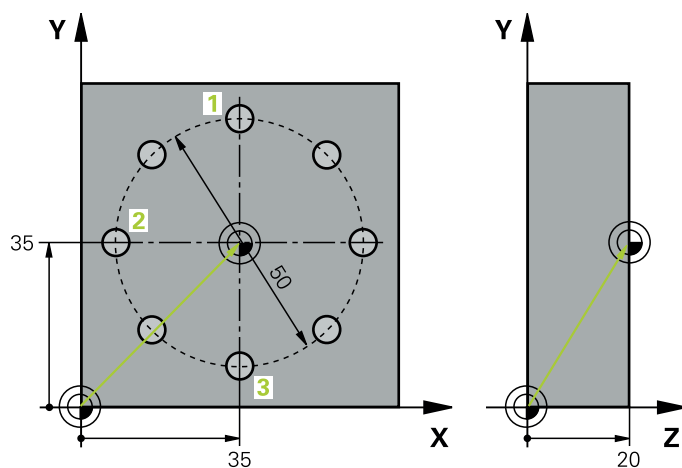


- **Q325** = Polar coordinate angle for touch point 1
- **Q247** = Stepping angle for calculating the touch points 2 to 4
- **Q305** = Write to row number 5 of the preset table
- **Q303** = Write the calculated preset to the preset table
- **Q381** = Also set the preset in the touch probe axis
- **Q365** = 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	;TO MEASURING HEIGHT ~
Q305=+5	;NUMBER IN TABLE ~
Q331=+0	;PRESET ~
Q332=+10	;PRESET ~
Q303=+1	;MEAS. VALUE TRANSFER ~
Q381=+1	;PROBE THE TOUCH-PROBE 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	;STRAIGHT LINE
3 END PGM 413 MM	

8.5.22 Example: Presetting on top surface of workpiece and at center of a bolt hole circle

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



- **Q291** = Polar coordinate angle for 1st hole center **1**
- **Q292** = Polar coordinate angle for 2nd hole center **2**
- **Q293** = Polar coordinate angle for 3rd 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 THE TOUCH-PROBE 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	

8.6 Checking the workpiece (#17 / #1-05-1)

8.6.1 Fundamentals of touch probe cycles 0, 1, and 420 to 431

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 touch probe cycle you can define if the control is to

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

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.

Outputting a measuring log to the screen:

If you execute Cycles **42x** and **43x** in combination with Cycle **441 FAST PROBING** and want to output a measuring log to the screen, you must program parameter **Q400=1** in Cycle **441**. Otherwise the control will not interrupt and the measuring log will not be displayed on the screen.

Saving the 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.

Example

Log file for touch probe cycle **421**:

Measuring log for touch probe cycle 421 Hole measuring

Date: 30-06-2005

Time: 6:55:04

Measuring program: TNC:\GEH35712\CHECK1.H

Type of dimension (0 = MM / 1 = INCH): 0

Nominal values:

Center in reference axis: 50.0000

Center in minor axis: 65.0000

Diameter: 12.0000

Given limit values:

Maximum limit for center in reference axis: 50.1000

Minimum limit for center in reference axis: 49.9000

Maximum limit for center in minor axis: 65.1000

Minimum limit for center in minor axis: 64.9000

Maximum dimension for hole: 12.0450

Minimum dimension for hole: 12.0000

Actual values:

Center in reference axis: 50.0810

Center in minor axis: 64.9530

Diameter: 12.0259

Deviations:

Center in reference axis: 0.0810

Center in minor axis: -0.0470

Diameter: 0.0259

Further measuring results: Measuring height: -5.0000

End of measuring log**Measurement results in Q parameters**

The control saves the measurement results of the respective touch probe 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.

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.

Tolerance monitoring

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

Tool monitoring

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

- the tool radius should be compensated for 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 corrected as follows:

The control always compensates for 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 corrected. The control also monitors the breakage tolerance, which is defined in column LBREAK.

You can poll whether re-working is necessary via parameter **Q181** in the NC program (**Q181=1**: rework required).

Compensating for an indexed tool

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

- **Q50** = "TOOL NAME"
- **FN 18: SYSREAD Q0 = ID990 NR10 IDX0**; specify the number of the **QS** parameter in **IDX**
- **Q0**= **Q0** +0.2; add the index of the basic tool number
- In the cycle: **Q330** = **Q0**; use the indexed tool 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 program run if the measured deviation is greater than the breakage tolerance of the tool. At the same time, the tool will be deactivated in the tool table (column TL = L).

Reference system for measurement results

The control transfers all measurement results, which reference the active coordinate system, or as the case may be, the shifted or/and rotated/tilted coordinate system, to the result parameters and the log file.

8.6.2 Cycle 0 REF. PLANE (#17 / #1-05-1)

ISO programming

G55

Application

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

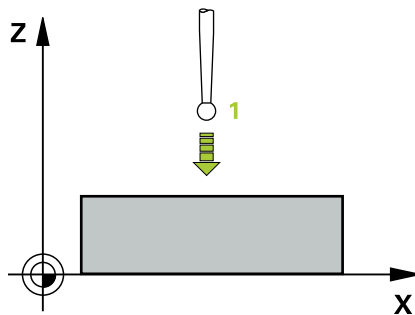
i Instead of Cycle **0 REF. PLANE**, HEIDENHAIN recommends using the more powerful Cycle **1400 POSITION PROBING**.

Related topics

- Cycle **1400 POSITION PROBING**

Further information: "Cycle 1400 POSITION PROBING (#17 / #1-05-1)", Page 278

Cycle sequence



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

Notes

NOTICE

Danger of collision!

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

- Pre-position to a position where there is no danger of collision when the programmed pre-positioning point is approached

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

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
```

8.6.3 Cycle 1 POLAR PRESET (#17 / #1-05-1)

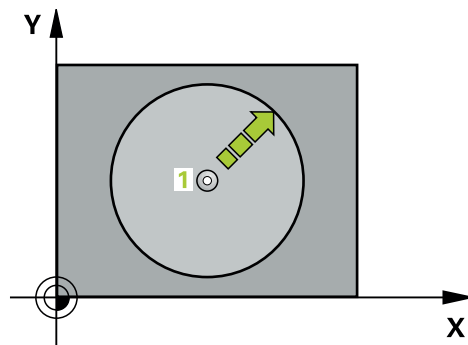
ISO programming

NC syntax is available only in Klartext programming.

Application

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

Cycle sequence



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

Notes

NOTICE

Danger of collision!

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

- Pre-position to a position where there is no danger of collision when the programmed pre-positioning point is approached

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The probing axis defined in the cycle specifies the probing plane:
Probing axis X: X/Y plane
Probing axis Y: Y/Z plane
Probing axis Z: Z/X plane

Cycle parameters

Help graphic	Parameter
	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 ANGLE:+30
```

```
13 TCH PROBE 1.2 X+0 Y+10 Z+3
```

8.6.4 Cycle 420 MEASURE ANGLE (#17 / #1-05-1)

ISO programming

G420

Application

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



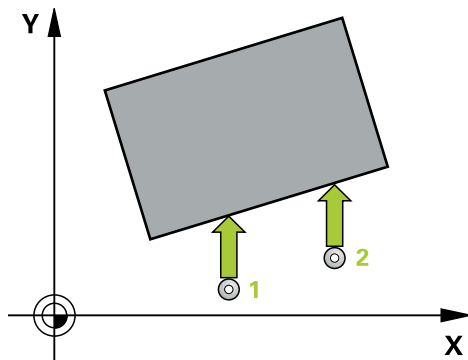
Instead of Cycle **420 MEASURE ANGLE**, HEIDENHAIN recommends using the more powerful Cycle **1410 PROBING ON EDGE**.

Related topics

- Cycle **1410 PROBING ON EDGE**

Further information: "Cycle 1410 PROBING ON EDGE (#17 / #1-05-1)", Page 170

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 2 Next, the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (**F** column).
- 3 The touch probe then moves to the next touch point **2** and probes again.
- 4 The control returns the touch probe to the clearance height and saves the measured angle in the following Q parameter:

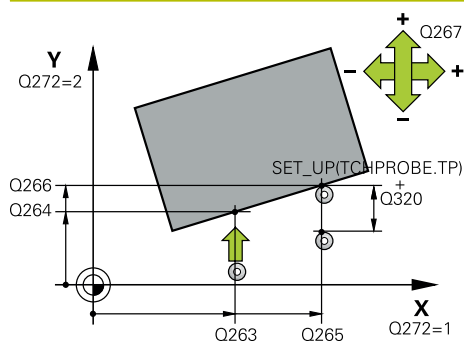
Q parameter number	Meaning
Q150	The measured angle is referenced to the main axis of the working plane.

Notes

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- If touch probe axis = measuring axis, you can measure the angle in the direction of the A axis or B axis:
 - If you want to measure the angle in the direction of the A axis, set **Q263** equal to **Q265** and **Q264** unequal to **Q266**.
 - If you want to measure the angle in the direction of the B axis, set **Q263** not equal to **Q265** and **Q264** equal to **Q266**.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

Cycle parameters**Help graphic****Parameter****Q263 1st measuring point in 1st axis?**

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

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

Q264 1st measuring point in 2nd axis?

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

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

Q265 2nd measuring point in 1st axis?

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

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

Q266 2nd measuring point in 2nd axis?

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

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

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

Axis in which the measurement will be made:

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

Input: **1, 2, 3**

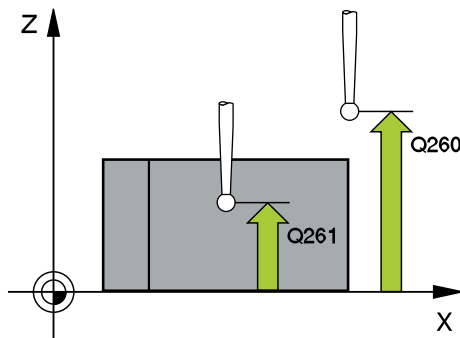
Q267 Trav. direction 1 (+1=+ / -1=-)?

Direction in which the touch probe will approach the workpiece:

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

Input: **-1, +1**

Help graphic



Parameter

Q261 Measuring height in probe axis?

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

Input: -99999.9999...+99999.9999

Q320 Set-up clearance?

Additional distance between measuring point and ball tip. The touch probe movement will start with an offset of the sum of **Q320**, **SET_UP**, and the ball-tip radius, even when probing in the tool axis direction. This value has an incremental effect.

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

Q260 Clearance height?

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

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

Q301 Move to clearance height (0/1)?

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

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: 0, 1

Q281 Measuring log (0/1/2)?

Define whether the control will create a measuring log:

1: Create a measuring log: The control will save the **log file named TCHPR420.TXT** in the folder that also contains the associated NC program.

2: Interrupt program run and display the measuring log on the control screen (you can later resume the NC program run with **NC Start**)

Input: 0, 1, 2

Example

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

8.6.5 Cycle 421 MEASURE HOLE (#17 / #1-05-1)

ISO programming

G421

Application

Touch probe cycle **421** measures the center point and diameter of a hole (or circular pocket). If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

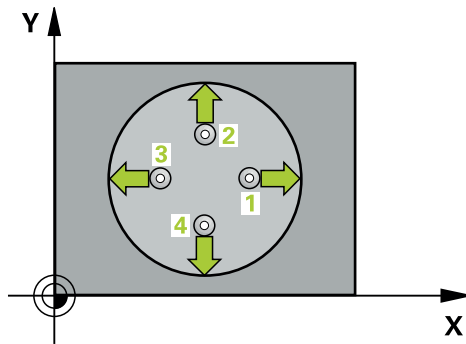
i Instead of Cycle **421 MEASURE HOLE**, HEIDENHAIN recommends using the more powerful Cycle **1401 CIRCLE PROBING**.

Related topics

■ Cycle **1401 CIRCLE PROBING**

Further information: "Cycle 1401 CIRCLE PROBING (#17 / #1-05-1)", Page 282

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 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.
- 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

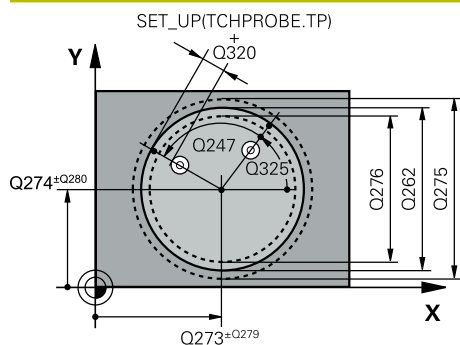
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The nominal diameter **Q262** must be between the minimum and maximum dimension (**Q276/Q275**).
- Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

Cycle parameters

Help graphic



Parameter

Q273 Center in 1st axis (nom. value)?

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

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

Q274 Center in 2nd axis (nom. value)?

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

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

Q262 Nominal diameter?

Enter the diameter of the hole.

Input: **0...99999.9999**

Q325 Starting angle?

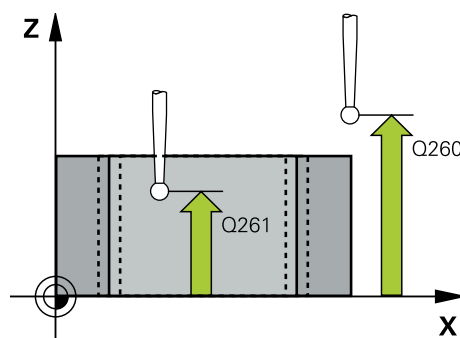
Angle between the main axis of the working plane and the first touch point. This value has an absolute effect.

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

Q247 Intermediate stepping angle?

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

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



Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

Q301 Move to clearance height (0/1)?

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

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Help graphic	Parameter
	Q275 Maximum limit of size for hole? Maximum permissible diameter for the hole (circular pocket) Input: 0...99999.9999
	Q276 Minimum limit of size? Minimum permissible diameter for the hole (circular pocket) Input: 0...99999.9999
	Q279 Tolerance for center 1st axis? Permissible position deviation in the main axis of the working plane. Input: 0...99999.9999
	Q280 Tolerance for center 2nd axis? Permissible position deviation in the secondary axis of the working plane. Input: 0...99999.9999
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: 0: Do not create a measuring log 1: Create a measuring log: The control will save the log file named TCHPR421.TXT by default in the directory that also contains the associated NC program. 2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with NC Start . Input: 0, 1, 2
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: 0: Do not interrupt program run; no error message 1: Interrupt program run and output an error message Input: 0, 1
	Q330 Tool for monitoring? (optional) Define whether the control should perform tool monitoring: 0: Monitoring not active > 0: Number or name of the tool used for machining. Via 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 Further information: "Tool monitoring", Page 321

Help graphic	Parameter
	Q423 No. probe points in plane (4/3)? (optional) 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 (optional) Specify the path function to be used by the tool for moving between the measuring points if "traverse to clearance height" (Q301 = 1) is active. 0: Move in a straight line between machining operations 1: Move along a circular arc on the pitch circle diameter between machining operations Input: 0, 1
	Parameters Q498 and Q531 have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

Example

11 TCH PROBE 421 MEASURE HOLE ~	
Q273=+50	;CENTER IN 1ST AXIS ~
Q274=+50	;CENTER IN 2ND AXIS ~
Q262=+15.25	;NOMINAL DIAMETER ~
Q325=+0	;STARTING ANGLE ~
Q247=+60	;STEPPING ANGLE ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+20	;CLEARANCE HEIGHT ~
Q301=+1	;TO CLEARANCE HEIGHT ~
Q275=+15.34	;MAXIMUM LIMIT ~
Q276=+15.16	;MINIMUM LIMIT ~
Q279=+0.1	;TOLERANCE 1ST CENTER ~
Q280=+0.1	;TOLERANCE 2ND CENTER ~
Q281=+1	;WRITE LOG FILE ~
Q309=+0	;NO ERROR REACTION ~
Q330=+0	;TOOL ~
Q423=+4	;NO. OF PROBE POINTS ~
Q365=+1	;CIRCLE ~
Q498=+0	;DO NOT ROTATE TURNING TOOL 180° ~
Q531=+0	;ANGLE OF INCIDENCE

8.6.6 Cycle 422 MEAS. CIRCLE OUTSIDE (#17 / #1-05-1)

ISO programming

G422

Application

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



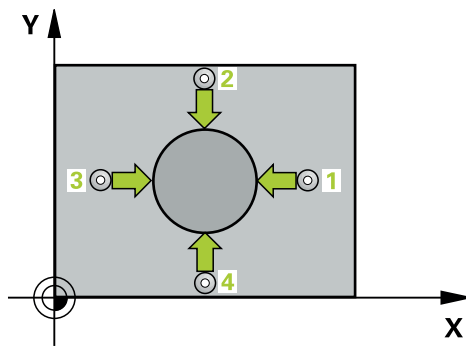
Instead of Cycle **422 MEAS. CIRCLE OUTSIDE**, HEIDENHAIN recommends using the more powerful Cycle **1401 CIRCLE PROBING**.

Related topics

■ Cycle **1401 CIRCLE PROBING**

Further information: "Cycle 1401 CIRCLE PROBING (#17 / #1-05-1)", Page 282

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 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.
- 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

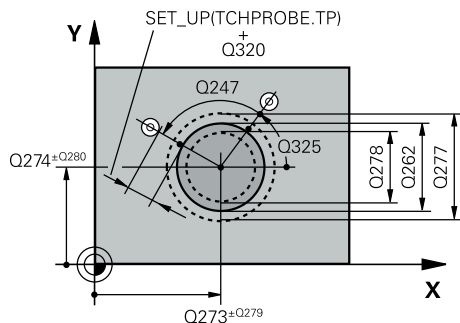
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The smaller the stepping angle, the less accurately the control can calculate the hole dimensions. Minimum input value: 5°.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

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

Cycle parameters

Help graphic



Parameter

Q273 Center in 1st axis (nom. value)?

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

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

Q274 Center in 2nd axis (nom. value)?

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

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

Q262 Nominal diameter?

Enter the diameter of the stud.

Input: **0...99999.9999**

Q325 Starting angle?

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

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

Q247 Intermediate stepping angle?

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

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

Q261 Measuring height in probe axis?

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

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

Q320 Set-up clearance?

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

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

Q260 Clearance height?

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

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

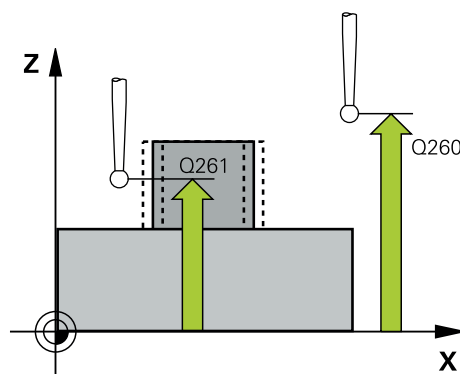
Q301 Move to clearance height (0/1)?

