HEIDENHAIN

User's Manual
Touch Probe Cycles

TNC 320

NC Software
340 551-03
340 554-03

English (en)
7/2008

TNC Model, Software and Features

This manual describes functions and features provided by TNCs as of the following NC software numbers.

TNC model	NC software number
TNC 320	340 551-03
TNC 320 programming station	340 554-03

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

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

- Tool measurement with the TT

Please contact your machine tool builder to become familiar with the features of your machine.

Many machine manufacturers, as well as HEIDENHAIN, offer programming courses for the TNCs. We recommend these courses as an effective way of improving your programming skill and sharing information and ideas with other TNC users.

User's Manual:

All TNC functions that have no connection with touch probes are described in the User's Manual of the TNC 320. Please contact HEIDENHAIN if you need a copy of this User's Manual. ID 550 671-xx

Software options

The TNC features various software options that can be enabled by your machine tool builder. Each option is to be enabled separately and contains the following respective functions:

Software option 1
Cylinder surface interpolation (Cycles 27, 28 and 39)
Feed rate in mm/min on rotary axes: **M116**
Tilting the machining plane (Cycle 19 and 3-D ROT soft key in the manual operating mode)
Circle in 3 axes with tilted working plane

Feature content level (upgrade functions)

Along with software options, significant further improvements of the TNC software are managed via the **F**eature **C**ontent **L**evel upgrade functions. Functions subject to the FCL are not available simply by updating the software on your TNC.

All upgrade functions are available to you without surcharge when you receive a new machine.

Upgrade functions are identified in the manual with **FCL n,** where **n** indicates the sequential number of the feature content level.

You can purchase a code number in order to permanently enable the FCL functions. For more information, contact your machine tool builder or HEIDENHAIN.

Intended place of operation

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

New functions with 340 55x-03

- The TNC now also supports datum administration via the preset table (see "Writing the measured values from touch probe cycles in the preset table," page 28)
- The TNC now also supports the tilting of working planes on machine tools with swivel heads and tilting tables
- Cycle 240 centering
- Cycle 208 helical finish milling: milling method (climb milling / conventional up-cut milling) can now be selected
- Cycle 209 tapping with chip breaking: now with rapid retraction
- Touch probe cycles 400 to 405 for automatic measurement and compensation of workpiece misalignment newly added (see Touch Probe Cycles User's Manual)
- Touch probe cycles 408 to 419 for automatic datum setting newly added (see Touch Probe Cycles User's Manual)
- Touch probe cycles 420 to 431 for automatic tool measurement newly added (see Touch Probe Cycles User's Manual)
- Touch probe cycles 480 (30) to 483 (33) for automatic tool measurement newly added (see Touch Probe Cycles User's Manual)
- Cycle 19 WORKING PLANE and 3D-ROT soft key
- Backspace dialog key

Contents

Introduction 1

Touch Probe Cycles in the Manual and Electronic Handwheel Modes 2

Touch Probe Cycles for Automatic Workpiece Inspection 3

Touch Probe Cycles for Automatic Tool Measurement 4

1 Working with Touch Probe Cycles 15

1.1 General Information on Touch Probe Cycles 16
Method of function 16
Consider a basic rotation in the Manual Operation mode 16
Touch probe cycles in the Manual and Electronic Handwheel modes 16
Touch probe cycles for automatic operation 17
1.2 Before You Start Working with Touch Probe Cycles 19
Maximum traverse to touch point: DIST in touch probe table 19
Set-up clearance to touch point: SET_UP in touch probe table 19
Orient the infrared touch probe to the programmed probe direction: TRACK in touch probe table 19
Touch trigger probe, probing feed rate: F in touch probe table 20
Touch trigger probe, rapid traverse for positioning: FMAX 20
Touch trigger probe, rapid traverse for positioning: F_PREPOS in touch probe table 20
Multiple measurements 20
Confidence range for multiple measurement 20
Running touch probe cycles 21
1.3 Touch probe table 22
General 22
Editing touch probe tables 22
Touch probe data 23