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

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**



Help graphic	Parameter
	Q277 Maximum limit of size for stud? Maximum permissible diameter for the stud. Input: 0...99999.9999
	Q278 Minimum limit of size for stud? Minimum permissible diameter for the stud. Input: 0...99999.9999
	Q279 Tolerance for center 1st axis? Permissible position deviation in the main axis of the working plane. Input: 0...99999.9999
	Q280 Tolerance for center 2nd axis? Permissible position deviation in the secondary axis of the working plane. Input: 0...99999.9999
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: 0: Do not create a measuring log 1: Create a measuring log: The control will save the log file named TCHPR422.TXT in the folder that also contains the associated NC program. 2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with NC Start . Input: 0, 1, 2
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: 0: Do not interrupt program run; no error message 1: Interrupt program run and output an error message Input: 0, 1
	Q330 Tool for monitoring? Define whether the control should perform tool monitoring: 0: Monitoring not active > 0: Tool number in tool table TOOL.T Input: 0...99999.9 or max. 255 characters Further information: "Tool monitoring", Page 321
	Q423 No. probe points in plane (4/3)? (optional) Define whether the control will use three or four touch points to measure the circle: 3: Use three measuring points 4: Use four measuring points (default setting) Input: 3, 4

Help graphic**Parameter****Q365 Type of traverse? Line=0/arc=1** (optional)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 operationsInput: **0, 1**

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

Example

11 TCH PROBE 422 MEAS. CIRCLE OUTSIDE ~	
Q273=+50	;CENTER IN 1ST AXIS ~
Q274=+50	;CENTER IN 2ND AXIS ~
Q262=+75	;NOMINAL DIAMETER ~
Q325=+90	;STARTING ANGLE ~
Q247=+30	;STEPPING ANGLE ~
Q261=-5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+10	;CLEARANCE HEIGHT ~
Q301=+0	;TO MEASURING HEIGHT ~
Q277=+35.15	;MAXIMUM LIMIT ~
Q278=+34.9	;MINIMUM LIMIT ~
Q279=+0.05	;TOLERANCE 1ST CENTER ~
Q280=+0.05	;TOLERANCE 2ND CENTER ~
Q281=+1	;WRITE LOG FILE ~
Q309=+0	;NO ERROR REACTION ~
Q330=+0	;TOOL ~
Q423=+4	;NO. OF PROBE POINTS ~
Q365=+1	;CIRCLE ~
Q498=+0	;DO NOT ROTATE TURNING TOOL 180° ~
Q531=+0	;ANGLE OF INCIDENCE

8.6.7 Cycle 423 MEAS. RECTAN. INSIDE (#17 / #1-05-1)

ISO programming

G423

Application

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



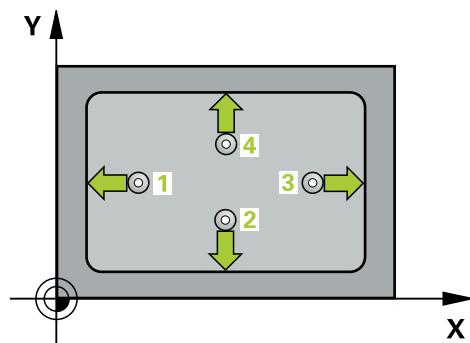
Instead of Cycle **423 MEAS. RECTAN. INSIDE**, HEIDENHAIN recommends using the more powerful Cycle **1403 RECTANGLE PROBING**.

Related topics

■ Cycle 1403 RECTANGLE PROBING

Further information: "Cycle 1403 RECTANGLE PROBING (#17 / #1-05-1)", Page 292

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 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

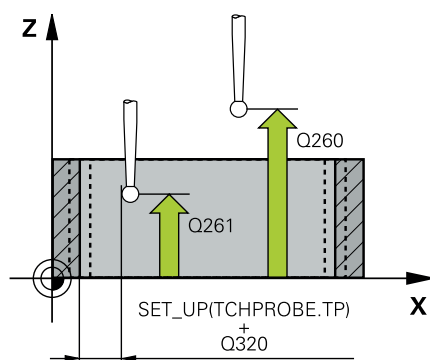
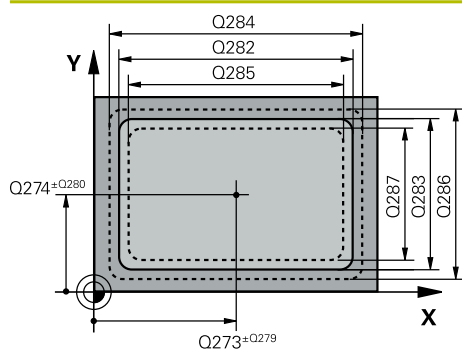
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- If the dimensions of the pocket and the set-up clearance do not permit pre-positioning in the proximity of the touch points, the control always starts probing from the center of the pocket. In this case, the touch probe does not return to the clearance height between the four measuring points.
- Tool monitoring is dependent on the deviation of the first side length.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

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

Cycle parameters

Help graphic



Parameter

Q273 Center in 1st axis (nom. value)?

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

Input: **-99999.9999...+99999.9999**

Q274 Center in 2nd axis (nom. value)?

Center of the pocket in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q282 1st side length (nominal value)?

Pocket length, parallel to the main axis of the working plane

Input: **0...99999.9999**

Q283 2nd side length (nominal value)?

Pocket length, parallel to the secondary axis of the working plane

Input: **0...99999.9999**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** (or **PREDEF**)

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** (or **PREDEF**)

Help graphic	Parameter
	Q301 Move to clearance height (0/1)? Define how the touch probe will move between the measuring points: 0: Move to measuring height between measuring points 1: Move to clearance height between measuring points Input: 0, 1
	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
	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

Help graphic**Parameter****Q330 Tool for monitoring?**

Define whether the control should perform tool monitoring:

0: Monitoring not active

> 0: Tool number in tool table TOOL.T

Input: **0...99999.9** or max. **255** characters

Further information: "Tool monitoring", Page 321

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	;TO CLEARANCE HEIGHT ~
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	;WRITE LOG FILE ~
Q309=+0	;NO ERROR REACTION ~
Q330=+0	;TOOL

8.6.8 Cycle 424 MEAS. RECTAN. OUTS. (#17 / #1-05-1)

ISO programming

G424

Application

Touch probe cycle **424** finds the center, length, and width of a rectangular stud. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.



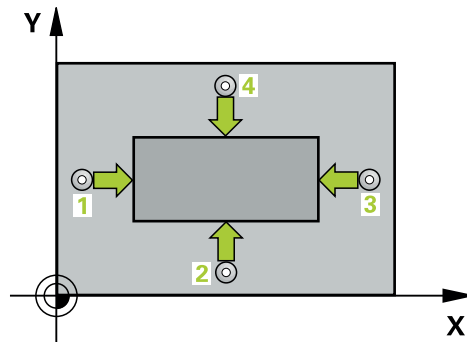
Instead of Cycle **424 MEAS. RECTAN. OUTS.**, HEIDENHAIN recommends using the more powerful Cycle **1403 RECTANGLE PROBING**.

Related topics

■ Cycle 1403 RECTANGLE PROBING

Further information: "Cycle 1403 RECTANGLE PROBING (#17 / #1-05-1)", Page 292

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 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

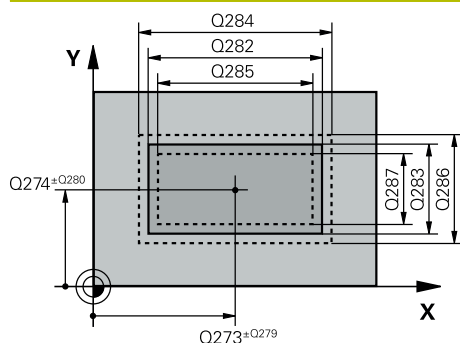
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- Tool monitoring is dependent on the deviation of the first side length.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic



Parameter

Q273 Center in 1st axis (nom. value)?

Center of the stud in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q274 Center in 2nd axis (nom. value)?

Center of the stud in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q282 1st side length (nominal value)?

Length of stud parallel to the main axis of the working plane

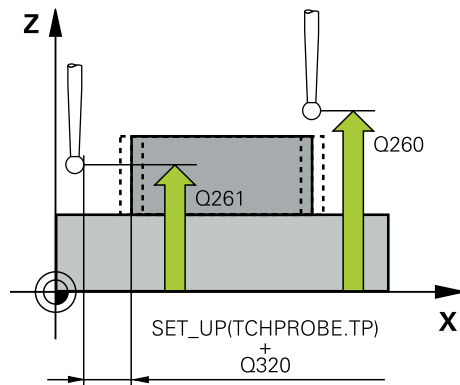
Input: **0...99999.9999**

Q283 2nd side length (nominal value)?

Length of stud parallel to the secondary axis of the working plane

Input: **0...99999.9999**

Help graphic



Parameter

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** (or **PREDEF**)

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** (or **PREDEF**)

Q301 Move to clearance height (0/1)?

Define how the touch probe will move between the measuring points:

0: Move to measuring height between measuring points

1: Move to clearance height between measuring points

Input: **0, 1**

Q284 Max. size limit 1st side length?

Maximum permissible length for the stud

Input: **0...99999.9999**

Q285 Min. size limit 1st side length?

Minimum permissible length for the stud

Input: **0...99999.9999**

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**

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 TCHPR424.TXT** in the folder that also contains the .h file

2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.

Input: **0, 1, 2**

Q309 PGM stop if tolerance exceeded?

Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message:

0: Do not interrupt program run; no error message

1: Interrupt program run and output an error message

Input: **0, 1**

Q330 Tool for monitoring?

Define whether the control should perform tool monitoring:

0: Monitoring not active

> 0: Number or name of the tool used for machining. Via 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

Further information: "Tool monitoring", Page 321

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	;TO MEASURING HEIGHT ~
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	;WRITE LOG FILE ~
Q309=+0	;NO ERROR REACTION ~
Q330=+0	;TOOL

8.6.9 Cycle 425 MEASURE INSIDE WIDTH (#17 / #1-05-1)

ISO programming

G425

Application

Touch probe cycle **425** measures the position and width of a slot (or pocket). If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation value in a Q parameter.



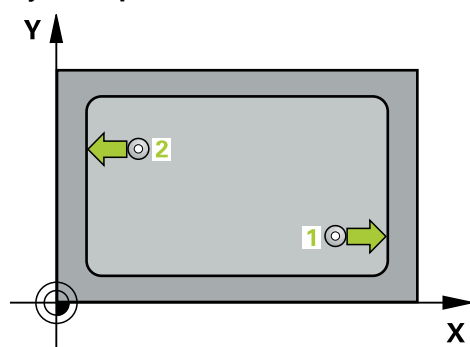
Instead of Cycle **425 MEASURE INSIDE WIDTH**, HEIDENHAIN recommends using the more powerful Cycle **1404 PROBE SLOT/RIDGE**.

Related topics

- Cycle **1404 PROBE SLOT/RIDGE**

Further information: "Cycle 1404 PROBE SLOT/RIDGE (#17 / #1-05-1)", Page 298

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 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:

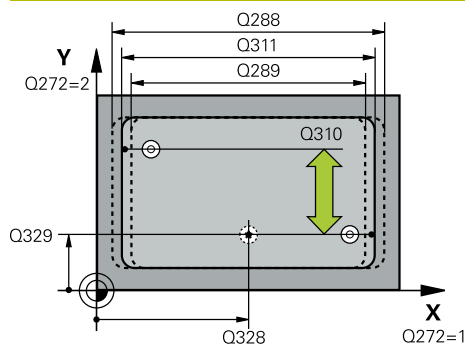
Q parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

Notes

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The nominal length **Q311** must be between the minimum and maximum dimension (**Q276/Q275**).

Cycle parameters**Help graphic****Parameter****Q328 Starting point in 1st axis?**

Starting point for probing in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q329 Starting point in 2nd axis?

Starting point for probing in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q310 Offset for 2nd measurement (+/-)?

Distance by which the touch probe is offset before the second measurement. If you enter 0, the control does not offset the touch probe. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q272 Measuring axis (1=1st / 2=2nd)?

Axis in the working plane in which the measurement will be performed:

- 1: Main axis = measuring axis
- 2: Secondary axis = measuring axis

Input: **1, 2**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999 (or PREDEF)**

Q311 Nominal length?

Nominal value of the length to be measured

Input: **0...99999.9999**

Q288 Maximum limit of size?

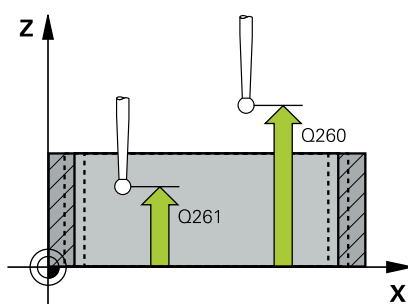
Maximum permissible length

Input: **0...99999.9999**

Q289 Minimum limit of size?

Minimum permissible length

Input: **0...99999.9999**



Help graphic	Parameter
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: 0: Do not create a measuring log 1: Create a measuring log: The control will save the log file named TCHPR425.TXT in the folder that also contains the .h file 2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with NC Start. Input: 0, 1, 2
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: 0: Do not interrupt program run; no error message 1: Interrupt program run and output an error message Input: 0, 1
	Q330 Tool for monitoring? Define whether the control should perform tool monitoring: 0: Monitoring not active > 0: Number or name of the tool used for machining. Via 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 Further information: "Tool monitoring", Page 321
	Q320 Set-up clearance? (optional) Additional distance between touch point and ball tip. Q320 is added to SET_UP (touch probe table), and is only active when the preset is probed in the touch probe axis. This value has an incremental effect. Input: 0...99999.9999 (or PREDEF)
	Q301 Move to clearance height (0/1)? (optional) Define how the touch probe will move between the measuring points: 0: Move to measuring height between measuring points 1: Move to clearance height between measuring points Input: 0, 1

Example

11 TCH PROBE 425 MEASURE INSIDE WIDTH ~	
Q328=+75	;STARTNG PNT 1ST AXIS ~
Q329=-12.5	;STARTNG PNT 2ND AXIS ~
Q310=+0	;OFFS. 2ND MEASUREMNT ~
Q272=+1	;MAIN AXIS ~
Q261=-5	;MEASURING HEIGHT ~
Q260=+10	;CLEARANCE HEIGHT ~
Q311=+25	;NOMINAL LENGTH ~
Q288=+25.05	;MAXIMUM LIMIT ~
Q289=+25	;MINIMUM LIMIT ~
Q281=+1	;WRITE LOG FILE ~
Q309=+0	;NO ERROR REACTION ~
Q330=+0	;TOOL ~
Q320=+0	;SET-UP CLEARANCE ~
Q301=+0	;TO MEASURING HEIGHT

8.6.10 Cycle 426 MEASURE RIDGE WIDTH (#17 / #1-05-1)

ISO programming

G426

Application

Touch probe cycle **426** measures the position and width of a ridge. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.



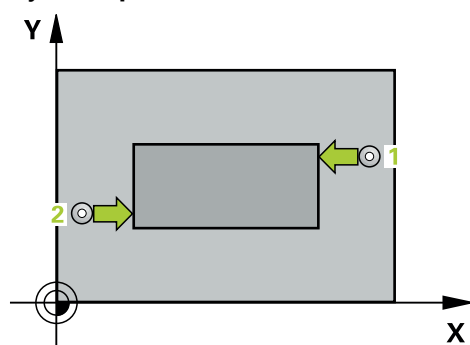
Instead of Cycle **426 MEASURE RIDGE WIDTH**, HEIDENHAIN recommends using the more powerful Cycle **1404 PROBE SLOT/RIDGE**.

Related topics

- Cycle **1404 PROBE SLOT/RIDGE**

Further information: "Cycle 1404 PROBE SLOT/RIDGE (#17 / #1-05-1)", Page 298

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.
Further information: "Positioning logic", Page 88
- 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

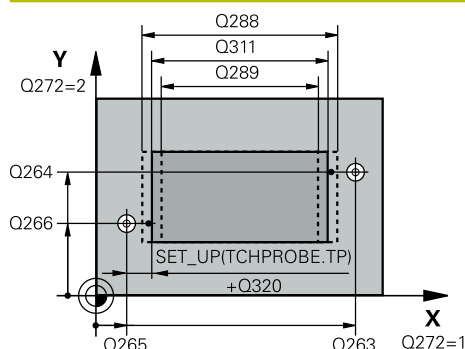
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q264 1st measuring point in 2nd axis?

Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q265 2nd measuring point in 1st axis?

Coordinate of the second touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q266 2nd measuring point in 2nd axis?

Coordinate of the second touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q272 Measuring axis (1=1st / 2=2nd)?

Axis in the working plane in which the measurement will be performed:

- 1: Main axis = measuring axis
- 2: Secondary axis = measuring axis

Input: **1, 2**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

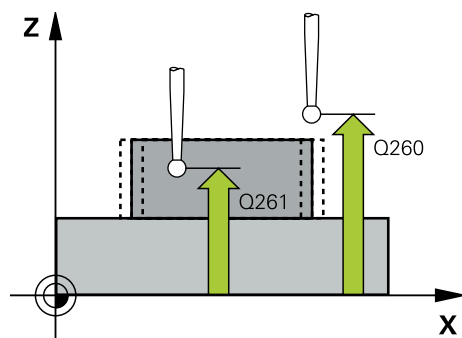
Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999 (or PREDEF)**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999 (or PREDEF)**



Help graphic	Parameter
	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
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: 0: Do not create a measuring log 1: Create a measuring log: The control will save the log file named TCHPR426.TXT in the folder that also contains the associated NC program. 2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with NC Start . Input: 0, 1, 2
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: 0: Do not interrupt program run; no error message 1: Interrupt program run and output an error message Input: 0, 1
	Q330 Tool for monitoring? Define whether the control should perform tool monitoring: 0: Monitoring not active > 0: Number or name of the tool used for machining. Via 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 Further information: "Tool monitoring", Page 321

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	;MINOR 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	;WRITE LOG FILE ~
Q309=+0	;NO ERROR REACTION ~
Q330=+0	;TOOL

8.6.11 Cycle 427 MEASURE COORDINATE (#17 / #1-05-1)

ISO programming

G427

Application

Touch probe cycle **427** measures a coordinate in a selectable axis and saves the value in a Q parameter. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.



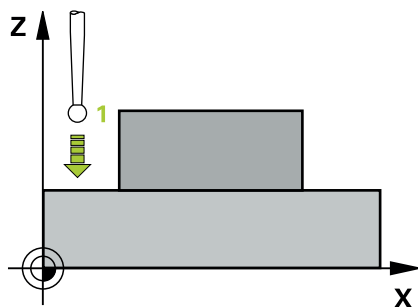
Instead of Cycle **427 MEASURE COORDINATE**, HEIDENHAIN recommends using the more powerful Cycle **1400 POSITION PROBING**.

Related topics

■ Cycle 1400 POSITION PROBING

Further information: "Cycle 1400 POSITION PROBING (#17 / #1-05-1)", Page 278

Cycle sequence



- 1 The control positions the touch probe to the pre-position of the first touch point **1**, using positioning logic.

Further information: "Positioning logic", Page 88

- 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 parameters:

Q parameter number	Meaning
Q160	Measured coordinate
Q168	Deviation of the measured coordinate

Notes

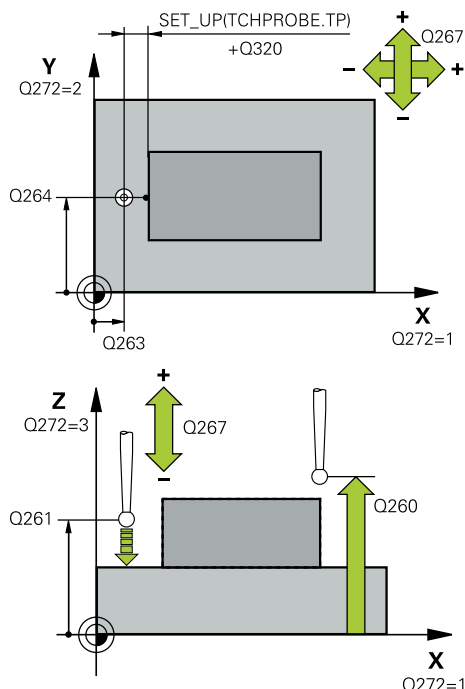
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- If an axis of the active working plane is defined as the measuring axis (**Q272** = 1 or 2), the control will perform a tool radius compensation. The control determines the direction of compensation from the defined traversing direction (**Q267**).
- If the touch probe axis is defined as the measuring axis (**Q272** = 3), the control will perform a tool length compensation.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The measuring height **Q261** must be between the minimum and maximum dimension (**Q276/Q275**).
- Parameters **Q498** and **Q531** have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q264 1st measuring point in 2nd axis?

Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q261 Measuring height in probe axis?

Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: **0...99999.9999** (or **PREDEF**)

Q272 Meas. axis (1/2/3, 1=ref. axis)?

Axis in which the measurement will be made:

- 1:** Main axis = measuring axis
- 2:** Secondary axis = measuring axis
- 3:** Touch probe axis = measuring axis

Input: **1, 2, 3**

Q267 Trav. direction 1 (+1=+ / -1=-)?

Direction in which the touch probe will approach the workpiece:

- 1:** Negative traverse direction
- +1:** Positive traverse direction

Input: **-1, +1**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** (or **PREDEF**)

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 program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.

Input: **0, 1, 2**

Help graphic	Parameter
	Q288 Maximum limit of size? (optional) Maximum permissible value Input: -99999.9999...+99999.9999
	Q289 Minimum limit of size? (optional) Minimum permissible value Input: -99999.9999...+99999.9999
	Q309 PGM stop if tolerance exceeded? (optional) 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? (optional) Define whether the control should perform tool monitoring: 0: Monitoring not active > 0: Number or name of the tool used for machining. Via 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 Further information: "Tool monitoring", Page 321
	Parameters Q498 and Q531 have no effect in this cycle. You do not need to make any entries. These parameters have only been integrated for reasons of compatibility. If, for example, you import a program of the TNC 640 turning and milling control, you will not receive an error message.

Example

11 TCH PROBE 427 MEASURE COORDINATE ~	
Q263=+35	;1ST POINT 1ST AXIS ~
Q264=+45	;1ST POINT 2ND AXIS ~
Q261=+5	;MEASURING HEIGHT ~
Q320=+0	;SET-UP CLEARANCE ~
Q272=+3	;TOOL AXIS ~
Q267=-1	;NEGATIVE TRAVERSE DIRECTION ~
Q260=+20	;CLEARANCE HEIGHT ~
Q281=+1	;WRITE LOG FILE ~
Q288=+5.1	;MAXIMUM LIMIT ~
Q289=+4.95	;MINIMUM LIMIT ~
Q309=+0	;NO ERROR REACTION ~
Q330=+0	;TOOL ~
Q498=+0	;DO NOT ROTATE TURNING TOOL 180° ~
Q531=+0	;ANGLE OF INCIDENCE

8.6.12 Cycle 430 MEAS. BOLT HOLE CIRC (#17 / #1-05-1)

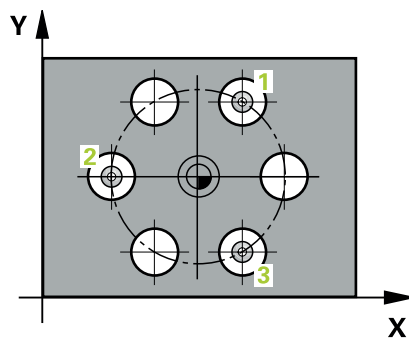
ISO programming

G430

Application

Touch probe cycle **430** finds the center and diameter of a bolt hole circle by probing three holes. If you define the corresponding tolerance values in the cycle, the control makes a nominal-to-actual value comparison and saves the deviation values in Q parameters.

Cycle sequence



- 1 The control positions the touch probe at the entered center of the first hole **1**, using positioning logic.
- Further information:** "Positioning logic", Page 88
- 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

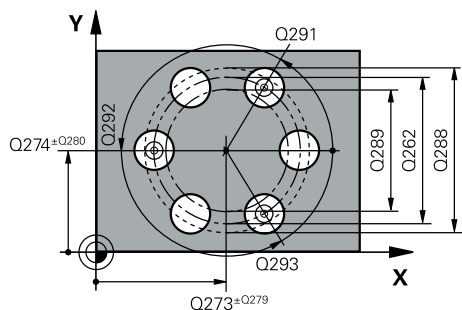
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- Cycle **430** only monitors for tool breakage; there is no automatic tool compensation.
- The control will reset an active basic rotation at the beginning of the cycle.

Note on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.

Cycle parameters

Help graphic



Parameter

Q273 Center in 1st axis (nom. value)?

Bolt hole circle center (nominal value) in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q274 Center in 2nd axis (nom. value)?

Bolt hole circle center (nominal value) in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q262 Nominal diameter?

Enter the diameter of the hole.

Input: **0...99999.9999**

Q291 Polar coord. angle of 1st hole?

Polar coordinate angle of the first hole center in the working plane. This value has an absolute effect.

Input: **-360.000...+360.000**

Q292 Polar coord. angle of 2nd hole?

Polar coordinate angle of the second hole center in the working plane. This value has an absolute effect.

Input: **-360.000...+360.000**

Q293 Polar coord. angle of 3rd hole?

Polar coordinate angle of the third hole center in the working plane. This value has an absolute effect.

Input: **-360.000...+360.000**

Q261 Measuring height in probe axis?

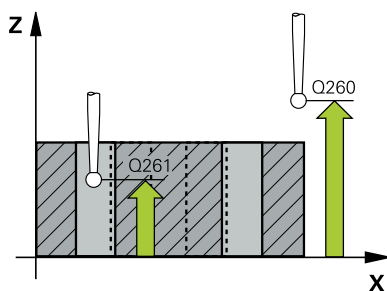
Coordinate of the ball tip center in the touch probe axis in which the measurement will be performed. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** (or **PREDEF**)



Help graphic	Parameter
	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
	Q280 Tolerance for center 2nd axis? Permissible position deviation in the secondary axis of the working plane. Input: 0...99999.9999
	Q281 Measuring log (0/1/2)? Define whether the control will create a measuring log: 0: Do not create a measuring log 1: Create a measuring log: The control will save the log file named TCHPR430.TXT in the folder that also contains the associated NC program 2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with NC Start . Input: 0, 1, 2
	Q309 PGM stop if tolerance exceeded? Define whether in the event of a violation of tolerance limits the control will interrupt program run and output an error message: 0: Do not interrupt program run; no error message 1: Interrupt program run and output an error message Input: 0, 1
	Q330 Tool for monitoring? Define whether the control should perform tool monitoring: 0: Monitoring not active > 0: Number or name of the tool used for machining. Via 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 Further information: "Tool monitoring", Page 321

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	;WRITE LOG FILE ~
Q309=+0	;NO ERROR REACTION ~
Q330=+0	;TOOL

8.6.13 Cycle 431 MEASURE PLANE (#17 / #1-05-1)

ISO programming

G431

Application

Touch probe cycle **431** finds the angles of a plane by measuring three points. It saves the measured values in the Q parameters.



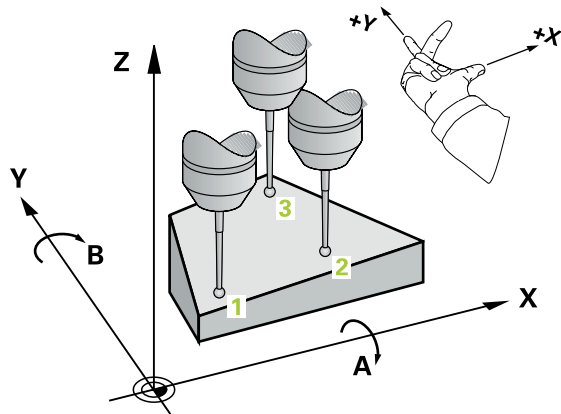
Instead of Cycle **431 MEASURE PLANE**, HEIDENHAIN recommends using the more powerful Cycle **1420 PROBING IN PLANE**.

Related topics

- Cycle **1420 PROBING IN PLANE**

Further information: "Cycle 1420 PROBING IN PLANE (#17 / #1-05-1)", Page 200

Cycle sequence



- 1 The control positions the touch probe to the programmed touch point **1**, using positioning logic and measures the first plane point there. The control offsets the touch probe by the set-up clearance in the direction opposite to the direction of probing.
Further information: "Positioning logic", Page 88
- 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 When finished, the control returns the touch probe to the clearance height and saves the measured angle values in the Q parameters listed below:

Q parameter number	Meaning
Q158	Projection angle of the A axis
Q159	Projection angle of the B axis
Q170	Spatial angle A
Q171	Spatial angle B
Q172	Spatial angle C
Q173 to Q175	Measured values in the touch probe axis (first to third measurement)

Notes

NOTICE

Risk of collision!

If you save the angle values in the preset table and then tilt the tool by programming **PLANE SPATIAL** with **SPA** = 0; **SPB** = 0; **SPC** = 0, there are multiple solutions in which the tilting axes are at 0. There is a risk of collision!

- Make sure to program **SYM (SEQ)** + or **SYM (SEQ)** -

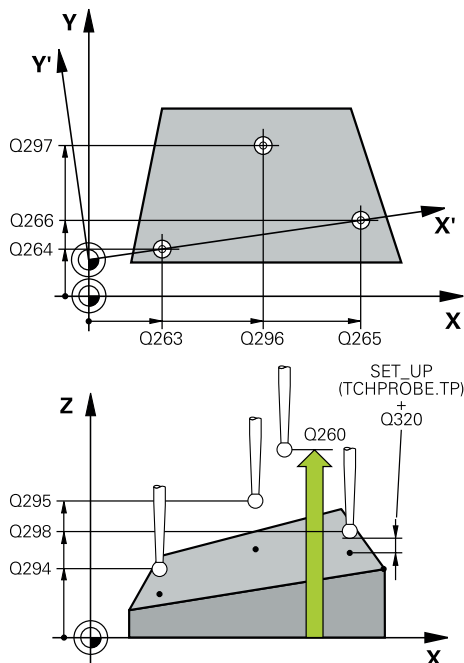
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- The control can calculate the angle values only if the three measuring points are not positioned on a straight line.
- The control will reset an active basic rotation at the beginning of the cycle.

Notes on programming

- Before defining this cycle, you must have programmed a tool call to define the touch probe axis.
- The spatial angles that are needed for the **Tilt working plane** function are saved in parameters **Q170** to **Q172**. With the first two measuring points, you also specify the direction of the main axis when tilting the working plane.
- The third measuring point determines the direction of the tool axis. Define the third measuring point in the direction of the positive Y axis to ensure that the position of the tool axis in a clockwise coordinate system is correct.

Cycle parameters

Help graphic



Parameter

Q263 1st measuring point in 1st axis?

Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect.

Input: -99999.9999...+99999.9999

Q264 1st measuring point in 2nd axis?

Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: -99999.9999...+99999.9999

Q294 1st measuring point in 3rd axis?

Coordinate of the first touch point in the touch probe axis. This value has an absolute effect.

Input: -99999.9999...+99999.9999

Q265 2nd measuring point in 1st axis?

Coordinate of the second touch point in the main axis of the working plane. This value has an absolute effect.

Input: -99999.9999...+99999.9999

Q266 2nd measuring point in 2nd axis?

Coordinate of the second touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: -99999.9999...+99999.9999

Q295 2nd measuring point in 3rd axis?

Coordinate of the second touch point in the touch probe axis. This value has an absolute effect.

Input: -99999.9999...+99999.9999

Q296 3rd measuring point in 1st axis?

Coordinate of the third touch point in the main axis of the working plane. This value has an absolute effect.

Input: -99999.9999...+99999.9999

Q297 3rd measuring point in 2nd axis?

Coordinate of the third touch point in the secondary axis of the working plane. This value has an absolute effect.

Input: -99999.9999...+99999.9999

Q298 3rd measuring point in 3rd axis?

Coordinate of the third touch point in the touch probe axis. This value has an absolute effect.

Input: -99999.9999...+99999.9999

Q320 Set-up clearance?

Additional distance between touch point and ball tip. **Q320** is active in addition to the **SET_UP** column in the touch probe table. This value has an incremental effect.

Input: 0...99999.9999 (or **PREDEF**)

Help graphic

Parameter

Q260 Clearance height?

Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect.

Input: **-99999.9999...+99999.9999** (or **PREDEF**)

Q281 Measuring log (0/1/2)?

Define whether the control will create a measuring log:

0: Do not create a measuring log

1: Create a measuring log: The control will save the **log file named TCHPR431.TXT** in the folder that also contains the associated NC program

2: Interrupt program run and display the measuring log on the control screen. Resume the NC program run with **NC Start**.

Input: **0, 1, 2**

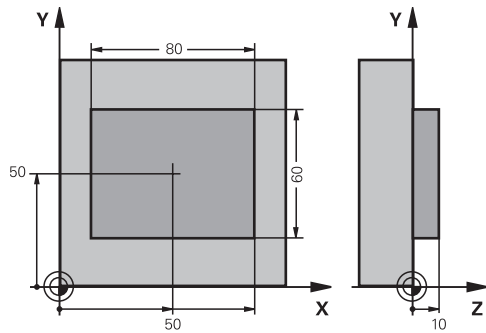
Example

11 TCH PROBE 431 MEASURE PLANE ~	
Q263=+20	;1ST POINT 1ST AXIS ~
Q264=+20	;1ST POINT 2ND AXIS ~
Q294=-10	;1ST POINT 3RD AXIS ~
Q265=+50	;2ND PNT IN 1ST AXIS ~
Q266=+80	;2ND PNT IN 2ND AXIS ~
Q295=+0	;2ND PNT IN 3RD AXIS ~
Q296=+90	;3RD PNT IN 1ST AXIS ~
Q297=+35	;THIRD POINT 2ND AXIS ~
Q298=+12	;3RD PNT IN 3RD AXIS ~
Q320=+0	;SET-UP CLEARANCE ~
Q260=+5	;CLEARANCE HEIGHT ~
Q281=+1	;WRITE LOG FILE

8.6.14 Example: Measuring and reworking a rectangular stud

Program sequence

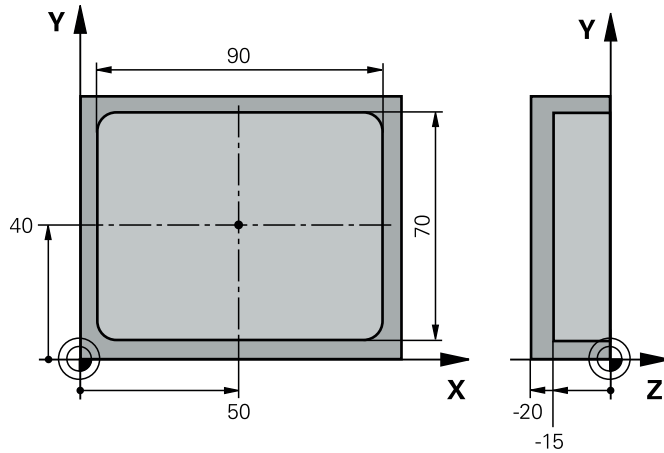
- Rough the rectangular stud with 0.5 mm finishing allowance
- Measure the rectangular stud
- Finish the rectangular stud, taking the measured values into account



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 ;TO MEASURING HEIGHT ~	
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 ;NO LOG FILE ~	
Q309=+0 ;NO ERROR REACTION ~	
Q330=+0 ;TOOL	
9 Q1 = Q1 - Q164	; Calculate the length in X based on the measured deviation

10 Q2 = Q2 - Q165	; Calculate the length in Y based on the measured deviation
11 L Z+100 R0 FMAX	; Retract the touch probe
12 TOOL CALL 25 Z S8000	; Tool call: finishing
13 L Z+100 R0 FMAX M3	; Retract the tool
14 CALL LBL 1	; Call the subprogram for machining
15 L Z+100 R0 FMAX	
16 M30	; End of program run
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 MILLING ~	
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 ;RIGHT ~	
Q215=+0 ;ROUGHING AND FINISHING ~	
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	

8.6.15 Example: Probing a rectangular pocket and recording the results



0 BEGIN PGM TOUCHPROBE_2 MM	
1 TOOL CALL 600 Z	; Tool call: touch probe
2 L Z+100 R0 FMAX	; Retract the touch probe
3 TCH PROBE 423 MEAS. RECTAN. INSIDE ~	
Q273=+50 ;CENTER IN 1ST AXIS ~	
Q274=+40 ;CENTER IN 2ND AXIS ~	
Q282=+90 ;FIRST SIDE LENGTH ~	
Q283=+70 ;2ND SIDE LENGTH ~	
Q261=-5 ;MEASURING HEIGHT ~	
Q320=+2 ;SET-UP CLEARANCE ~	
Q260=+20 ;CLEARANCE HEIGHT ~	
Q301=+0 ;TO MEASURING HEIGHT ~	
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 ;WRITE LOG FILE ~	
Q309=+0 ;NO ERROR REACTION ~	
Q330=+0 ;TOOL	
4 L Z+100 R0 FMAX	; Retract the tool
5 M30	; End of program run
6 END PGM TOUCHPROBE_2 MM	

8.7 Probing a position in the plane or in space (#17 / #1-05-1)

8.7.1 Cycle 3 MEASURING (#17 / #1-05-1)

ISO programming

NC syntax is available only in Klartext programming.

Application

Touch probe cycle **3** measures any position on the workpiece in a selectable probing direction. Unlike other touch probe cycles, Cycle **3** enables you to enter the measuring range **SET UP** and feed rate **F** directly. Also, the touch probe retracts by a definable value **MB** after determining the measured value.

Cycle sequence

- 1 The touch probe moves from the current position at the specified feed rate in the defined probing direction. Use polar angles to define the probing direction in the cycle.
- 2 After the control has saved the position, the touch probe stops. The control saves the X, Y, Z coordinates of the probe-tip center in three successive Q parameters. The control does not conduct any length or radius compensations. You define the number of the first result parameter in the cycle.
- 3 Finally, the control retracts the touch probe by the value that you defined in parameter **MB** in the direction opposite to the probing direction.

Return parameters

Q parameter number	Meaning
Q1*	First measured position in the X axis
Q2*	First measured position in the Y axis
Q3*	First measured position in the Z axis
Q4*	Result ■ 0: Valid probing result ■ -1: No touch point found, stylus not deflected ■ -2: Stylus already deflected at beginning of cycle

*The number of the Q parameter can deviate from this example. You define the number of the first result parameter in the cycle under **3.1**. The further results are in the immediately following Q parameters.

Notes



The exact behavior of touch probe cycle **3** is defined by your machine manufacturer or a software manufacturer who uses it within specific touch probe cycles.

- This cycle can be executed only in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The **DIST** (maximum traverse to touch point) and **F** (probing feed rate) touch-probe data, which are effective in other touch probe cycles, do not apply in touch probe cycle **3**.
- Remember that the control always writes to four successive Q parameters.
- If the control was not able to determine a valid touch point, the NC program is run without an error message. In this case the control assigns the value -1 to the fourth result parameter so that you can deal with the error yourself.
- The control retracts the touch probe by at most the retraction distance **MB**, but not beyond the starting point of the measurement. This rules out any collision during retraction.



The **FN 17: SYSWRITE ID990 NR6** function allows setting whether the cycle runs through the probe input X12 or X13.

Cycle parameters

Help graphic	Parameter
	Parameter number for result? Enter the number of the Q parameter to which you want the control to assign the first measured coordinate (X). The Y and Z values, as well as the reaction, will be written to the immediately following Q parameters. Input: 0...1999 Further information: "Return parameters", Page 374
	Probing axis? Enter the axis in whose direction the touch probe will move and confirm with the ENT key. Input: X, Y, or Z
	Probing angle? This angle defines the probing direction. The angle refers to the probe axis. Confirm with the ENT key. Input: -180...+180
	Maximum measuring range? Enter the maximum distance from the starting point by which the touch probe will move. Confirm with ENT . Input: 0...999999999
	Feed rate measurement Enter the measuring feed rate in mm/min. Input: 0...3000

Help graphic**Parameter****Maximum retraction distance?**

Traverse path in the direction opposite to the probing direction, after the stylus was deflected. The control returns the touch probe to a point no farther than the starting point, so that there can be no collision.