2 Touch Probe Cycles in the Manual and Electronic Handwheel Modes 25

2.1 Introduction 26
Overview 26
Selecting probe cycles 26
Writing the measured values from touch probe cycles in datum tables 27
Writing the measured values from touch probe cycles in the preset table 28
2.2 Calibrating a touch trigger probe 29
Introduction 29
Calibrating the effective length 29
Calibrating the effective radius and compensating center misalignment 30
Displaying calibration values 31
2.3 Compensating Workpiece Misalignment 32
Introduction 32
Measuring the basic rotation 32
Saving the basic rotation in the preset table 33
Displaying a basic rotation 33
To cancel a basic rotation 33
2.4 Setting the Datum with a 3-D Touch Probe 34
Introduction 34
Datum setting in any axis 34
Corner as datum 35
Circle center as datum 36
2.5 Measuring Workpieces with a 3-D Touch Probe 37
Introduction 37
To find the coordinate of a position on an aligned workpiece: 37
Finding the coordinates of a corner in the working plane 37
To measure workpiece dimensions 38
To find the angle between the angle reference axis and a side of the workpiece 39

3 Touch Probe Cycles for Automatic Workpiece Inspection 41

3.1 Measuring Workpiece Misalignment 42
Overview 42
Characteristics common to all touch probe cycles for measuring workpiece misalignment 43
BASIC ROTATION (touch probe cycle 400, DIN/ISO: G400) 44
BASIC ROTATION from two holes (touch probe cycle 401, DIN/ISO: G401) 46
BASIC ROTATION over two studs (touch probe cycle 402, DIN/ISO: G402) 49
BASIC ROTATION compensation via rotary axis (touch probe cycles 403, DIN/ISO: G403) 52
Setting a BASIC ROTATION (touch probe cycle 404, DIN/ISO: G404) 56
Compensating workpiece misalignment by rotating the C axis (touch probe cycle 405, DIN/ISO: G405) 57
3.2 Presetting automatically 61
Overview 61
Characteristics common to all touch probe cycles for datum setting 63
DATUM SLOT CENTER (touch probe Cycle 408, DIN/ISO: G408) 65
DATUM RIDGE CENTER (touch probe cycle 409, DIN/ISO: G409) 68
DATUM FROM INSIDE OF RECTANGLE (touch probe cycle 410, DIN/ISO: G410) 71
DATUM FROM OUTSIDE OF RECTANGLE (touch probe cycle 411, DIN/ISO: G411) 74
DATUM FROM INSIDE OF CIRCLE (touch probe cycle 412, DIN/ISO: G412) 76
DATUM FROM OUTSIDE OF CIRCLE (touch probe cycle 413, DIN/ISO: G413) 81
DATUM FROM OUTSIDE OF CORNER (touch probe cycle 414, DIN/ISO: G414) 85
DATUM FROM INSIDE OF CORNER (touch probe cycle 415, DIN/ISO: G415) 88
DATUM CIRCLE CENTER (touch probe cycle 416, DIN/ISO: G416) 91
DATUM IN TOUCH PROBE AXIS (touch probe cycle 417, DIN/ISO: G417) 94
Datum at center of 4 holes (touch probe cycle 418, DIN/ISO: G418) 96
DATUM IN ONE AXIS (touch probe cycle 419, DIN/ISO: G419) 99

3.3 Automatic Workpiece Measurement 105
Overview 105
Recording the results of measurement 106
Measurement results in Q parameters 107
Classification of results 107
Tolerance monitoring 108
Tool monitoring 108
Reference system for measurement results 109
REFERENCE PLANE (touch probe cycle 0, DIN/ISO: G55) 110
DATUM PLANE (touch probe cycle 1) 111
MEASURE ANGLE (touch probe cycle 420, DIN/ISO: G420) 112
MEASURE HOLE (touch probe cycle 421, DIN/ISO: G421) 114
MEASURE OUTER WIDTH (touch probe cycle 422, DIN/ISO: G422) 117
MEASURE INSIDE RECTANGLE (touch probe cycle 423, DIN/ISO: G423) 120
MEASURE RECTANGLE FROM OUTSIDE (touch probe cycle 424, ISO: G424) 123
MEASURE INSIDE WIDTH (touch probe cycle 425, DIN/ISO: G425) 126
MEASURE OUTER RIDGE (touch probe cycle 426, DIN/ISO: G426) 128
MEASURE COORDINATE (touch probe cycle 427, DIN/ISO: G427) 130
MEAS. BOLT HOLE CIRC (touch probe cycle 430, DIN/ISO: G430) 133
MEASURE PLANE (touch probe cycle 431, DIN/ISO: G431) 136
3.4 Special Cycles 143
Overview 143
MEASURING (touch probe cycle 3) 144