Input: **0...999999999**

Reference system? (0=ACT/1=REF)

Define whether the probing direction and measurement result will be referenced to the current coordinate system (**ACT**, can be shifted or rotated) or the machine coordinate system (**REF**):

0: Perform the probing operation in the current system and save the measurement result in the **ACT** system

1: Perform the probing operation in the machine-based REF system. Save the measurement result in the REF system.

Input: **0, 1**

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 **-2** 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

8.7.2 Cycle 4 MEASURING IN 3-D (#17 / #1-05-1)

ISO programming

NC syntax is available only in Klartext programming.

Application

Touch probe cycle **4** measures any position on the workpiece in the probing direction defined by a vector. Unlike other touch probe cycles, Cycle **4** enables you to enter the probing distance and probing feed rate directly. You can also define the distance by which the touch probe retracts after acquiring the probed value.

Cycle **4** is an auxiliary cycle that can be used for probing with any touch probe (TS or TT). The control does not provide a cycle for calibrating the TS touch probe in any probing direction.

Cycle sequence

- 1 The control moves the touch probe from the current position at the entered feed rate in the defined probing direction. Define the probing direction in the cycle by using a vector (delta values in X, Y and Z).
- 2 After the control has saved the position, the control stops the probe movement. The control saves the X, Y, Z coordinates of the probing position in three successive Q parameters. You define the number of the first parameter in the cycle. If you are using a TS touch probe, the probe result is corrected by the calibrated center offset.
- 3 Finally, the control retracts the touch probe in the direction opposite to the direction of probing. You define the traverse distance in the **MB** parameter; 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 program run! There is a danger of collision!

- Make sure that all touch points can be reached.

- This cycle can be executed only in the **FUNCTION MODE MILL** and **FUNCTION MODE TURN** machining modes.
- The control retracts the touch probe by at most the retraction distance **MB**, but not beyond the starting point of the measurement. This rules out any collision during retraction.
- Remember that the control always writes to four successive Q parameters.

Cycle parameters

Help graphic	Parameter
	Parameter number for result? Enter the number of the Q parameter to which you want the control to assign the first measured coordinate (X). The Y and Z values, as well as the reaction, will be written to the immediately following Q parameters. Input: 0...1999
	Relative measuring path in X? X component of the direction vector defining the direction in which the touch probe will move. Input: -999999999...+999999999
	Relative measuring path in Y? Y component of the direction vector defining the direction in which the touch probe will move. Input: -999999999...+999999999
	Relative measuring path in Z? Z component of the direction vector defining the direction in which the touch probe will move. Input: -999999999...+999999999
	Maximum measuring range? Enter the maximum distance from the starting point by which the touch probe will move along the direction vector. Input: -999999999...+999999999
	Feed rate measurement Enter the measuring feed rate in mm/min. Input: 0...3000
	Maximum retraction distance? Traverse path in the direction opposite the probing direction, after the stylus was deflected. Input: 0...999999999
	Reference system? (0=ACT/1=REF) Define whether the result of probing will be saved in the input coordinate system (ACT), or with respect to the machine coordinate system (REF): 0: Save the measurement result in the ACT system 1: Save the measurement result in the REF system Input: 0, 1

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

8.7.3 Cycle 444 PROBING IN 3-D (#17 / #1-05-1)

ISO programming

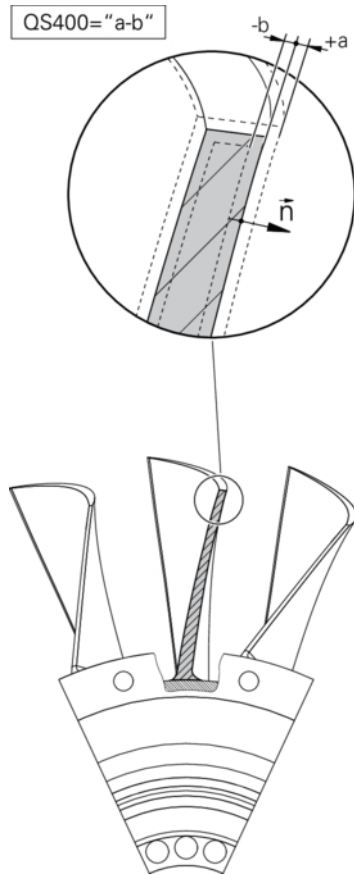
G444

Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.



Cycle **444** checks one specific point on the surface of a part. This cycle is used, for example, to measure free-form surfaces of moldmaking parts. It can be determined whether a point on the surface of the part lies in an undersize or oversize range compared to a nominal coordinate. The operator can subsequently perform further machining steps, such as reworking.

Cycle **444** probes any point in three dimensions and determines the deviation from a nominal coordinate. A normal vector, defined in parameters **Q581**, **Q582**, and **Q583**, is used for this purpose. The normal vector is perpendicular to an imagined surface in which the nominal coordinate is located. The normal vector points away from the surface and does not determine the probing path. It is advisable to determine the normal vector with the help of a CAD or CAM system. A tolerance range **QS400** defines the permissible deviation between the actual and nominal coordinate along the normal vector. This way you define, for example, that the program is to be interrupted if an undersize is detected. Additionally, the control outputs a log and the deviations are stored in the Q parameters listed below.

Cycle 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 86
- 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 **Q320** in the direction opposite to the probing direction.

Result parameters

The control stores the probing results in the following parameters:

Q parameter number	Meaning
Q151	Measured position in main axis
Q152	Measured position in secondary axis
Q153	Measured position in tool axis
Q161	Measured deviation in main axis
Q162	Measured deviation in secondary axis
Q163	Measured deviation in tool axis
Q164	Measured 3D deviation ■ Less than 0: Undersize ■ Greater than 0: Oversize
Q183	Workpiece status: ■ - 1 = undefined ■ 0 = good ■ 1 = Rework ■ 2 = Scrap

Log function

Once probing has finished, the control generates a log in HTML format. The log includes the results from the main, secondary, and tool axes as well as the 3D error. The control saves the log in the same folder in which the *.h file is located (as long as no path has been configured for **FN 16**).

The log contains the following data on the main, secondary, and tool axes:

- Actual probing direction (as a vector in the input system). The value of the vector corresponds to the configured probing path.
- Defined nominal coordinate
- If a tolerance **QS400** was defined: Upper and lower dimensions are output, as well as the determined deviation along the normal vector
- Ascertained actual coordinate
- Colored display of the values (green for "good," orange for "rework," red for "scrap")

Notes

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- In order to obtain exact results from the touch probe being used, you need to perform 3D calibration before executing Cycle **444**. 3D calibration requires 3D-ToolComp.
- Cycle **444** generates a measuring log in HTML format.
- An error message is output if Cycle **8 MIRRORING**, Cycle **11 SCALING FACTOR**, or Cycle **26 AXIS-SPECIFIC SCALING** is active before Cycle **444** is run.
- For probing, an active TCPM will be taken into account. While the TCPM is active, probing of positions is possible even if the position resulting from the **Tilt working plane** function is inconsistent with the current position of the rotary axes.
- If your machine is equipped with a feedback-controlled spindle, you should activate angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3D touch probe.
- Cycle **444** references all coordinates to the input system.
- The control writes the measured values to return parameters.

Further information: "Application", Page 379

- The workpiece status good/rework/scrap is set via Q parameter **Q183**, independent of parameter **Q309**.

Further information: "Application", Page 379

Notes 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.

Cycle parameters

Help graphic	Parameter
	Q263 1st measuring point in 1st axis? Coordinate of the first touch point in the main axis of the working plane. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q264 1st measuring point in 2nd axis? Coordinate of the first touch point in the secondary axis of the working plane. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q294 1st measuring point in 3rd axis? Coordinate of the first touch point in the touch probe axis. This value has an absolute effect. Input: -99999.9999...+99999.9999
	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 active in addition to the SET_UP column in the touch probe table. This value has an incremental effect. Input: 0...99999.9999 (or PREDEF)
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect. Input: -99999.9999...+99999.9999 (or PREDEF)

Help graphic**Parameter****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	;NO ERROR REACTION

8.8 Influencing cycle runs (#17 / #1-05-1)

8.8.1 Cycle 441 FAST PROBING (#17 / #1-05-1)

ISO programming

G441

Application

You can use touch probe cycle **441** to globally specify various touch probe parameters (e.g., the positioning feed rate) for all subsequently used touch probe cycles.



In this cycle, no machine movements will be performed.

Program interruption Q400=1

Parameter **Q400 INTERRUPTION** allows interrupting the cycle run and displaying the obtained results.

Program interruption by **Q400** is effective in the following touch probe cycles:

- Touch probe cycles for checking the workpiece: **421** to **427**, **430** and **431**
- Cycle **444 PROBING IN 3-D**
- Touch probe cycles for measuring the kinematics: **45x**
- Touch probe cycles for calibrating: **46x**
- Touch probe cycles **14xx**

Cycles 421 to 427, 430 and 431:

The control displays the results obtained during a program interruption in an **FN 16** monitor output.

Cycles 444, 45x, 46x, 14xx:

The control automatically shows the results obtained during a program interruption in an HTML log in the path: **TNC:\TCHPRlast.html**. You can open the HTML log in the **Document** workspace.

Notes

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- **END PGM, M2, M30** reset the global settings of Cycle **441**.
- Cycle parameter **Q399** depends on your machine configuration. Your machine manufacturer is responsible for the setting of whether the touch probe can be oriented through an NC program.
- Even if your machine has separate potentiometers for rapid traverse and feed rate, you can control the feed rate with the feed rate potentiometer only, even with **Q397=1**.
- If **Q371** is unequal to **0** and the stylus does not move in cycles **14xx**, the control will terminate the cycle. The control returns the touch probe to the clearance height and saves the workpiece status **3** in Q parameter **Q183**. The NC program continues.
Workpiece status **3**: Stylus does not move
- If you execute this cycle in combination with Cycles **42x** or **43x** and want to output a measuring log to the screen, you must program **Q400=1**. Otherwise the control will not interrupt and the measuring log will not be displayed on the screen.
- HEIDENHAIN recommends activating angle tracking by setting **Q399=1** in order to avoid measuring inaccuracies in the probing results.

Note regarding machine parameters

- The machine parameter **maxTouchFeed** (no. 122602) allows the machine manufacturer to limit the feed rate. You define the maximum absolute feed rate in this machine parameter.

Cycle parameters

Help graphic	Parameter
	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

Help graphic	Parameter
	Q400 Automatic interruption? Define whether the control will interrupt program run and output the measurement results on the screen following a touch probe cycle: 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 Further information: "Program interruption Q400=1", Page 385
	Q371 Touch point not reached? (optional) Define how the control behaves when the stylus does not move within the DIST value of the touch probe table. 0: The control interrupts the NC program with an error message saying that the touch point cannot be reached. This is standard behavior. 1: The control displays a warning and terminates the touch probe cycle. The NC program continues. Is effective only in the 14xx cycles. 2: The control displays no warning and terminates the touch probe cycle. The NC program continues. Is effective only in the 14xx cycles. Input: 0, 1, 2

Example

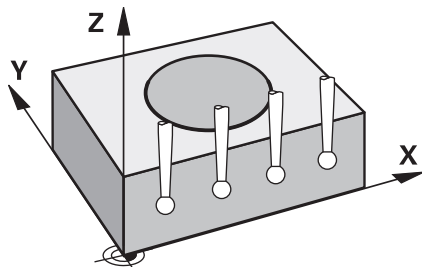
11 TCH PROBE 441 FAST PROBING ~	
Q396=+3000	;POSITIONING FEEDRATE ~
Q397=+0	;POSITIONING FEED RATE ~
Q399=+1	;PROBE ORIENTATION ACTIVE ~
Q400=+1	;STOP UPON SCREEN OUTPUT ~
Q371=+0	;ABORT

8.8.2 Cycle 1493 EXTRUSION PROBING (#17 / #1-05-1)

ISO programming

G1493

Application



Cycle **1493** allows you to repeat the touch points of specific touch probe cycles along a straight line. In the cycle, you define the direction and the length of the extrusion, as well as the number of extrusion points.

The repetitions allow you, for example, to perform multiple measurements at different heights and to determine deviations based on the deflection of the tool. You can also use the extrusion to increase the accuracy during probing. Multiple measuring points help you ascertain contamination on the workpiece or rough surfaces.

In order to activate the repetition of specific touch points, you need to define Cycle **1493** before the touch probe cycle. Depending on the definition, this cycle will remain active for only the next cycle or for the entire NC program. The control interprets the extrusion in the input coordinate system **I-CS**.

The following cycles are capable of performing extrusions:

- **PROBING IN PLANE** (Cycle **1420**, ISO: **G1420**) (#17 / #1-05-1), see Page 200
- **PROBING ON EDGE** (Cycle **1410**, ISO: **G1410**) (#17 / #1-05-1), see Page 170
- **PROBING TWO CIRCLES** (Cycle **1411**, ISO: **G1411**) (#17 / #1-05-1), see Page 177
- **INCLINED EDGE PROBING** (Cycle **1412**, ISO: **G1412**) (#17 / #1-05-1), see Page 185
- **INTERSECTION PROBING** (Cycle **1416**, DIN/ISO: **G1416**) (#17 / #1-05-1), see Page 192
- **POSITION PROBING** (Cycle **1400**, ISO: **G1400**) (#17 / #1-05-1), see Page 278
- **CIRCLE PROBING** (Cycle **1401**, ISO: **G1401**) (#17 / #1-05-1), see Page 282
- **PROBE SLOT/RIDGE** (Cycle **1404**, ISO: **G1404**) (#17 / #1-05-1), see Page 298
- **PROBE POSITION OF UNDERCUT** (Cycle **1430**, ISO: **G1430**) (#17 / #1-05-1), see Page 304
- **PROBE SLOT/RIDGE UNDERCUT** (Cycle **1434**, ISO: **G1434**) (#17 / #1-05-1), see Page 310

Result parameter Q

The control saves the results of the touch probe cycle in the following Q parameters:

Q parameter number	Meaning
Q970	Maximum deviation of the first touch point position
Q971	Maximum deviation of the second touch point position
Q972	Maximum deviation of the third touch point position
Q973	Maximum deviation of diameter 1
Q974	Maximum deviation of diameter 2
Q975	Maximum deviation of the width

Result parameter QS

The control saves the individual results of all measuring points of an extrusion in the QS parameters **QS97x**. The result is ten characters long. The results are separated from each other by a space.

Example: **QS970 = 0.12345678 -0.1234567 -0.1134567 0.11234567**

QS parameter number	Meaning
QS970	Deviation of the position of the first probed object of an extrusion
QS971	Deviation of the position of the second probed object of an extrusion
QS972	Deviation of the position of the third probed object of an extrusion
QS973	Deviations of diameter 1
QS974	Deviations of diameter 2
QS975	Deviations of the width

You can convert the individual results in the NC program, using string processing into numerical values and use them in evaluations, for example.

Example:

A touch probe cycle produces the following results within QS parameter **QS970**:

QS970 = 0.12345678 -0.1234567

The example below shows how to convert the results produced into numerical values.

11 QS0 = SUBSTR (SRC_QS970 BEG0 LEN10)	; Read out the first result from QS970
12 QL1 = TONUMB (SRC_QS0)	; Convert alphanumeric value from QS0 to a numerical value and assign it to QL0
13 QS0 = SUBSTR (SRC_QS970 BEG11 LEN10)	; Read out the second result from QS970
14 QL2 = TONUMB (SRC_QS0)	; Convert alphanumeric value from QS0 to a numerical value and assign it to QL2

Further information: Programming and Testing User's Manual

Log function

Once probing has finished, the control generates a log file in HTML format. The log file contains the results of the 3D deviation in graphical and tabular form. The control saves the log file in the same folder in which the NC program is located.

The log file contains the following data in the main axis, secondary axis and tool axis depending on the selected cycle (e.g., circle center point and diameter):

- Actual probing direction (as a vector in the input system). The value of the vector corresponds to the configured probing path
- Defined nominal coordinate
- Upper and lower dimensions, as well as the determined deviation along the normal vector
- Measured actual coordinate
- Color coding of the values:
 - Green: Pass
 - Orange: Rework
 - Red: Scrap
- Extrusion points:

The horizontal axis represents the direction for the extrusion. The blue points are the individual measuring points. The red lines indicate the lower limit and the upper limit of the dimensions. If a value violates a specified tolerance, the control will show the area in red color in the graphic.

Notes

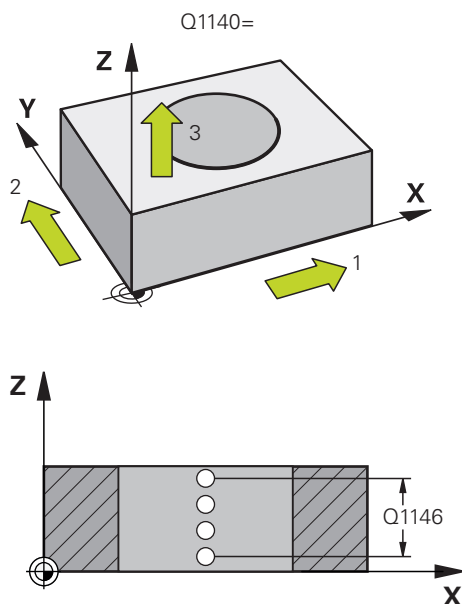
- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- If **Q1145 > 0** and **Q1146 = 0**, then the control will perform the number of extrusion points at the same position.
- If you use Cycle **1401 CIRCLE PROBING** or Cycle **1411 PROBING TWO CIRCLES** to perform an extrusion, the direction for the extrusion must be **Q1140=+3**; otherwise, the control will output an error message.
- If you use Cycle **1404 PROBE SLOT/RIDGE** to perform an extrusion, the direction for the extrusion in the main axis must be **Q1140=+1**, or in the tool axis it must be **Q1140=+3**; otherwise, the control will output an error message.
- When defining the **TRANSFER POSITION Q1120>0** within a touch probe cycle, the control will correct the preset by the mean of deviations. The control calculates this mean from all measured extrusion points of the probing object according to the programmed **TRANSFER POSITION Q1120**.

Example:

- Nominal position of touch point 1: 2.35 mm
- Results: **QS970** = 2.30000000 2.35000000 2.40000000 2.50000000
Mean: 2.38750000 mm
The preset is corrected by the mean from the nominal position, in this case by 0.0375 mm.

Cycle parameters

Help graphic



Parameter

Q1140 Direction for extrusion (1-3)?

- 1: Extrusion in the direction of the main axis
- 2: Extrusion in the direction of the secondary axis
- 3: Extrusion in the direction of the tool axis

Input: 1, 2, 3

Q1145 Number of extrusion points?

Number of measuring points that the cycle repeats over the length of the extrusion Q1146.

Input: 1...99

Q1146 Length of extrusion?

Length over which the measuring points are repeated.

Input: -9999.99...+9999.99

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.

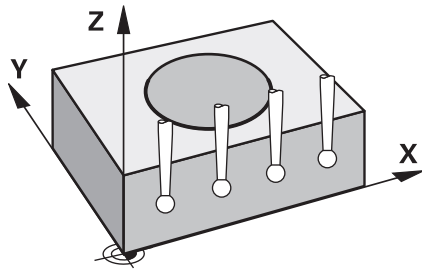
Input: 0, 1

Example

11 TCH PROBE 1493 EXTRUSION PROBING ~	
Q1140=+3	;TOOL AXIS ~
Q1145=+1	;EXTRUSION POINTS ~
Q1146=+0	;EXTRUSION LENGTH ~
Q1149=+0	;ONLY NEXT PROBING CYCLE

Example

With this example you can ascertain a form deviation, such as can occur when the tool is deflected. To do so you must first define Cycle **1493 EXTRUSION PROBING** and specify whether it applies to just the subsequent cycle or for the entire program. After this cycle you can program any Cycle **14xx**. This enables you to ascertain form deviations on various objects and actively intervene in the process (e.g., exchange tools).