4 Touch Probe Cycles for Automatic Tool Measurement 147

4.1 Tool Measurement with the TT Tool Touch Probe 148
Overview 148
Setting the machine parameters 148
Entries in the tool table TOOL.T 150
4.2 Available Cycles 152
Overview 152
Differences between Cycles 31 to 33 and Cycles 481 to 483 152
Calibrating the TT (touch probe cycle 30 or 480, DIN/ISO: G480) 153
Measuring the tool length (touch probe cycle 31 or 481, DIN/ISO: G481) 154
Measuring the tool radius (touch probe cycle 32 or 482, ISO: G482) 157
Measuring the tool radius (touch probe cycle 33 or 483, ISO: G483) 159

1

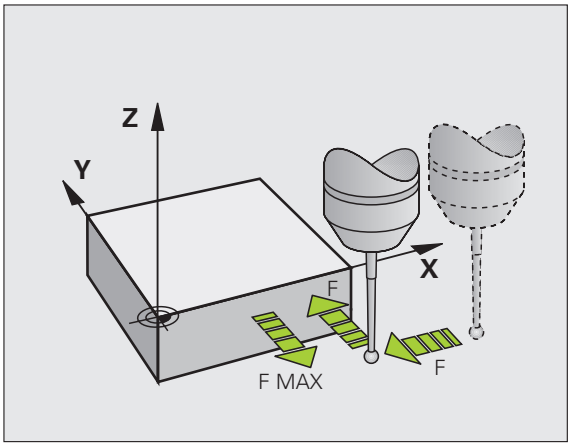
Working with Touch Probe Cycles

1.1 General Information on Touch Probe Cycles

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

Method of function

Whenever the TNC executes a touch probe cycle, the 3-D touch probe approaches with a probing feed rate determined by the machine tool builder in a direction determined by you. The probing feed rate is defined in a machine parameter (see "Before You Start Working with Touch Probe Cycles" later in this chapter).



When the probe stylus contacts the workpiece,

- the 3-D touch probe transmits a signal to the TNC: 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 TNC displays an error message (distance: **DIST** from touch probe table).

Consider a basic rotation in the Manual Operation mode

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

Touch probe cycles in the Manual and Electronic Handwheel modes

In the Manual Operation and El. Handwheel modes, the TNC provides touch probe cycles that allow you to:

- Calibrate the touch probe
- Compensate workpiece misalignment
- Workpiece presetting

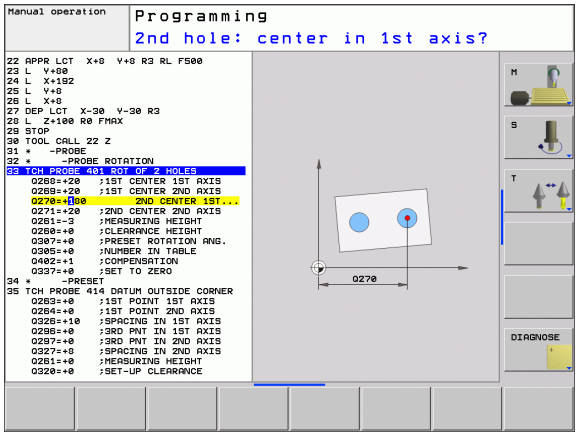
Touch probe cycles for automatic operation

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

- Calibrating the touch probe (Chapter 3)
- Compensating workpiece misalignment (Chapter 3)
- Setting reference points (Chapter 3)
- Automatic workpiece inspection (Chapter 3)
- Automatic workpiece measurement (Chapter 4)

You can program the touch probe cycles in the Programming operating mode via the TOUCH PROBE KEY. Like the most recent fixed cycles, touch probe cycles use Q parameters with numbers of 400 and above as transfer parameters. Parameters with specific functions that are required in several cycles always have the same number: For example, Q260 is always assigned the clearance height, Q261 the measuring height, etc.

To simplify programming, the TNC shows a graphic during cycle definition. In the graphic, the parameter that needs to be entered is highlighted.



Defining the touch probe cycle in the Programming mode of operation

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

- Select the desired probe cycle, for example datum setting. Cycles for automatic tool measurement are available only if your machine has been prepared for them.

- Select a cycle, e.g. datum setting at pocket. The TNC initiates the programming dialog and asks for all required input values. At the same time a graphic of the input parameters is displayed in the right screen window. The parameter that is asked for in the dialog prompt is highlighted.
- Enter all parameters requested by the TNC and conclude each entry with the ENT key.
- The TNC ends the dialog when all required data has been entered.

Group of measuring cycles	Soft key	Page
Cycles for automatic measurement and compensation of workpiece misalignment		Page 42
Cycles for automatic workpiece presetting		Page 61
Cycles for automatic workpiece inspection		Page 105
Special cycles		Page 143
Cycles for automatic tool measurement (enabled by the machine tool builder)		Page 148

Example: NC blocks

```
5 TCH PROBE 410 DATUM INSIDE RECTAN.
     Q321=+50      ;CENTER 1ST AXIS
     Q322=+50      ;CENTER 2ND AXIS
     Q323=60       ;FIRST SIDE LENGTH
     Q324=20       ;SECOND SIDE LENGTH
     Q261=-5       ;MEASURING HEIGHT
     Q320=0        ;SET-UP CLEARANCE
     Q260=+20      ;CLEARANCE HEIGHT
     Q301=0        ;MOVE TO CLEARANCE
     Q305=10       ;NO. IN TABLE
     Q331=+0       ;DATUM
     Q332=+0       ;DATUM
     Q303=+1       ;MEAS. VALUE TRANSFER
     Q381=1        ;PROBE IN TS AXIS
     Q382=+85      ;1ST CO. FOR TS AXIS
     Q383=+50      ;2ND CO. FOR TS AXIS
     Q384=+0       ;3RD CO. FOR TS AXIS
     Q333=+0       ;DATUM
```

1.2 Before You Start Working with Touch Probe Cycles

To make it possible to cover the widest possible range of applications for measuring tasks, machine parameters enable you to determine the behavior common to all touch probe cycles. If you run several touch probes on your machine tool, these settings are globally valid for all touch probes.

In addition, settings are available in the touch probe table that you can separately define for each touch probe. These settings allow you to adapt the behavior of each touch probe or a specific application (see "Touch probe table" on page 22).

Maximum traverse to touch point: DIST in touch probe table

If the stylus is not deflected within the path defined in DIST, the TNC outputs an error message.

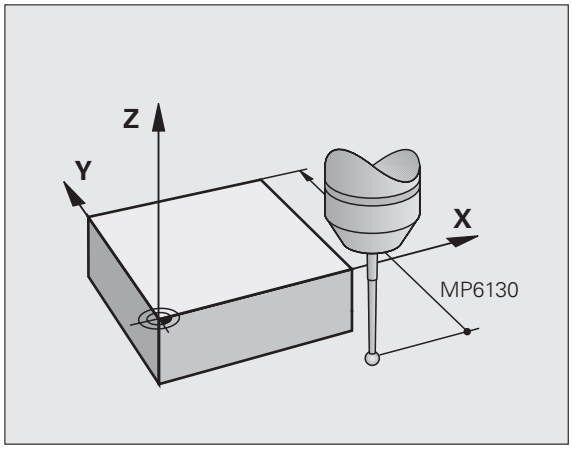
Set-up clearance to touch point: SET_UP in touch probe table

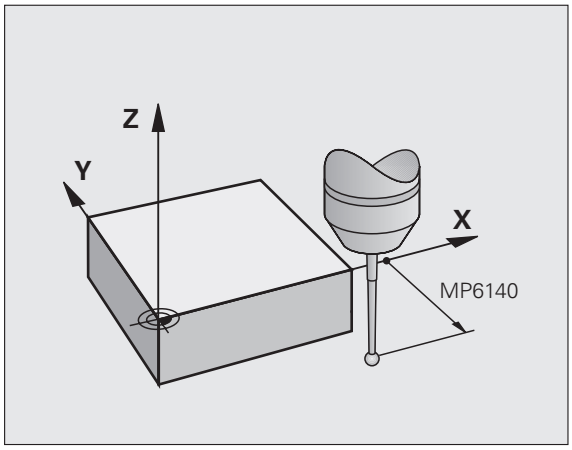
In SET_UP you define how far from the defined (or calculated) touch point the TNC is to pre-position the touch probe. The smaller the value you enter, the more exactly must you define the touch point position. In many touch probe cycles you can also define a setup clearance that is added to SET_UP.