Program sequence

- Cycle **1493 EXTRUSION PROBING**
 - **Q1140=+1**: Extrusion in the direction of the main axis
 - **Q1145=+4**: Number of extrusion points
 - **Q1149=+0**: The extrusion is effective for only the next cycle
- Cycle **1410 POSITION PROBING**
 - **Q372=+2**: Probing direction in minor axis

0 BEGIN PGM TOUCHPROBE MM	
1 TOOL CALL 600 Z	
2 TCH PROBE 1493 EXTRUSION PROBING ~	
Q1140=+1 ;MAIN AXIS ~	
Q1145=+4 ;EXTRUSION POINTS ~	
Q1146=+80 ;EXTRUSION LENGTH ~	
Q1149=+0 ;ONLY NEXT PROBING CYCLE	
3 TCH PROBE 1400 POSITION PROBING ~	
Q1100=+15 ;1ST POINT REF AXIS ~	
Q1101=+0 ;1ST POINT MINOR AXIS ~	
Q1102=-10 ;1ST POINT TOOL AXIS ~	
Q372=+2 ;MINOR AXIS PLUS ~	
Q320=+0 ;SET-UP CLEARANCE ~	
Q260=+50 ;CLEARANCE HEIGHT ~	
Q1125=+2 ;BEFORE AND AFTER PROBING PROCESS ~	
Q309=+0 ;NO ERROR REACTION ~	
Q1120=+0 ;TRANSFER POSITION	
4 CALL PGM 35	; Call the part program
5 END PGM TOUCHPROBE MM	

9

**Touch-probe
cycles for tools
(#17 / #1-05-1)**

9.1 Overview

Measurement of milling cutters

Cycle		Call	Further information
481	CAL. TOOL LENGTH (#17 / #1-05-1) ■ Measuring the tool length	DEF-active	Page 403
482	CAL. TOOL RADIUS (#17 / #1-05-1) ■ Measuring the tool radius	DEF-active	Page 406
483	MEASURE TOOL (#17 / #1-05-1) ■ Measuring the tool length and radius	DEF-active	Page 410

Lathe tool measurement

Cycle		Call	Further information
485	MEASURE LATHE TOOL (#17 / #1-05-1) ■ Measurement of turning tools	DEF-active	Page 414

9.2 Conditional stops in touch probe cycles

If your machine has an override controller, you can activate conditional stops during program run. If you activate conditional stops with the **In cycle call** selection, the control does not interrupt program run during any touch probe cycles.

Further information: User's Manual for Setup and Program Run

9.3 Fundamentals

9.3.1 Application

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

Related topics

- Calibrate the tool touch probe

Further information: "Calibrating a tool touch probe (#17 / #1-05-1)", Page 117

9.3.2 Measuring a tool of length 0



Refer to your machine manual!

The optional machine parameter **maxToolLengthTT** (no. 122607) enables the machine manufacturer to define a maximum tool length for the tool measurement cycles.



HEIDENHAIN recommends that you always define tools with their actual tool length if possible.

The tool measuring cycles measure tools automatically. You can also measure tools defined with a length **L** of 0 in the tool table. To do this, the machine manufacturer must define a maximum tool length value in the optional machine parameter **maxToolLengthTT** (no. 122607). The control starts a search in which the actual tool length is roughly determined in the first step. This is followed by a fine measurement.

Cycle sequence

- 1 The tool travels to a clearance height centered above the touch probe.
The clearance height equals the value of the optional machine parameter **maxToolLengthTT** (no. 122607).
- 2 The control performs a rough measurement with the spindle standing still.
When measuring a stationary tool, the control will use the feed rate for probing defined in the machine parameter **probingFeed** (no. 122709).
- 3 The control saves the roughly measured length.
- 4 The control performs a fine measurement with the values from the tool measuring cycle.

Notes

NOTICE**Risk of collision!**

If the machine manufacturer fails to define the optional machine parameter **maxToolLengthTT** (no. 122607), there will be no tool search. The control pre-positions the tool with a length of 0. Risk of collision!

- ▶ Observe the machine parameter value in the machine manual.
- ▶ Define tools with the actual tool length **L**

NOTICE**Risk of collision!**

Risk of collision if the tool is longer than the value of the optional machine parameter **maxToolLengthTT** (no. 122607)!

- ▶ Observe the machine parameter value in the machine manual

9.3.3 Setting machine parameters

- The touch probe cycles **480, 481, 482, 483, 484** can be hidden with the optional machine parameter **hideMeasureTT** (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).

Spindle speed setting

When measuring a rotating tool, the control automatically calculates the spindle speed and feed rate for probing.

The spindle speed is calculated as follows:

$$n = \text{maxPeriphSpeedMeas} / (r \cdot 0.0063) \text{ where}$$

Abbreviation	Definition
n	Rotational speed [rpm]
maxPeriphSpeedMeas (No. 122712)	Maximum permissible cutting speed in m/min
r	Active tool radius [mm]

Setting of the feed rate

The probing feed rate is calculated as follows:

$$v = \text{measuring tolerance} \cdot n$$

Abbreviation	Definition
v	Probing feed rate [mm/min]
Measuring tolerance	Measuring tolerance [mm], depending on maxPeriphSpeedMeas (no. 122712)
n	Shaft speed [rpm]

probingFeedCalc (no. 122710) determines the calculation of the probing feed rate.

The control provides the following options:

- **ConstantTolerance**
- **VariableTolerance**
- **ConstantFeed**

ConstantTolerance:

The measuring tolerance remains constant—regardless of the tool radius. With very large tools, however, the feed rate for probing is reduced to zero. The lower you set the maximum permissible rotational speed (**maxPeriphSpeedMeas** (no. 122712)) and the permissible tolerance (**measureTolerance1** (no. 122715)), the sooner you will encounter this effect.

VariableTolerance:

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	$4 \cdot \text{measureTolerance1}$

ConstantFeed:

The measuring feed rate remains constant; the measuring error, however, rises linearly with the increase in tool radius:

$$\text{Measuring tolerance} = (r \cdot \text{measureTolerance1}) / 5 \text{ mm) where}$$

Abbreviation	Definition
r	Active tool radius [mm]
measureTolerance1	Maximum permissible error of measurement

Setting for consideration of parallel axes and changes in the kinematics

Refer to your machine manual.

Using the optional machine parameter **calPosType** (no. 122606), the machine manufacturer defines whether the position of parallel axes and changes in the kinematics should be considered for calibration and measuring. A change in kinematics might for example be a head change.

Auxiliary or parallel axes cannot be probed, regardless of the setting of the optional machine parameter **calPosType** (no. 122606).

If the machine manufacturer changes the setting of the optional machine parameter, you need to recalibrate the tool touch probe.

9.3.4 Entries in the tool table for milling tools

Abbr.	Inputs	Dialog
CUT	The number of teeth of the tool for automatic tool measurement or cutting data calculation (maximum of 20 teeth)	Number of teeth?
LTOL	Permitted tool length deviation in wear detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column TL (status L). Input: 0.0000...5.0000	Wear tolerance: length?
RTOL	Permitted tool radius deviation in wear detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column TL (status L). Input: 0.0000...5.0000	Wear tolerance: radius?
DIRECT.	Cutting direction of the tool for automatic tool measurement with a rotating tool. Input: -, +	Cutting direction (M3 = -)?
R-OFFS	Position of tool upon length measurement, offset between the probe contact center and the tool center for automatic tool measurement. Default setting: No value entered (offset = tool radius) Input: -99999.9999...+99999.9999	Tool offset: radius?
L-OFFS	Position of tool upon radius measurement, distance between the probe contact top edge and the tool tip for automatic tool measurement. Is added to the offsetToolAxis machine parameter (no. 122707). Input: -99999.9999...+99999.9999	Tool offset: length?
LBREAK	Permitted tool length deviation in breakage detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column TL (status L). Input: 0.0000...9.0000	Breakage tolerance: length?

Abbr.	Inputs	Dialog
RBREAK	Permitted tool radius deviation in breakage detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column TL (status L). Input: 0.0000...9.0000	Breakage tolerance: radius?

Input examples for common tool types

Tool type	CUT	R-OFFS	L-OFFS
Drill	No function	0: No offset required because tool tip is to be measured	
End mill	4: four cutting edges	R: Offset required because the tool diameter is greater than the contact plate diameter of the TT	0: No additional offset required during radius measurement. Offset from offsetToolAxis (no. 122707) used.
Spherical cutter with a diameter of 10 mm	4: four cutting edges	0: No offset required because the south pole of the ball is to be measured.	5: At a diameter of 10 mm, the tool radius will be defined as offset. If this is not the case, the diameter of the spherical cutter will be measured too far down. So the tool diameter will not be correct.

9.4 Measurement of milling cutters (#17 / #1-05-1)

9.4.1 Cycle 481 CAL. TOOL LENGTH (#17 / #1-05-1)

ISO programming

G481

Application



Refer to your machine manual!

For measuring the tool length, program touch probe cycle **481**. 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.

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 be executed only 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**.
- You can use the **CAL_PIN** (calibration pin) tool type only for comparing measured values. To do so, program **Q340** with the value **2**.

Cycle parameters

Help graphic	Parameter
	Q340 Tool measurement mode (0-2)? Define whether and how the measured data will be entered in the tool table. 0: The measured tool length is written to column L of tool table TOOL.T, and the tool compensation is set to DL = 0. If there is already a value in TOOL.T, it will be overwritten. 1: The measured tool length is compared to the tool length L from TOOL.T. The control calculates the deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation is also available in the Q parameter Q115. If the delta value is greater than the permissible tool length tolerance for wear or break detection, the control will lock the tool (status L in TOOL.T). 2: The measured tool length is compared to the tool length L from TOOL.T. The control calculates the deviation from the stored value and writes it to Q parameter Q115. Nothing is entered under L or DL in the tool table. Input: 0, 1, 2
	Q260 Clearance height? Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from safetyDistStylus). Input: -99999.9999...+99999.9999
	Q341 Probe the teeth? 0=no/1=yes Define whether the control will measure the individual teeth (maximum of 20 teeth) Input: 0, 1

Example

11 TOOL CALL 12 Z	
12 TCH PROBE 481 CAL. TOOL LENGTH ~	
Q340=+1	;ENTER DELTA VALUES ~
Q260=+100	;CLEARANCE HEIGHT ~
Q341=+1	;MEASURE INDIVIDUAL TEETH

9.4.2 Cycle 482 CAL. TOOL RADIUS (#17 / #1-05-1)

ISO programming

G482

Application



Refer to your machine manual!

If you want to measure the tool radius, program the touch probe cycle **482**. Select via input parameters by which of two methods the tool radius is to be measured:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth

The control pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the face of the milling tool to the upper edge of the touch probe head is defined in **offsetToolAxis** (no. 122707). The control probes the tool radially while it is rotating.

If you have programmed a subsequent measurement of individual teeth, the control will measure the radius of each tooth with the aid of oriented spindle stops.

Further information: "Notes for individual tooth measurement Q341=1", Page 407

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 be executed only 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 use the **CAL_PIN** (calibration pin) tool type only for comparing measured values. To do so, program **Q340** with the value **2**.

Note regarding machine parameters

- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual tooth measurements.
- Cylindrical tools with diamond surfaces can be measured while the spindle is stationary. To do so, in the tool table define the number of teeth **CUT** as 0 and adjust the machine parameter **CfgTT**. Refer to your machine manual.

Notes for individual tooth measurement Q341=1**NOTICE****Caution: Danger to the tool and workpiece!**

Individual tooth measurement of tools with a large angle of twist may result in a failure of the control to identify tool wear or a broken tool. In this case, tool and workpiece damage may result during subsequent machining operations.

- ▶ Check the workpiece dimensions (for example, by using a workpiece touch probe)
- ▶ Check the workpiece optically in order to exclude broken tools

If the maximum angle of twist is exceeded, you should not carry out individual tooth measurement.

On tools with an even distribution of teeth, a maximum angle of twist can be defined as follows:

$$\varepsilon = 90 - \operatorname{atan} \left(\frac{h[tt]}{\frac{R \times 2 \times \pi}{x}} \right)$$

Abbreviation	Definition
ε	Maximum angle of twist
h[tt]	Height of tool touch probe contact
R	Tool radius
x	Number of teeth of tool



On tools with an uneven distribution of teeth, there is no calculation formula for the maximum angle of twist. Check these tools optically in order to exclude breaks. You can measure wear indirectly by measuring the workpiece.

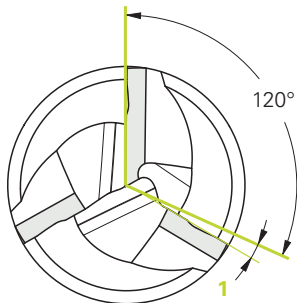
NOTICE**Caution: Possible material damage!**

Individual tooth measurement of tools with an uneven distribution of teeth may cause the control to identify non-existing wear. The higher the angle deviation and the larger the tool radius, the more probably this behavior can occur. If the control compensates for the tool incorrectly after individual tooth measurement, the workpiece may have to be rejected.

- Check the workpiece dimensions during subsequent machining operations

Individual tooth measurement of tools with an uneven distribution of teeth may cause the control to identify non-existing breakage and lock the tool.

The higher the angle deviation **1** and the larger the tool radius, the more probably this behavior can occur.



1 Angle deviation

Cycle parameters

Help graphic	Parameter
	Q340 Tool measurement mode (0-2)? Define whether and how the measured data will be entered in the tool table. 0: The measured tool radius is written to column R of the TOOL.T tool table, and the tool compensation is set to DR = 0. If there is already a value in TOOL.T, it will be overwritten. 1: The measured tool radius is compared to the tool radius R from TOOL.T. The control calculates the deviation from the stored value and enters it into TOOL.T as the delta value DR. The deviation 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). 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. Input: 0, 1, 2
	Q260 Clearance height? Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from safetyDistStylus). Input: -99999.9999...+99999.9999
	Q341 Probe the teeth? 0=no/1=yes Define whether the control will measure the individual teeth (maximum of 20 teeth) Input: 0, 1

Example

11 TOOL CALL 12 Z	
12 TCH PROBE 482 CAL. TOOL RADIUS ~	
Q340=+1	;ENTER DELTA VALUES ~
Q260=+100	;CLEARANCE HEIGHT ~
Q341=+1	;MEASURE INDIVIDUAL TEETH

9.4.3 Cycle 483 MEASURE TOOL (#17 / #1-05-1)

ISO programming

G483

Application



Refer to your machine manual!

To measure both the length and radius of a tool, program the touch probe cycle **483**. 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 **481** and **482**.

Further information: "Notes for individual tooth measurement of radius Q341=1", Page 411

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 be executed only 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 use the **CAL_PIN** (calibration pin) tool type only for comparing measured values. To do so, program **Q340** with the value **2**.

Note regarding machine parameters

- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual tooth measurements.
- Cylindrical tools with diamond surfaces can be measured while the spindle is stationary. To do so, in the tool table define the number of teeth **CUT** as 0 and adjust the machine parameter **CfgTT**. Refer to your machine manual.

Notes for individual tooth measurement of radius Q341=1**NOTICE****Caution: Danger to the tool and workpiece!**

Individual tooth measurement of tools with a large angle of twist may result in a failure of the control to identify tool wear or a broken tool. In this case, tool and workpiece damage may result during subsequent machining operations.

- ▶ Check the workpiece dimensions (for example, by using a workpiece touch probe)
- ▶ Check the workpiece optically in order to exclude broken tools

If the maximum angle of twist is exceeded, you should not carry out individual tooth measurement.

On tools with an even distribution of teeth, a maximum angle of twist can be defined as follows:

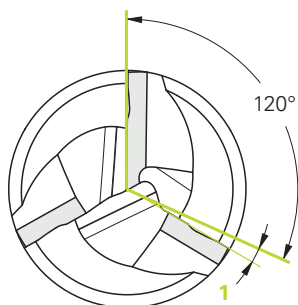
$$\varepsilon = 90 - \operatorname{atan} \left(\frac{h[tt]}{\frac{R \times 2 \times \pi}{x}} \right)$$

Abbreviation	Definition
ε	Maximum angle of twist
h[tt]	Height of tool touch probe contact
R	Tool radius
x	Number of teeth of tool



On tools with an uneven distribution of teeth, there is no calculation formula for the maximum angle of twist. Check these tools optically in order to exclude breaks. You can measure wear indirectly by measuring the workpiece.

The higher the angle deviation **1** and the larger the tool radius, the more probably this behavior can occur.



1 Angle deviation

Cycle parameters

Help graphic	Parameter
	Q340 Tool measurement mode (0-2)? Define whether and how the measured data will be entered in the tool table. 0: The measured tool length and the measured tool radius are written to columns L and R of the TOOL.T tool table, and the tool compensation is set to DL = 0 and DR = 0. If there is already a value in TOOL.T, it will be overwritten. 1: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R in TOOL.T. The control calculates the deviation from the stored value and enters them into TOOL.T as the delta values DL and DR. The deviation is also available in 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). 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. Input: 0, 1, 2
	Q260 Clearance height? Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from safetyDistStylus). Input: -99999.9999...+99999.9999
	Q341 Probe the teeth? 0=no/1=yes Define whether the control will measure the individual teeth (maximum of 20 teeth) Input: 0, 1

Example

11 TOOL CALL 12 Z	
12 TCH PROBE 483 MEASURE TOOL ~	
Q340=+1	;ENTER DELTA VALUES ~
Q260=+100	;CLEARANCE HEIGHT ~
Q341=+1	;MEASURE INDIVIDUAL TEETH

9.5 Measuring turning tools

9.5.1 Cycle 485 MEASURE LATHE TOOL

ISO programming

G485

Application



Refer to your machine manual!

Machine and control must be specially prepared by the machine manufacturer for use of this cycle.

Cycle **485 MEASURE LATHE TOOL** is available for the measurement of turning tools with a tool touch probe from HEIDENHAIN. The probe contact must have a cuboid shape. The control measures the tool in a fixed programmed sequence.

Cycle run

- 1 The control positions the turning tool to the clearance height.
- 2 The turning tool is oriented based on the entries in **TO** and **ORI**.
- 3 The control moves the tool to the measuring position in the main axis; the traverse movement is interpolated in the main and secondary axes.
- 4 Then the turning tool moves to the measuring position in the tool axis.
- 5 The tool is measured. Depending on the definition of **Q340**, either tool dimensions are changed or the tool is locked
- 6 The measuring result is transferred to the result parameter **Q199**.
- 7 After the measurement has been performed, the control positions the tool in the tool axis to the clearance height.

Result parameter Q199:

Result	Meaning
0	Tool dimensions within the tolerance LTOL / RTOL Tool is not locked
1	Tool dimensions outside the tolerance LTOL / RTOL Tool is locked
2	Tool dimensions outside the tolerance LBREAK / RBREAK Tool is locked

The cycle uses the following entries from toolturn.trn:

Abbr.	Entries	Dialog
ZL	Tool length 1 (Z direction)	Tool length 1?
XL	Tool length 2 (X direction)	Tool length 2?
DZL	Delta value of tool length 1 (Z direction), is added to ZL	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 cutter radius into account in turning cycles, and performs tool tip radius compensation	Cutting edge radius?
TO	Tool orientation: From the tool orientation, the control determines the position of the tool cutting edge and, depending on the selected tool type, additional information such as the tool angle direction, position of the tool reference point, etc. This information is necessary, for example, for calculating the tool tip radius compensation, milling cutter radius compensation, plunge angle, etc.	Tool orientation?
ORI	Spindle orientation angle: Angle of the indexable insert to the main axis	Angle of spindle orientation?
TYPE	Type of turning tool: Roughing tool ROUGH , finishing tool FINISH , threading tool THREAD , recessing tool RECESS , button tool BUTTON , recess-turning tool RECTURN	Type of turning tool

Further information: "Tool orientation (TO) that is supported for the following types of turning tools (TYPE)", Page 416

Tool orientation (TO) that is supported for the following types of turning tools (TYPE)

TYPE	Supported TO with possible limitations	Non-supported TO
ROUGH, FINISH	■ 1 ■ 7 ■ 2, only XL ■ 3, only XL ■ 5, only XL ■ 6, only XL ■ 8, only ZL ■ 18	■ 4 ■ 9
BUTTON	■ 1 ■ 7 ■ 2, only XL ■ 3, only XL ■ 5, only XL ■ 6, only XL ■ 8, only ZL	■ 4 ■ 9
RECESS, RECTURN	■ 1 ■ 7 ■ 8 ■ 2 ■ 3, only XL ■ 5, only XL	■ 4 ■ 6 ■ 9
THREAD	■ 1 ■ 7 ■ 8 ■ 2 ■ 3, only XL ■ 5, only XL	■ 4 ■ 6 ■ 9

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 be executed only in the **FUNCTION MODE MILL** machining mode.
- Before you begin the cycle, you must run a **TOOL CALL** with the tool axis **Z**.
- If you define **YL** and **DYL** with a value outside of ± 5 mm, the tool won't reach the tool touch probe.
- The cycle does not support **SPB-INSERT** (angular offset). You must enter the value 0 in **SPB-INSERT**, otherwise the control will generate an error message.

Note regarding machine parameters

- The cycle depends on the optional machine parameter **CfgTTRectStylus** (no. 114300). Refer to your machine manual.

Cycle parameters

Help graphic	Parameter
	Q340 Tool measurement mode (0-2)? Use of the measured values: 0: The measured values are entered in ZL and XL. If values are already entered in the tool table, they will be overwritten. DZL and DXL will be reset to 0. TL will not be changed 1: The measured values ZL and XL are compared with the values from the tool table. These values will not be changed. The control then calculates the deviations of ZL and XL, and enters these in DZL and DXL. If the delta values are larger than the permissible wear or breakage tolerance, the control locks the tool (TL = Tool Locked). In addition, the deviation is also entered in the Q parameters Q115 and Q116 2: The measured values ZL and XL as well as DZL and DXL are compared with the values from the tool table, but are not changed. If the values are larger than the permissible wear or breakage tolerance, the control locks the tool (TL = Tool Locked). Input: 0, 1, 2
	Q260 Clearance height? Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from safetyDistStylus). Input: -99999.9999...+99999.9999

Example

11 TOOL CALL 12 Z	
12 TCH PROBE 485 MEASURE LATHE TOOL ~	
Q340=+1	;ENTER DELTA VALUES ~
Q260=+100	;CLEARANCE HEIGHT

10

**Touch-probe cycles
for measuring the
kinematics**

10.1 Overview

Cycle	Call	Further information
450 SAVE KINEMATICS (#17 / #1-05-1) and (#48 / #2-01-1) ■ Storing the active machine kinematic configuration ■ Restoring previously saved kinematic configuration	DEF-active	Page 425
451 MEASURE KINEMATICS (#17 / #1-05-1) and (#48 / #2-01-1) ■ Automatic checking of the machine kinematic configuration ■ Optimizing the machine kinematic configuration	DEF-active	Page 429
452 PRESET COMPENSATION (#17 / #1-05-1) and (#48 / #2-01-1) ■ Automatic checking of the machine kinematic configuration ■ Optimizing the kinematic transformation chain of the machine	DEF-active	Page 445

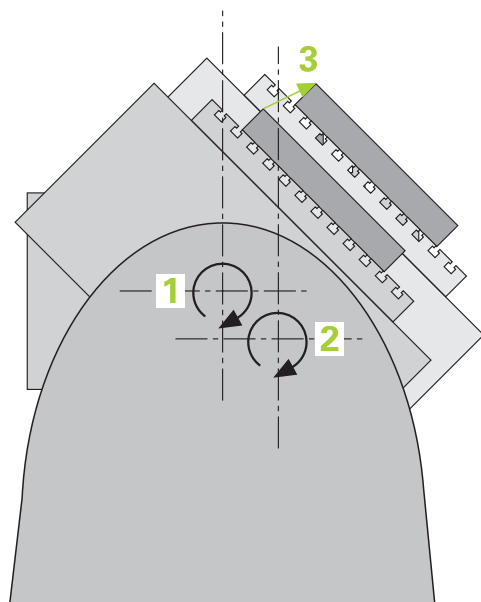
10.2 Conditional stops in touch probe cycles

If your machine has an override controller, you can activate conditional stops during program run. If you activate conditional stops with the **In cycle call** selection, the control does not interrupt program run during any touch probe cycles.

Further information: User's Manual for Setup and Program Run

10.3 Fundamentals of kinematics measurement (#48 / #2-01-1)

10.3.1 Fundamentals



Accuracy requirements are becoming increasingly stringent, particularly in the area of 5-axis machining. Complex parts must be manufactured with both precision and reproducible accuracy, including over extended periods of time.

Some of the reasons for inaccuracy in multi-axis machining are deviations between the kinematic model saved in the control (see **1** in the figure) and the kinematic conditions actually existing on the machine (see **2** in the figure). When the rotary axes are positioned, these deviations cause inaccuracy of the workpiece (see **3** in the figure). It is therefore necessary for the model to approach reality as closely as possible.

The **KinematicsOpt** function of the control is an important component that helps you meet these complex requirements in real life: a 3D touch probe cycle measures the rotary axes on your machine fully automatically, regardless of whether they are realized as tables or spindle heads. For this purpose, a calibration sphere is attached at any position on the machine table, and measured with a resolution that you define. During cycle definition, you simply define for each rotary axis the area that you want to measure.

From the measured values, the control calculates the static tilting accuracy. The software minimizes the positioning error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematics table.

10.3.2 Requirements



Refer to your machine manual.

The Adv. Function Set 1 (#8 / #1-01-1) software option must be enabled.

The KinematicsOpt (#48 / #2-01-1) software option must be enabled.

Machine and control must be specially prepared by the machine manufacturer for use of this cycle.

Requirements for using KinematicsOpt:



The machine manufacturer must have defined the machine parameters for **CfgKinematicsOpt** (no. 204800) in the configuration data.

- **maxModification** (no. 204801) specifies the tolerance limit starting from which the control is to display a message if the changes made to the kinematic data exceed this limit value
- **maxDevCalBall** (no. 204802) defines how much the measured radius of the calibration sphere may deviate from the entered cycle parameter
- **mStrobeRotAxPos** (no. 204803) defines an M function that is specifically configured by the machine manufacturer and is used to position the rotary axes

- The 3D touch probe used for the measurement must be calibrated.
- The cycles can only be carried out with the tool axis Z.
- A calibration sphere with an exactly known radius and sufficient rigidity must be attached to any position on the machine table.
- The kinematics description of the machine must be complete and correct, and the transformation dimensions must have been entered with an accuracy of approx. 1 mm.
- The complete machine geometry must have been measured (by the machine manufacturer during commissioning).



HEIDENHAIN recommends using the calibration spheres **KKH 250 (ordering number: 655475-01)** or **KKH 80 (ordering 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.

10.3.3 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 functions for coordinate transformation must be active. Risk of collision!

- ▶ Do not activate the following cycles before the use of touch probe cycles:
 - **TRANS DATUM**
 - **TRANS MIRROR**
 - **TRANS ROTATION**
 - **TRANS SCALE**
 - Cycle **8 MIRRORING**
 - Cycle **10 ROTATION**
 - Cycle **11 SCALING FACTOR**
 - 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 **mStrokeRotAxPos** (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.

10.4 Storing, measuring and optimizing kinematics (#48 / #2-01-1)

10.4.1 Cycle 450 SAVE KINEMATICS (#48 / #2-01-1)

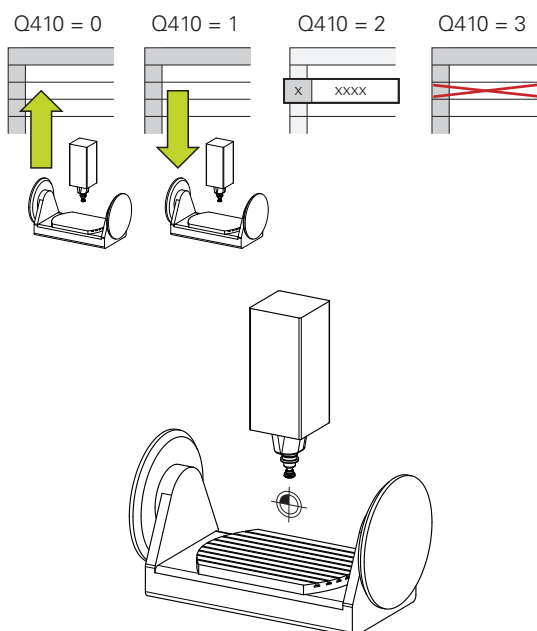
ISO programming
G450

Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.



With touch probe cycle **450** you can save the active machine kinematic configuration or restore a previously saved one. The saved data can be displayed and deleted. 16 memory spaces in total are available.

Notes



Only save and restore data with Cycle **450** while no tool carrier kinematics configuration that includes transformations is active.

- This cycle can be executed only 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 will be 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).

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 does not function if mode 2 has been selected. Wildcards can be used for searches in modes 1 and 3 (Restore and Delete). If the control finds several possible data records because of the wildcards, the control restores the mean values of the data (mode 1) or deletes all selected data records after confirmation (mode 3). You can use the following wildcards in searches: ?: 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 are available.

Saving the current kinematics

11 TCH PROBE 450 SAVE KINEMATICS ~	
Q410=+0	;BACKUP THE KINEMATICS ~
Q409=+947	;MEMORY DESIGNATION

Restoring data records

11 TCH PROBE 450 SAVE KINEMATICS ~	
Q410=+1	;RESTORE THE KINEMATICS ~
Q409=+948	;MEMORY DESIGNATION

Displaying all saved data records

11 TCH PROBE 450 SAVE KINEMATICS ~	
Q410=+2	;DISPLAY MEMORY STATUS ~
Q409=+949	;MEMORY DESIGNATION

Deleting data records

11 TCH PROBE 450 SAVE KINEMATICS ~	
Q410=+3	;DELETE DATA RECORD ~
Q409=+950	;MEMORY DESIGNATION

Logging function

After running Cycle **450**, the control creates a log (**TCHPRAUTO.html**) containing the following information:

- Creation date and time of the log
- Name of the NC program from which the cycle was run
- Designator of the current kinematics
- Active tool

The other data in the log vary depending on the selected mode:

- Mode 0: Logging of all axis entries and transformation entries of the kinematics chain that the control has saved.
- Mode 1: Logging of all transformation entries before and after restoring the kinematics configuration.
- Mode 2: List of the saved data records
- Mode 3: List of the deleted data records

10.4.2 Cycle 451 MEASURE KINEMATICS (#48 / #2-01-1)

ISO programming

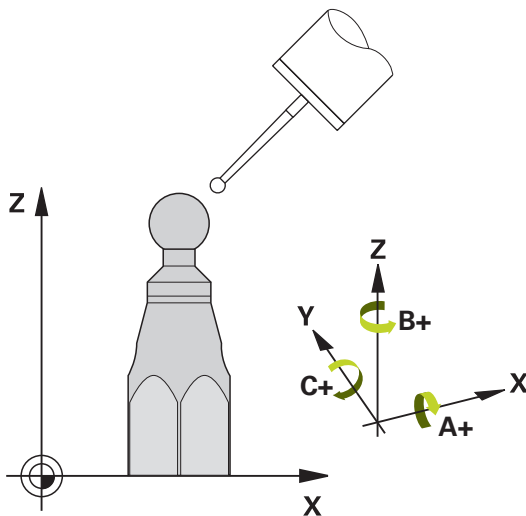
G451

Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.



Touch probe cycle **451** enables you to check and, if required, optimize the kinematics of your machine. Use the 3D TS touch probe to measure a HEIDENHAIN calibration sphere that you have attached to the machine table.

The control will determine the static tilting accuracy. The software minimizes the spatial error arising from the tilting movements and, at the end of the measurement process, automatically saves the machine geometry in the respective machine constants of the kinematics description.

Cycle 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.

Result parameter Q

The control saves the results of the touch probe cycle in the following Q parameters:

Q parameter number	Meaning
Q141	Standard deviation measured in the A axis (–1 if axis was not measured)
Q142	Standard deviation measured in the B axis (–1 if axis was not measured)
Q143	Standard deviation measured in the C axis (–1 if axis was not measured)
Q144	Optimized standard deviation in the A axis (–1 if axis was not optimized)
Q145	Optimized standard deviation in the B axis (–1 if axis was not optimized)
Q146	Optimized standard deviation in the C axis (–1 if axis was not optimized)
Q147	Offset error in X direction, for manual transfer to the corresponding machine parameter
Q148	Offset error in Y direction, for manual transfer to the corresponding machine parameter
Q149	Offset error in Z direction, for manual transfer to the corresponding machine parameter

Result parameter QS

The control saves the measured position faults of rotary axes in the QS parameters **QS144 to QS146**. Each result is ten characters long. The results are separated from each other by a space.

Example: **QS146** = "0.01234567 -0.0123456 0.00123456 -0.0012345"

Q parameter number	Meaning
QS144	Position error of A axis $E_{Y0A} E_{Z0A} E_{B0A} E_{C0A}$
QS145	Position error of B axis $E_{Z0B} E_{X0B} E_{C0B} E_{A0B}$
QS146	Position error of C axis $E_{X0C} E_{Y0C} E_{A0C} E_{B0C}$



Position faults are deviations from the ideal axis position and are marked by four characters.

Example: E_{X0C} = Position error of the C axis in X direction.

You can convert the individual results in the NC program, using string processing into numerical values and use them in evaluations, for example.

Example:

The cycle produces the following results within the QS parameter **QS146**:

QS146 = "0.01234567 -0.0123456 0.00123456 -0.0012345"

The example below shows how to convert the results produced into numerical values.

11 QS0 = SUBSTR (SRC_QS146 BEG0 LEN10)	; Read out the first result E_{X0C} from QS146
12 QL0 = TONUMB (SRC_QS0)	; Convert alphanumeric value from QS0 to a numerical value and assign it to QL0
13 QS0 = SUBSTR (SRC_QS146 BEG11 LEN10)	; Read out the second result E_{Y0C} from QS146
14 QL1 = TONUMB (SRC_QS0)	; Convert alphanumeric value from QS0 to a numerical value and assign it to QL1
15 QS0 = SUBSTR (SRC_QS146 BEG22 LEN10)	; Read out the third result E_{A0C} from QS146
16 QL2 = TONUMB (SRC_QS0)	; Convert alphanumeric value from QS0 to a numerical value and assign it to QL2
17 QS0 = SUBSTR (SRC_QS146 BEG33 LEN10)	; Read out the forth result E_{B0C} from QS146
18 QL3 = TONUMB (SRC_QS0)	; Convert alphanumeric value from QS0 to a numerical value and assign it to QL3

Further information: Programming and Testing User's Manual

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° minus +90°) / (4 minus 1) = -60°
 - Measuring point 1 = +90°
 - Measuring point 2 = +30°
 - Measuring point 3 = -30°
 - Measuring point 4 = -90°
- Example: start angle = +90°, end angle = +270°
 - Start angle = +90°
 - End angle = +270°
 - No. of measuring points = 4
 - Stepping angle resulting from the calculation
= (270° minus 90°) / (4 minus 1) = +60°
 - Measuring point 1 = +90°
 - Measuring point 2 = +150°
 - Measuring point 3 = +210°
 - Measuring point 4 = +270°

Machines with Hirth-coupled axes

NOTICE

Danger of collision!

In order to be positioned, the axis must move out of the Hirth grid. If necessary, the control rounds the calculated measuring positions so that they fit into the Hirth grid (depending on the start angle, end angle and number of measuring points). There is a danger of collision!

- ▶ So remember to leave a large enough set-up clearance to prevent any risk of collision between the touch probe and calibration sphere
- ▶ Also ensure that there is enough space to reach the set-up clearance (software limit switch)

NOTICE

Danger of collision!

Depending on the machine configuration, the control cannot position the rotary axes automatically. If this is the case, you need a special M function from the machine manufacturer, enabling the control to move the rotary axes. The machine manufacturer must have entered the number of the M function in the machine parameter **mStrobeRotAxPos** (no. 204803) for this purpose. There is a danger of collision!

- ▶ Note the documentation of the machine manufacturer



- Define a retraction height greater than 0 if the Adv. Function Set 2 (#9 / #4-01-1) software option is not available.
- The measured positions are calculated from the start angle, end angle, and number of measurements for the respective axis and from the Hirth grid.

Example calculation of measuring positions for an A axis:

Start angle **Q411** = -30

End angle **Q412** = +90

Number of measuring points **Q414** = 4

Hirth grid = 3°

Calculated stepping angle = (**Q412** minus **Q411**) / (**Q414** minus 1)

Calculated stepping angle = (90° minus -30°) / (4 minus 1) = 120° / 3 = 40°

Measuring position 1 = **Q411** + 0 * stepping angle = -30° → -30°

Measuring position 2 = **Q411** + 1 * stepping angle = +10° → 9°

Measuring position 3 = **Q411** + 2 * stepping angle = +50° → 51°

Measuring position 4 = **Q411** + 3 * stepping angle = +90° → 90°

Choice of number of measuring points

To save time, you can make a rough optimization with a small number of measuring points (1 or 2), for example when commissioning the machine.

You then make a fine optimization with a medium number of measuring points (recommended value = approx. 4). Higher numbers of measuring points do not usually improve the results. Ideally, you should distribute the measuring points evenly over the tilting range of the axis.

This is why you should measure an axis with a tilting range of 0° to 360° at three measuring points, namely at 90°, 180° and 270°. Thus, define a starting angle of 90° and an end angle of 270°.

If you want to check the accuracy accordingly, you can also enter a higher number of measuring points in the **Check** mode.



If a measuring point has been defined at 0°, it will be ignored because the reference measurement is always done at 0°.

Choice of the calibration sphere position on the machine table

In principle, you can fix the calibration sphere to any accessible position on the machine table and also on fixtures or workpieces. The following factors should positively influence the result of measurement:

- On machines with rotary tables/tilting tables: Clamp the calibration sphere as far as possible away from the center of rotation.
- On machines with very large traverse paths: Clamp the calibration sphere as closely as possible to the position intended for subsequent machining.



Position the calibration sphere on the machine table so that there can be no collisions during the measuring process.

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.

Notes on the accuracy



If required, deactivate the lock on the rotary axes for the duration of the calibration. Otherwise it may falsify the results of measurement. The machine manual provides further information.

The geometrical and positioning errors of the machine influence the measured values and therefore also the optimization of a rotary axis. For this reason there will always be a certain amount of error.

If there were no geometrical and positioning errors, any values measured by the cycle at any point on the machine at a certain time would be exactly reproducible. The greater the geometrical and positioning errors are, the greater is the dispersion of measured results when you perform measurements at different positions.

The dispersion output by the control in the measurement log is a measure of the machine's static tilting accuracy. However, the measuring circle radius and the number and position of measuring points have to be included in the evaluation of accuracy. One measuring point alone is not enough to calculate dispersion. For only one point, the result of the calculation is the spatial error of that measuring point.