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

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

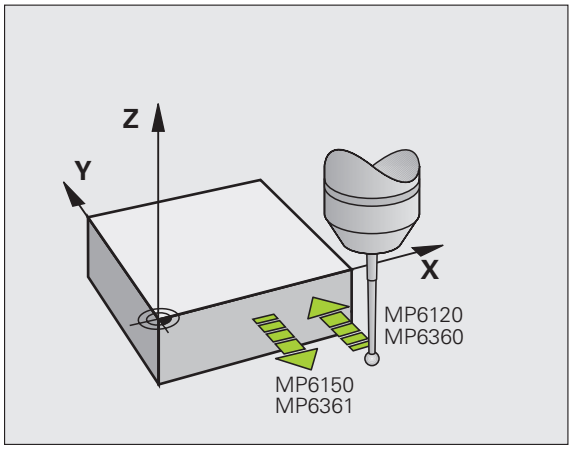
If you change TRACK = ON, you must recalibrate the touch probe.





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

In F you define the feed rate at which the TNC is to probe the workpiece.



Touch trigger probe, rapid traverse for positioning: FMAX

In FMAX you define the feed rate at which the TNC pre-positions the touch probe, or positions it between measuring points.

Touch trigger probe, rapid traverse for positioning: F_PREPOS in touch probe table

In F_PREPOS you define whether the TNC is to position the touch probe at the feed rate defined in FMAX or at rapid traverse.

- Input value = FMAX_PROBE: Position at feed rate from FMAX
- Input value = FMAX_MACHINE: Pre-position at rapid traverse

Multiple measurements

To increase measuring certainty, the TNC can run each probing process up to three times in sequence. Define the number of measurements in machine parameter **Probe Settings** > **Configuration of probe behavior** > **Automatic mode: Multiple measurements with probe function**. If the measured position values differ too greatly, the TNC outputs an error message (the limit value is defined in **confidence range for multiple measurement**). With multiple measurement it is possible to detect random errors, e.g., from contamination.

If the measured values lie within the confidence interval, the TNC saves the mean value of the measured positions.

Confidence range for multiple measurement

When you perform a multiple measurement, you store the value that the measured values may vary in **Probe Settings** > **Configuration of probe behavior** > **Automatic mode: Confidence range for multiple measurement**. If the difference in the measured values exceeds the value defined by you, the TNC outputs an error message.

Running touch probe cycles

All touch probe cycles are DEF active. This means that the TNC runs the cycle automatically as soon as the TNC executes the cycle definition in the program run.

When running touch probe cycles, no cycles must be active for conversion of coordinates (Cycle 7 DATUM, Cycle 8 REFLECTION, Cycle 10 ROTATION, Cycles 11 and 26 SIZE FACTOR and Cycle 19 WORKING PLANE).

You can also run the touch probe cycles 408 to 419 during an active basic rotation. Make sure, however, that the basic rotation angle does not change when you use Cycle 7 DATUM SHIFT with datum tables after the measuring cycle.

Touch probe cycles with a number greater than 400 position the touch probe according to a positioning logic:

- If the current coordinate of the south pole of the stylus is less than the coordinate of the clearance height (defined in the cycle), the TNC retracts the touch probe in the probe axis to the clearance height and then positions it in the working plane to the first starting position.
- If the current coordinate of the south pole of the stylus is greater than the coordinate of the clearance height, the TNC first positions the probe in the working plane to the first starting position and then moves it immediately to the measuring height in the touch probe axis.

1.3 Touch probe table

General

Various data is stored in the touch probe table that defines behavior with the probe process. If you run several touch probes on your machine tool, you can save separate data for each touch probe.

Editing touch probe tables

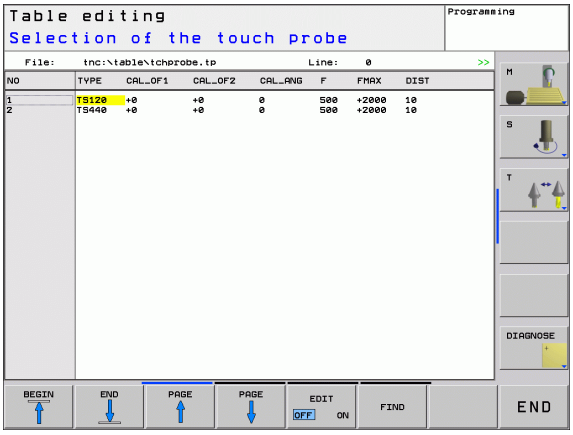
To edit the touch probe table, proceed as follows:

- Select the Manual operating mode

- To select the touch probe functions, press the PROBE FUNCTION soft key. The TNC displays additional soft keys — see table above.

- Select the touch probe table: Press the TOUCH PROBE TABLE soft key.

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



Touch probe data

Abbr.	Inputs	Dialog
NO	Number of the touch probe: Enter this number in the tool table (column: **TP_NO**) under the appropriate tool number	–
TYPE	Selection of the touch probe used	**Selection of touch probe?**
CAL_OF1	Offset of the touch probe axis to the spindle axis for the reference axis	**TS center misalignmt. ref. axis? [mm]**
CAL_OF2	Offset of the touch probe axis to the spindle axis for the minor axis	**TS center misalignmt. ref. axis? [mm]**
CAL_ANG	The TNC orients the touch probe to the orientation angle before calibration or probing (if orientation is possible)	**Spindle angle for calibration?**
F	Feed rate at which the TNC is to probe the workpiece.	**Probing feed rate? [mm/min]**
FMAX	Feed rate at which the touch probe pre-positions, or is positioned between the measuring points	**Rapid traverse in probing cycle? [mm/min]**
DIST	If the stylus is not deflected within the defined path, the TNC outputs an error message	**Maximum measuring path? [mm]**
SET_UP	In **SET_UP** you define how far from the defined (or calculated) touch point the TNC is to pre-position the touch probe. The smaller the value you enter, the more exactly must you define the touch point position. In many touch probe cycles you can also define a setup clearance in addition that is added to Machine Parameter **SET_UP**	**Set-up clearance? [mm]**
F_PREPOS	Defining speed with pre-positioning: ■ Pre-positioning with speed from **FMAX**: **FMAX_PROBE** ■ Pre-positioning with machine rapid traverse: **FMAX_MACHINE**	**Pre-positioning at rap. traverse? ENT/NO ENT**
TRACK	To increase measuring accuracy, you can use **TRACK = ON** to have an infrared touch probe oriented in the programmed probe direction before every probe process. In this way the stylus is always deflected in the same direction: ■ **ON**: Perform spindle tracking ■ **OFF**: Do not perform spindle tracking	**Orient touch probe cycles? Yes=ENT, No=NOENT**

2

Touch Probe Cycles in the Manual and Electronic Handwheel Modes

2.1 Introduction

Overview

The following touch probe cycles are available in the Manual mode:

Function	Soft key	Page
Calibrate the effective length		Page 29
Calibrate the effective radius		Page 30
Measure a basic rotation using a line		Page 32
Set the datum in any axis		Page 34
Set a corner as datum		Page 35
Set a circle center as datum		Page 36
Touch probe system data management		Page 22

Selecting probe cycles

- Select the Manual Operation or Electronic Handwheel mode of operation.

- To select the touch probe functions, press the TOUCH PROBE soft key. The TNC displays additional soft keys—see table above

- To select the probe cycle, press the appropriate soft key, for example PROBING ROT, and the TNC displays the associated menu.

Writing the measured values from touch probe cycles in datum tables

Use this function if you want to save measured values in the workpiece coordinate system. If you want to save measured values in the fixed machine coordinate system (REF coordinates), press the ENTER IN PRESET TABLE soft key (see "Writing the measured values from touch probe cycles in the preset table" on page 28).