If several rotary axes are moved simultaneously, their error values are combined. In the worst case they are added together.



If your machine is equipped with a feedback-controlled spindle, you should activate angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3D touch probe.

Backlash

Backlash is a small amount of play between the rotary or angle encoder and the table that occurs when the traverse direction is reversed. If the rotary axes have backlash outside of the control loop, for example because the angle measurement is performed with the motor encoder, this can result in significant error during tilting.

With input parameter **Q432**, you can activate backlash measurement. Enter an angle that the control uses as the traversing angle. The cycle will then carry out two measurements per rotary axis. If you take over the angle value 0, the control will not measure any backlash.



Backlash measurement is not possible if an M function for positioning the rotary axes is set in the optional **mStrobeRotAxPos** machine parameter (no. 204803) or if the axis is a Hirth axis.



Programming and operating notes:

- The control does not perform an automatic backlash compensation.
- If the measuring circle radius is < 1 mm, the control does not calculate the backlash. The larger the measuring circle radius, the more accurately the control can ascertain the rotary axis backlash.

Further information: "Log function", Page 444

Notes

NOTICE

Danger of collision!

If you run this cycle, a basic rotation or 3D basic rotation must not be active. The control will delete the values from the columns **SPA**, **SPB** and **SPC** of the preset table as needed. After the cycle, you need to set a basic rotation or 3D basic rotation again; otherwise, there is a danger of collision.

- ▶ Deactivate the basic rotation before running the cycle.
- ▶ Set the preset and the basic rotation again after optimization.

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.
- As with Cycles **451** and **452**, Cycle **453** ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.
- Before defining the cycle, you must set the preset to the center of the calibration sphere and activate it, or set input parameter **Q431** to 1 or 3, respectively.
- For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while probe monitoring is inactive.
- The control ignores cycle definition data that applies to inactive axes.
- A correction in the machine datum (**Q406=3**) is only possible if superimposed rotary axes on the spindle head side or table side are measured.
- If you have activated presetting before the calibration (**Q431** = 1/3), then move the touch probe to the set-up clearance (**Q320** + SET_UP) to a position approximately above the center of the calibration sphere before the start of the cycle.
- Programming in inches: The control always records the log data and results of measurement in millimeters.
- After measuring the kinematics, you must re-determine the preset.

Notes about machine parameters

- If the optional machine parameter **mStrobeRotAxPos** (no. 204803) is not equal to -1 (M function positions the rotary axis), then start a measurement only if all rotary axes are at 0°.
- In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the optional machine parameter **maxDevCalBall** (no. 204802), the control displays an error message and ends the measurement.
- For angle optimization, the machine manufacturer must adapt the configuration correspondingly.

Cycle parameters

Help graphic	Parameter
	Q406 Mode (0/1/2/3)? Define whether the control will check or optimize the active kinematics: 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. 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. 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. The KinematicsComp software option (#52 / #2-04-1) is required for compensation of angle errors. 3: Optimize the active machine kinematics: The control measures the kinematics in the rotary axes you have defined. It then automatically corrects the machine datum. It then optimizes angle and position errors. The KinematicsComp software option (#52 / #2-04-1) is required for this. Input: 0, 1, 2, 3
	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 active in addition to the SET_UP column in the touch probe table. This value has an incremental effect. Input: 0...99999.9999 (or PREDEF)
	Q408 Retraction height? 0: Do not move to any retraction height; the control moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C. > 0: Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter Q253. This value has an absolute effect. Input: 0...99999.9999

Help graphic	Parameter
	Q253 Feed rate for pre-positioning? Define the traversing speed of the tool during pre-positioning in mm/min. Input: 0...99999.9999 or FMAX, FAUTO, PREDEF
	Q380 Ref. angle in ref. axis? Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. This value has an absolute effect. Input: 0...360
	Q411 Starting angle in A axis? Starting angle in the A axis at which the first measurement will be made. This value has an absolute effect. Input: -359.9999...+359.9999
	Q412 End angle in A axis? End angle in the A axis at which the last measurement will be made. This value has an absolute effect. Input: -359.9999...+359.9999
	Q413 Angle of incidence in A axis? Angle of incidence in the A axis at which the other rotary axes will be measured. Input: -359.9999...+359.9999
	Q414 No. of meas. points in A (0...12)? Number of measuring points the control will use to measure the A axis. If the input value = 0, the control does not measure the respective axis. Input: 0...12
	Q415 Starting angle in B axis? Starting angle in the B axis at which the first measurement will be made. This value has an absolute effect. Input: -359.9999...+359.9999
	Q416 End angle in B axis? End angle in the B axis at which the last measurement will be made. This value has an absolute effect. Input: -359.9999...+359.9999
	Q417 Angle of incidence in B axis? Angle of incidence in the B axis at which the other rotary axes will be measured. Input: -359.999...+360.000

Help graphic	Parameter
	Q418 No. of meas. points in B (0...12)? Number of measuring points the control will use to measure the B axis. If the input value = 0, the control does not measure the respective axis. Input: 0...12
	Q419 Starting angle in C axis? Starting angle in the C axis at which the first measurement will be made. This value has an absolute effect. Input: -359.9999...+359.9999
	Q420 End angle in C axis? End angle in the C axis at which the last measurement will be made. This value has an absolute effect. Input: -359.9999...+359.9999
	Q421 Angle of incidence in C axis? Angle of incidence in the C axis at which the other rotary axes will be measured. Input: -359.9999...+359.9999
	Q422 No. of meas. points in C (0...12)? Number of measuring points the control will use to measure the C axis. If the input value = 0, the control does not measure the respective axis. Input: 0...12
	Q423 Number of probes? (optional) 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)? (optional) 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.?** (optional)

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	;BACKUP THE KINEMATICS ~
Q409=+5	;MEMORY DESIGNATION
13 TCH PROBE 451 MEASURE KINEMATICS ~	
Q406=+0	;CHECK ~
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	;NEVER SET A PRESET ~
Q432=+0	;BACKLASH, ANG. RANGE

Various modes (Q406)**Test mode Q406 = 0**

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- The control records the results of a possible position optimization but does not make any adjustments.

"Optimize position of rotary axes" mode Q406 = 1

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- During this, the control tries to change the position of the rotary axis in the kinematics model in order to achieve higher accuracy.
- The machine data are adjusted automatically.

"Optimize position and angle" mode Q406 = 2

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- After that, the position is optimized. No additional measurements are necessary for this; the control calculates the optimization of the position automatically.



Depending on the machine kinematics for correctly determining the angles, HEIDENHAIN recommends performing the measurement once with an inclination angle of 0°.

"Optimize machine datum, position, and angle" mode (Q406 = 3)

- The control measures the rotary axes in the positions defined and calculates the static accuracy of the tilting transformation.
- After that, the position is optimized. No additional measurements are necessary for this; the control calculates the optimization of the position automatically.



- For correct determination of the angular position errors, HEIDENHAIN recommends setting the affected rotary axis to an inclination angle of 0° for this measurement.
- After correcting a machine datum, the control tries to reduce the compensation of the associated angular position error (**locErrA/locErrB/locErrC**) of the measured rotary axis.

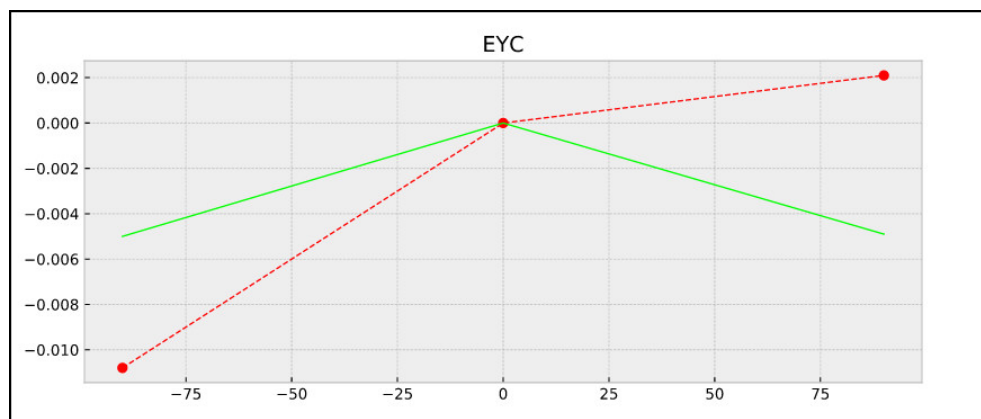
Position optimization of the rotary axes with preceding, automatic presetting and measurement of the rotary axis backlash

11 TOOL CALL "TOUCH_PROBE" Z	
12 TCH PROBE 451 MEASURE KINEMATICS ~	
Q406=+1	;OPTIMIZE POSITION ~
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	;AT BEGINNING ~
Q432=+0.5	;BACKLASH, ANG. RANGE

Log function

After running Cycle 451, the control creates a log (**TCHPRAUTO.html**) and saves it in the folder that also contains the associated NC program. This log contains the following data:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Tool name
- Active kinematics
- Mode used (0=Check/1=Optimize position/2=Optimize pose/3=Optimize machine datum and pose)
- Inclination angles
- For each measured rotary axis:
 - Starting angle
 - End angle
 - Number of measuring points
 - Measuring circle radius
 - Averaged backlash, if **Q423>0**
 - Positions of the axes
 - Standard deviation (scatter)
 - Maximum deviation
 - Angular error
 - Compensation values in all axes (preset shift)
 - Position before optimization of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
 - Position after optimization of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
 - Averaged positioning error and standard deviation of the positioning errors to 0
 - SVG files with graphs: measured and optimized errors of individual measurement positions.
 - Red curve: measured positions
 - Green curve: optimized values after cycle has run
 - Designation of the graph: axis designation depends on the rotary axis (e.g., EYC = component error in Y of axis C)
 - X axis of the graph: rotary axis position in degrees
 - Y axis of the graph: position deviations in mm



Sample measurement: EYC component error in Y of axis C

10.4.3 Cycle 452 PRESET COMPENSATION (#48 / #2-01-1)

ISO programming

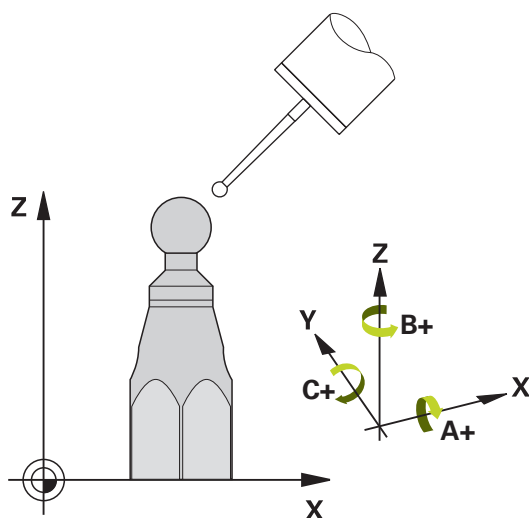
G452

Application



Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.



Touch probe cycle **452** optimizes the kinematic transformation chain of your machine (see "Cycle 451 MEASURE KINEMATICS (#48 / #2-01-1)", Page 429). 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 for it in the kinematics description.

Result parameter Q

Q parameter number	Meaning
Q141	Standard deviation measured in the A axis (-1 if axis was not measured)
Q142	Standard deviation measured in the B axis (-1 if axis was not measured)
Q143	Standard deviation measured in the C axis (-1 if axis was not measured)
Q144	Optimized standard deviation in the A axis (-1 if axis was not measured)
Q145	Optimized standard deviation in the B axis (-1 if axis was not measured)
Q146	Optimized standard deviation in the C axis (-1 if axis was not measured)
Q147	Offset error in X direction, for manual transfer to the corresponding machine parameter
Q148	Offset error in Y direction, for manual transfer to the corresponding machine parameter
Q149	Offset error in Z direction, for manual transfer to the corresponding machine parameter

Result parameter QS

The control saves the measured position faults of rotary axes in the QS parameters **QS144 to QS146**. Each result is ten characters long. The results are separated from each other by a space.

Example: **QS146** = "0.01234567 -0.0123456 0.00123456 -0.0012345"

Q parameter number	Meaning
QS144	Position error of A axis $E_{Y0A} E_{Z0A} E_{B0A} E_{C0A}$
QS145	Position error of B axis $E_{Z0B} E_{X0B} E_{C0B} E_{A0B}$
QS146	Position error of C axis $E_{X0C} E_{Y0C} E_{A0C} E_{B0C}$



Position faults are deviations from the ideal axis position and are marked by four characters.

Example: E_{X0C} = Position error of the C axis in X direction.

You can convert the individual results in the NC program, using string processing into numerical values and use them in evaluations, for example.

Example:

The cycle produces the following results within the QS parameter **QS146**:

QS146 = "0.01234567 -0.0123456 0.00123456 -0.0012345"

The example below shows how to convert the results produced into numerical values.

11 QS0 = SUBSTR (SRC_QS146 BEG0 LEN10)	; Read out the first result E_{X0C} from QS146
12 QL0 = TONUMB (SRC_QS0)	; Convert alphanumeric value from QS0 to a numerical value and assign it to QL0
13 QS0 = SUBSTR (SRC_QS146 BEG11 LEN10)	; Read out the second result E_{Y0C} from QS146
14 QL1 = TONUMB (SRC_QS0)	; Convert alphanumeric value from QS0 to a numerical value and assign it to QL1
15 QS0 = SUBSTR (SRC_QS146 BEG22 LEN10)	; Read out the third result E_{A0C} from QS146
16 QL2 = TONUMB (SRC_QS0)	; Convert alphanumeric value from QS0 to a numerical value and assign it to QL2
17 QS0 = SUBSTR (SRC_QS146 BEG33 LEN10)	; Read out the forth result E_{B0C} from QS146
18 QL3 = TONUMB (SRC_QS0)	; Convert alphanumeric value from QS0 to a numerical value and assign it to QL3

Further information: Programming and Testing User's Manual

Notes



In order to be able to perform a preset compensation, the kinematics must be specially prepared. The machine manual provides further information.

NOTICE

Danger of collision!

If you run this cycle, a basic rotation or 3D basic rotation must not be active. The control will delete the values from the columns **SPA**, **SPB** and **SPC** of the preset table as needed. After the cycle, you need to set a basic rotation or 3D basic rotation again; otherwise, there is a danger of collision.

- ▶ Deactivate the basic rotation before running the cycle.
- ▶ Set the preset and the basic rotation again after optimization.

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.
- As with Cycles **451** and **452**, Cycle **453** ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.
- Ensure that all functions for tilting the working plane are reset.
- Before defining the cycle, you must set the preset at the center of the calibration sphere and activate it.
- For rotary axes without separate position encoders, select the measuring points in such a way that you have to traverse an angle of 1° to the limit switch. The control needs this traverse for internal backlash compensation.
- For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while touch probe monitoring is inactive.
- Programming in inches: The control always records the log data and results of measurement in millimeters.



- If you interrupt the cycle during the measurement, the kinematic data might no longer be in the original condition. Save the active kinematic configuration before an optimization with Cycle **450**, so that in case of a failure the most recently active kinematic configuration can be restored.

Notes about machine parameters

- In the machine parameter **maxModification** (no. 204801), the machine manufacturer defines the permissible limit value for modifications of a transformation. If the kinematics data determined exceed the permissible limit value, the control displays a warning. Then you have to confirm acceptance of the determined values by pressing **NC Start**.
- In the machine parameter **maxDevCalBall** (no. 204802), the machine manufacturer defines the maximum deviation of the calibration sphere radius. In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the machine parameter **maxDevCalBall** (no. 204802), the control displays an error message and ends the measurement.

Cycle parameters

Help graphic	Parameter
	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 active in addition to the SET_UP column in the touch probe table. This value has an incremental effect. Input: 0...99999.9999 (or PREDEF)
	Q408 Retraction height? 0: Do not move to any retraction height; the control moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C. > 0: Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter Q253 . This value has an absolute effect. Input: 0...99999.9999
	Q253 Feed rate for pre-positioning? Define the traversing speed of the tool during pre-positioning in mm/min. Input: 0...99999.9999 or FMAX, FAUTO, PREDEF
	Q380 Ref. angle in ref. axis? Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. This value has an absolute effect. Input: 0...360
	Q411 Starting angle in A axis? Starting angle in the A axis at which the first measurement will be made. This value has an absolute effect. Input: -359.9999...+359.9999
	Q412 End angle in A axis? End angle in the A axis at which the last measurement will be made. This value has an absolute effect. Input: -359.9999...+359.9999
	Q413 Angle of incidence in A axis? Angle of incidence in the A axis at which the other rotary axes will be measured. Input: -359.9999...+359.9999

Help graphic	Parameter
	Q414 No. of meas. points in A (0...12)? Number of measuring points the control will use to measure the A axis. If the input value = 0, the control does not measure the respective axis. Input: 0...12
	Q415 Starting angle in B axis? Starting angle in the B axis at which the first measurement will be made. This value has an absolute effect. Input: -359.9999...+359.9999
	Q416 End angle in B axis? End angle in the B axis at which the last measurement will be made. This value has an absolute effect. Input: -359.9999...+359.9999
	Q417 Angle of incidence in B axis? Angle of incidence in the B axis at which the other rotary axes will be measured. Input: -359.999...+360.000
	Q418 No. of meas. points in B (0...12)? Number of measuring points the control will use to measure the B axis. If the input value = 0, the control does not measure the respective axis. Input: 0...12
	Q419 Starting angle in C axis? Starting angle in the C axis at which the first measurement will be made. This value has an absolute effect. Input: -359.9999...+359.9999
	Q420 End angle in C axis? End angle in the C axis at which the last measurement will be made. This value has an absolute effect. Input: -359.9999...+359.9999
	Q421 Angle of incidence in C axis? Angle of incidence in the C axis at which the other rotary axes will be measured. Input: -359.9999...+359.9999
	Q422 No. of meas. points in C (0...12)? Number of measuring points the control will use to measure the C axis. If the input value = 0, the control does not measure the respective axis. Input: 0...12
	Q423 Number of probes? Define the number of measuring points the control will use to measure the calibration sphere in the plane. Fewer measuring points increase speed, and more measuring points increase measurement precision. Input: 3...8

Help graphic**Parameter****Q432 Angular range of backlash comp.?** (optional)

Define the traversing angle the control will use to measure the rotary axis backlash. The traversing angle must be significantly larger than the actual backlash of the rotary axes. If input value = 0, the control does not measure the backlash.

Input: **-3...+3**

Calibration program

11	TOOL CALL "TOUCH_PROBE" Z
12	TCH PROBE 450 SAVE KINEMATICS ~
Q410	=+0 ;MODE ~
Q409	=+5 ;MEMORY DESIGNATION
13	TCH PROBE 452 PRESET COMPENSATION ~
Q407	=+12.5 ;SPHERE RADIUS ~
Q320	=+0 ;SET-UP CLEARANCE ~
Q408	=+0 ;RETR. HEIGHT ~
Q253	=+750 ;F PRE-POSITIONING ~
Q380	=+0 ;REFERENCE ANGLE ~
Q411	=-90 ;START ANGLE A AXIS ~
Q412	=+90 ;END ANGLE A AXIS ~
Q413	=+0 ;INCID. ANGLE A AXIS ~
Q414	=+0 ;MEAS. POINTS A AXIS ~
Q415	=-90 ;START ANGLE B AXIS ~
Q416	=+90 ;END ANGLE B AXIS ~
Q417	=+0 ;INCID. ANGLE B AXIS ~
Q418	=+2 ;MEAS. POINTS B AXIS ~
Q419	=-90 ;START ANGLE C AXIS ~
Q420	=+90 ;END ANGLE C AXIS ~
Q421	=+0 ;INCID. ANGLE C AXIS ~
Q422	=+2 ;MEAS. POINTS C AXIS ~
Q423	=+4 ;NO. OF PROBE POINTS ~
Q432	=+0 ;BACKLASH, ANG. RANGE

Adjustment of interchangeable heads



The head change function can vary depending on the individual machine tool. Refer to your machine manual.