With the ENTER IN DATUM TABLE soft key, the TNC can write the values measured after any touch probe cycle into a datum table active for machine operation:

- Select any probe function
- Enter the desired coordinates of the datum in the appropriate input boxes (depends on the touch probe cycle being run).
- Enter the datum number in the **Number in table=** input box.
- Enter the name of the datum table (complete path) in the **Datum table** input box.
- Press the ENTER IN DATUM TABLE soft key. The TNC saves the datum in the indicated datum table under the entered number.

Writing the measured values from touch probe cycles in the preset table

Use this function if you want to save measured values in the fixed machine coordinate system (REF coordinates). If you want to save measured values in the workpiece coordinate system, press the ENTER IN DATUM TABLE soft key(see "Writing the measured values from touch probe cycles in datum tables" on page 27).

With the ENTER IN PRESET TABLE soft key, the TNC can write the values measured during a probe cycle in the preset table. The measured values are then stored referenced to the machine-based coordinate system (REF coordinates). The preset table has the name PRESET.PR, and is saved in the directory TNC:\.

- Select any probe function
- Enter the desired coordinates of the datum in the appropriate input boxes (depends on the touch probe cycle being run).
- Enter the preset number in the **Number in table:** input box.
- Press the ENTER IN PRESET TABLE soft key. The TNC saves the datum in the preset table under the entered number.

2.2 Calibrating a touch trigger probe

Introduction

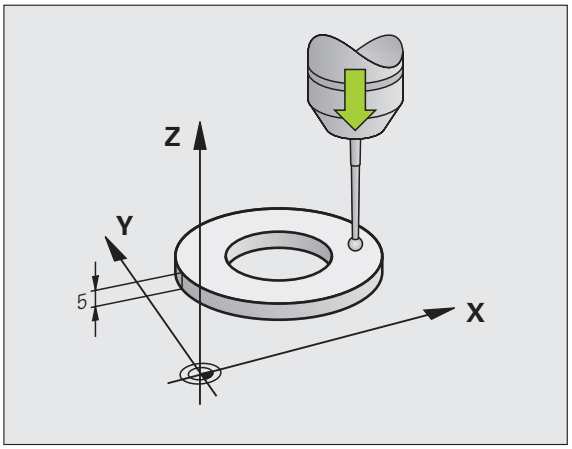
The touch probe must be calibrated in the following cases:

- Commissioning
- Stylus breakage
- Stylus exchange
- Change in the probe feed rate
- Irregularities caused, for example, when the machine heats up
- Changing of touch probe axis

During calibration, the TNC finds the "effective" length of the stylus and the "effective" radius of the ball tip. To calibrate the touch probe, clamp a ring gauge of known height and known internal radius to the machine table.

Calibrating the effective length

The effective length of the touch probe is always referenced to the tool datum. The machine tool builder usually defines the spindle tip as the tool datum.



- Set the datum in the spindle axis such that for the machine tool table Z=0.

- To select the calibration function for the touch probe length, press the TOUCH PROBE and CAL. L soft keys. The TNC then displays a menu window with four input fields
- Enter the tool axis (with the axis key)
- **Datum**: Enter the height of the ring gauge
- **Effective ball radius** and **Effective length** do not require input
- Move the touch probe to a position just above the ring gauge
- To change the traverse direction (if necessary), press a soft key or an arrow key
- To probe the upper surface of the ring gauge, press the machine START button

Calibrating the effective radius and compensating center misalignment

After the touch probe is inserted, it normally needs to be aligned exactly with the spindle axis. The misalignment is measured with this calibration function and compensated electronically.

The TNC rotates the 3-D touch probe by 180° for calibrating the center misalignment. The rotation is initiated by a miscellaneous function that is set by the machine tool builder in Machine Parameter mStrobeUTurn.