- ▶ Load the second interchangeable head.
- ▶ Insert the touch probe
- ▶ Measure the interchangeable head with Cycle **452**
- ▶ Measure only the axes that have actually been changed (in this example: only the A axis; the C axis is hidden with **Q422**)
- ▶ The preset and the position of the calibration sphere must not be changed during the entire process.
- ▶ All other interchangeable heads can be adjusted in the same way

Adjusting an interchangeable head

11 TOOL CALL "TOUCH_PROBE" Z	
12 TCH PROBE 452 PRESET COMPENSATION ~	
Q407=+12.5	;SPHERE RADIUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q408=+0	;RETR. HEIGHT ~
Q253=+2000	;F PRE-POSITIONING ~
Q380=+45	;REFERENCE ANGLE ~
Q411=-90	;START ANGLE A AXIS ~
Q412=+90	;END ANGLE A AXIS ~
Q413=+45	;INCID. ANGLE A AXIS ~
Q414=+4	;MEAS. POINTS A AXIS ~
Q415=-90	;START ANGLE B AXIS ~
Q416=+90	;END ANGLE B AXIS ~
Q417=+0	;INCID. ANGLE B AXIS ~
Q418=+2	;MEAS. POINTS B AXIS ~
Q419=+90	;START ANGLE C AXIS ~
Q420=+270	;END ANGLE C AXIS ~
Q421=+0	;INCID. ANGLE C AXIS ~
Q422=+0	;MEAS. POINTS C AXIS ~
Q423=+4	;NO. OF PROBE POINTS ~
Q432=+0	;BACKLASH, ANG. RANGE

The goal of this procedure is that the workpiece preset remains unchanged after changing rotary axes (head change).

In the following example, the adjustment of a fork head with A and C axes is described. The A axis is changed, whereas the C axis continues to be a part of the basic configuration.

- ▶ Insert the interchangeable head that will be used as a reference head.
- ▶ Clamp the calibration sphere
- ▶ Insert the touch probe
- ▶ Use Cycle **451** to measure the complete kinematics, including the reference head
- ▶ Define the preset (using **Q431** = 2 or 3 in Cycle **451**) after measuring the reference head

Measuring a reference head

11 TOOL CALL "TOUCH_PROBE" Z	
12 TCH PROBE 451 MEASURE KINEMATICS ~	
Q406=+1	;MODE ~
Q407=+12.5	;SPHERE RADIUS ~
Q320=+0	;SET-UP CLEARANCE ~
Q408=+0	;RETR. HEIGHT ~
Q253=+2000	;F PRE-POSITIONING ~
Q380=+45	;REFERENCE ANGLE ~
Q411=-90	;START ANGLE A AXIS ~
Q412=+90	;END ANGLE A AXIS ~
Q413=+45	;INCID. ANGLE A AXIS ~
Q414=+4	;MEAS. POINTS A AXIS ~
Q415=-90	;START ANGLE B AXIS ~
Q416=+90	;END ANGLE B AXIS ~
Q417=+0	;INCID. ANGLE B AXIS ~
Q418=+2	;MEAS. POINTS B AXIS ~
Q419=+90	;START ANGLE C AXIS ~
Q420=+270	;END ANGLE C AXIS ~
Q421=+0	;INCID. ANGLE C AXIS ~
Q422=+3	;MEAS. POINTS C AXIS ~
Q423=+4	;NO. OF PROBE POINTS ~
Q431=+3	;PRESET ~
Q432=+0	;BACKLASH, ANG. RANGE

Drift compensation



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 for 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.

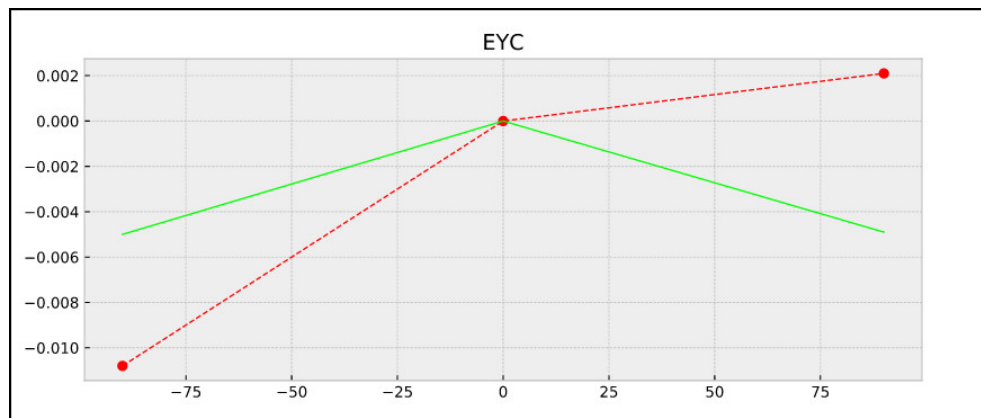
Drift compensation

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

Logging function

After running Cycle **452**, the control creates a log (**TCHPRAUTO.html**) and saves it in the folder that also contains the associated NC program. This log contains the following data:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Tool name
- Active kinematics
- Mode used
- Inclination angles
- For each measured rotary axis:
 - Starting angle
 - End angle
 - Number of measuring points
 - Measuring circle radius
 - Averaged backlash, if **Q423>0**
 - Positions of the axes
 - Standard deviation (scatter)
 - Maximum deviation
 - Angular error
 - Compensation values in all axes (preset shift)
 - Position before preset compensation of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
 - Position after preset compensation of the rotary axes checked (relative to the beginning of the kinematic transformation chain, usually the spindle nose)
 - Averaged positioning error
 - SVG files with graphs: measured and optimized errors of individual measurement positions.
 - Red curve: measured positions
 - Green curve: optimized values
 - Designation of the graph: axis designation depends on the rotary axis (e.g., EYC = deviations) of the Y axis in dependency of the C axis.
 - X axis of the graph: rotary axis position in degrees
 - Y axis of the graph: position deviations in mm



Sample measurement: EYC deviations of the Y axis in dependency of the C axis

10.4.4 Cycle 453 KINEMATICS GRID (#48 / #2-01-1)

ISO programming

G453

Application

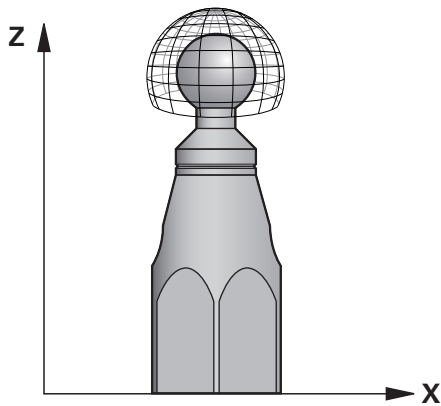


Refer to your machine manual.

The KinematicsOpt (#48 / #2-01-1) software option is required.

This function must be enabled and adapted by the machine manufacturer.

To use this cycle, your machine manufacturer needs to create and configure a compensation table (*.kco) first and enter some more settings.



Even if your machine was already optimized regarding positioning errors (e.g., via Cycle **451**), residual errors at the Tool Center Point (**TCP**) during tilting of the rotary axes may remain. These can result, for example, from component errors (e.g., a bearing error) with head rotation axes.

Cycle **453 KINEMATICS GRID** enables errors in swivel heads to be detected and compensated for in accordance with the rotary axis positions. With this cycle and using a 3D TS touch probe, you measure a HEIDENHAIN calibration sphere that you have attached to the machine table. The cycle then moves the touch probe automatically to positions in a grid-line arrangement around the calibration sphere. The machine manufacturer defines these swivel axis positions. You can arrange the positions in up to three dimensions. (Each dimension is a rotary axis.) After the probing operation on the sphere, compensation of the errors can be performed using a multi-dimensional table. The machine manufacturer defines this compensation table (*.kco) and specifies its storage location.

When using Cycle **453**, run it at different positions in the workspace. This allows you to check immediately if compensation with Cycle **453** has the desired positive effect on the machine's accuracy. Only when the desired improvements are achieved with the same compensation values at several positions is such a type of compensation suitable for the respective machine. If this is not the case, then the errors are to be sought outside the rotary axes.

Perform the measurement with Cycle **453** in an optimized condition regarding the rotary axis positioning errors. Use, for example, Cycle **451** before doing so.



HEIDENHAIN recommends using the calibration spheres **KKH 250 (order number 655475-01)** or **KKH 100 (order number 655475-02)**, which are particularly rigid and are designed especially for machine calibration. Please contact HEIDENHAIN if you have any questions in this regard.

The control then optimizes the accuracy of your machine. For this purpose, it automatically saves the compensation values resulting from a measurement in a compensation table (*.kco). (This applies to mode **Q406=1**.)

Cycle 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.

Various modes (Q406)

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

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.

Notes



The KinematicsOpt (#48 / #2-01-1) software option is required. This function must be enabled and adapted by the machine manufacturer. Your machine manufacturer defines the storage location of the compensation table (*.kco).

NOTICE

Danger of collision!

If you run this cycle, a basic rotation or 3D basic rotation must not be active. The control will delete the values from the columns **SPA**, **SPB** and **SPC** of the preset table as needed. After the cycle, you need to set a basic rotation or 3D basic rotation again; otherwise, there is a danger of collision.

- ▶ Deactivate the basic rotation before running the cycle.
- ▶ Set the preset and the basic rotation again after optimization.

- This cycle can be executed only in the **FUNCTION MODE MILL** machining mode.
- Before the beginning of the cycle, **M128** or **FUNCTION TCPM** must be switched off.
- As with Cycles **451** and **452**, Cycle **453** ends with active 3D-ROT in automatic mode, matching the position of the rotary axes.
- Before defining the cycle, you must set the preset to the center of the calibration sphere and activate it, or you set input parameter **Q431** to 1 or 3, respectively.
- For the positioning feed rate when moving to the probing height in the touch probe axis, the control uses the value from cycle parameter **Q253** or the **FMAX** value from the touch probe table, whichever is smaller. The control always moves the rotary axes at positioning feed rate **Q253**, while probe monitoring is inactive.
- Programming in inches: The control always records the log data and results of measurement in millimeters.
- If you have activated preset setting before the calibration (**Q431** = 1/3), then move the touch probe by the amount of the set-up clearance (**Q320** + **SET_UP**) to a position approximately above the center of the calibration sphere before the start of the cycle.



- If your machine is equipped with a feedback-controlled spindle, you should activate angle tracking in the touch probe table (**TRACK column**). This generally increases the accuracy of measurements with a 3D touch probe.

Notes about machine parameters

- In the machine parameter **mStrobeRotAxPos** (no. 204803), the machine manufacturer defines the maximum permissible modification of a transformation. If the value is not equal to -1 (M function positions the rotary axis), then start a measurement only if all rotary axes are at 0°.
- In the machine parameter **maxDevCalBall** (no. 204802), the machine manufacturer defines the maximum deviation of the calibration sphere radius. In every probing process the control first measures the radius of the calibration sphere. If the measured sphere radius differs from the entered sphere radius by more than the value you have defined in the machine parameter **maxDevCalBall** (no. 204802), the control displays an error message and ends the measurement.

Cycle parameters

Help graphic	Parameter
	Q406 Mode (-1/0/+1) 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. -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). (#52 / #2-04-1) 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). 1: Compensate for 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). (#52 / #2-04-1) Input: -1, 0, +1
	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 active in addition to the SET_UP column in the touch probe table. This value has an incremental effect. Input: 0...99999.9999 (or PREDEF)
	Q408 Retraction height? 0: Do not move to any retraction height; the control moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The control moves to the first measuring position in the sequence A, then B, then C. > 0: Retraction height in the untilted workpiece coordinate system to which the control positions the spindle axis before positioning a rotary axis. In addition, the control moves the touch probe in the working plane to the datum. Touch probe monitoring is not active in this mode. Define the positioning feed rate in parameter Q253. This value has an absolute effect. Input: 0...99999.9999
	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

Help graphic**Parameter****Q380 Ref. angle in ref. axis?**

Enter the reference angle (basic rotation) for acquiring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis. This value has an absolute effect.

Input: **0...360**

Q423 Number of probes?

Define the number of measuring points the control will use to measure the calibration sphere in the plane. Fewer measuring points increase speed, and more measuring points increase measurement precision.

Input: **3...8**

Q431 Preset (0/1/2/3)?

Define whether the control will automatically set the active preset at the center of the sphere:

0: Do not set the preset automatically at the center of the sphere: Set the preset manually before the start of the cycle

1: Set the preset automatically at the center of the sphere before measurement (the active preset will be overwritten): Pre-position the touch probe manually above the calibration sphere before the start of the cycle

2: Set the preset automatically at the center of the sphere after measurement (the active preset will be overwritten): Set the preset manually before the start of the cycle

3: Set the preset at the center of the sphere before and after measurement (the active preset will be overwritten): Pre-position the touch probe manually above the calibration sphere before the start of the cycle

Input: **0, 1, 2, 3**

Probing with Cycle 453

11 TCH PROBE 453 KINEMATICS GRID ~	
Q406=+0	;CHECK ~
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	;NEVER SET A PRESET

Logging function

After running Cycle **453**, the control creates a log (**TCHPRAUTO.html**) and saves it in the folder where the current NC program resides. It contains the following data:

- Date and time of log creation
- Path of the NC program from which the cycle was run
- Number and name of the currently active tool
- Mode
- Measured data: Standard deviation and maximum deviation
- Information at which position in degrees (°) the maximum deviation occurred
- Number of measuring positions

Index

A

About the product.....	35
About the User's Manual.....	21
Additional documentation.....	24
Application	
Help.....	29
Start/Login.....	51
Automatic preset setting	
Bolt hole circle.....	259
Center of 4 holes.....	269
Circle.....	282
Circular pocket (hole).....	234
Circular stud.....	240
Fundamentals of 4xx.....	212
Inside corner.....	253
Island.....	292
Outside corner.....	246
Pocket.....	292
Rectangular pocket.....	224
Rectangular stud.....	229
Reference plane.....	323
Ridge.....	298
Ridge center.....	219
Ridge undercut.....	310
Single axis.....	274
Single position.....	278
Slot.....	298
Slot center.....	214
Slot undercut.....	310
Sphere.....	287
Touch probe axis.....	265
Undercut position.....	304

B

Basic rotation.....	144
Over two holes.....	148
over two studs.....	153, 158
Setting directly.....	163

C

Calibration	
L-shaped stylus.....	102
Simple stylus.....	102
Tool touch probe.....	117
Workpiece touch probe.....	100
Calibration of tool touch probe	
Calibration of IR TT.....	120
Calibration of TT.....	118
Calibration of workpiece touch probe	
Length calibration.....	109
Radius calibration at the sphere.....	102
Radius calibration with ring...	111
Radius calibration with stud..	114
Classification of results.....	321

Contact.....	33
Context-sensitive help.....	31
Control's user interface.....	49

D

Determining workpiece misalignment	
Basic rotation.....	144
Basic rotation over two holes	148
Basic rotation over two studs.....	153, 158
Fundamentals of touch probe cycles 400-405.....	143
Inclined edge probing.....	185
Intersection probing.....	192
Probing in plane.....	200
Probing on edge.....	170
Probing two circles.....	177
Rotation around C axis.....	165
Setting the basic rotation.....	163

E

Export version.....	41
Extrusion probing.....	388

F

Fast probing.....	385
First steps.....	53
Programming.....	54

G

GLOBAL DEF.....	94
GUI.....	49

I

Integrated product aid TNCguide.....	28
--------------------------------------	----

K

Kinematics measurement	
Accuracy.....	436
Backlash.....	436
Fundamentals.....	422
Hirth coupling.....	433
Kinematics grid.....	457
Preset compensation.....	445
Storing kinematics.....	425

L

Licensing terms.....	48
----------------------	----

M

Measuring	
Angle.....	327
Bolt hole circle.....	363
Circle outside.....	335
Coordinate.....	358
Hole.....	330
Inside width.....	350

Plane.....	367
Rectangle inside.....	340
Rectangle outside.....	345
Ridge width.....	354
Measuring in 3D.....	377
Measuring with Cycle 3.....	374

N

Notes, types of.....	25
----------------------	----

O

Operating mode	
Machine.....	51
Manual.....	51
Overview.....	51
Start.....	51
Operational cycle parameter.....	75

P

Place of operation.....	38
Positioning logic.....	88
Probing in 3D.....	379
Programming with variables.....	93
Proper and intended operation.....	36

R

Recording measurement results.....	319
------------------------------------	-----

S

Safety precaution.....	39
Content.....	25
Selection function	
NC program as cycle.....	82
Software number.....	41
Software option.....	42

T

Target group.....	22
TNCguide.....	29
Tolerance monitoring.....	321
Tool compensation.....	321
Tool measurement	
Complete measurement.....	410
Fundamentals.....	398
Machine parameters.....	399
Tool length.....	403
Tool radius.....	406
Tool table.....	401
Turning tools, measuring.....	414
Touch probe cycles 14xx	
Circle probing.....	282
Inclined edge probing.....	185
Intersection probing.....	192
Island probing.....	292
Pocket probing.....	292
Position probing.....	278
Probing in plane.....	200
Probing on edge.....	170

- Probing two circles..... 177
- Ridge probing..... 298
- Ridge undercut probing..... 310
- Slot probing..... 298
- Slot undercut probing..... 310
- Sphere probing..... 287
- Undercut position probing..... 304
- Touch probe cycles for the workpiece
 - Checking the workpiece..... 319
 - Determining the preset..... 212
 - Probing a position in the plane or in space..... 374
- Touch probe cycles for tool
 - Measurement of milling cutter..... 403
- Touch probe cycles for tools
 - Lathe tools, measuring..... 414
- Touch probe cycles for workpieces
 - Determining the misalignment..... 143

U

- User interface of the control..... 49
- User's Manuals..... 23

V

- Variable..... 93

W

- Workpiece, checking automatically
 - Fundamentals..... 319
 - Polar preset..... 325
- Workpiece touch probe cycles
 - Influencing cycle runs..... 385

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany

☎ +49 8669 31-0

FAX +49 8669 32-5061

info@heidenhain.de

Technical support FAX +49 8669 32-1000

Measuring systems ☎ +49 8669 31-3104
service.ms-support@heidenhain.de

NC support ☎ +49 8669 31-3101
service.nc-support@heidenhain.de

NC programming ☎ +49 8669 31-3103
service.nc-pgm@heidenhain.de

PLC programming ☎ +49 8669 31-3102
service.plc@heidenhain.de

APP programming ☎ +49 8669 31-3106
service.app@heidenhain.de

www.heidenhain.com

www.klartext-portal.com

The Information Site for
HEIDENHAIN Controls

Klartext App

Klartext on your
mobile device

Google
Play Store

Apple
App Store



Touch probes and vision systems

HEIDENHAIN provides universal, high-precision touch probe systems for machine tools, for example for the exact determination of workpiece edge positions and for tool measurement. Proven technology, such as a wear-free optical sensor, collision protection, or integrated blower/flusher jets for cleaning the measuring point ensure the reliability and safety of the touch probes when measuring workpieces and tools. For even higher process reliability, the tools can be monitored conveniently with the vision systems and tool-breakage sensor from HEIDENHAIN.



For more details on touch probes and vision systems:

www.heidenhain.com/products/touch-probes-and-vision-systems