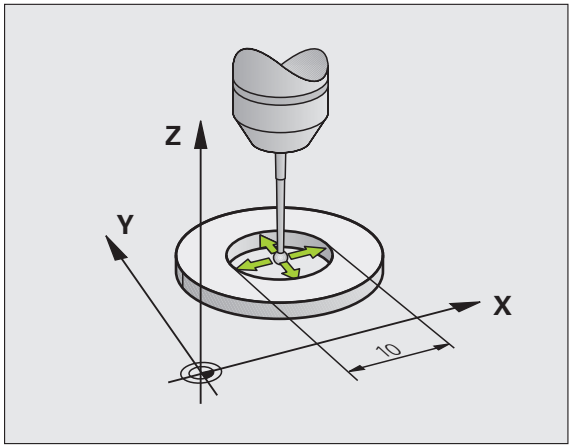
Proceed as follows for manual calibration:

- In the Manual Operation mode, position the ball tip in the bore of the ring gauge.

- To select the calibration function for the ball-tip radius and the touch probe center misalignment, press the CAL. R soft key.
- Select the tool axis and enter the radius of the ring gauge.
- To probe the workpiece, press the machine START button four times. The touch probe contacts a position on the bore in each axis direction and calculates the effective ball-tip radius.
- If you want to terminate the calibration function at this point, press the END soft key.

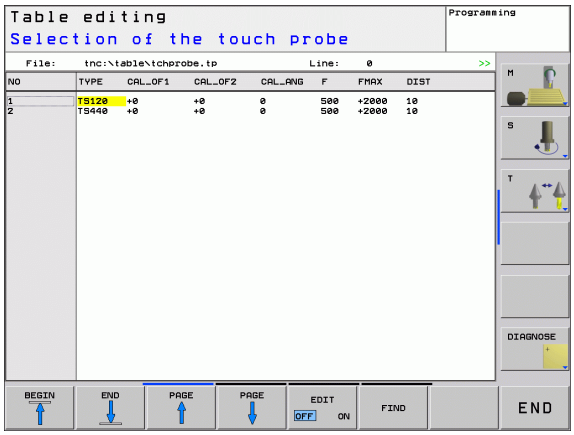
In order to be able to determine ball-tip center misalignment, the TNC needs to be specially prepared by the machine manufacturer. The machine tool manual provides further information.

- If you want to determine the ball-tip center misalignment, press the 180° soft key. The TNC rotates the touch probe by 180°.
- To probe the workpiece, press the machine START button four times. The touch probe contacts a position on the bore in each axis direction and calculates the ball-tip center misalignment.



Displaying calibration values

The TNC saves the effective length and effective radius of the touch probe in the tool table. The TNC saves the ball-tip center misalignment in the touch-probe table, in the CAL_OF1 (main axis) and CAL_OF2 (secondary axis) columns. You can display the values on the screen by pressing the TOUCH-PROBE TABLE soft key.



Make sure that you have activated the correct tool number before using the touch probe, regardless of whether you wish to run the touch probe cycle in automatic mode or manual mode.

The determined calibration values are first calculated after a tool call.

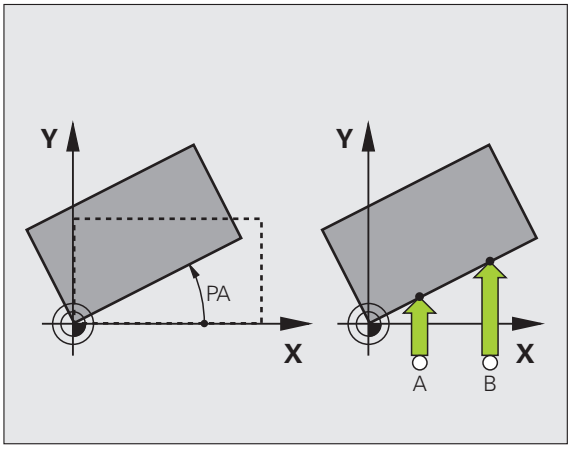
2.3 Compensating Workpiece Misalignment

Introduction

The TNC compensates workpiece misalignment by computing a "basic rotation."

For this purpose, the TNC sets the rotation angle to the desired angle with respect to the reference axis in the working plane. See figure at right.

The TNC saves the basic rotation, depending upon tool axis, in the columns SPA, SPB or SPC of the preset table. .



Select the probe direction perpendicular to the angle reference axis when measuring workpiece misalignment.

To ensure that the basic rotation is calculated correctly during program run, program both coordinates of the working plane in the first positioning block.

Measuring the basic rotation

- Select the probe function by pressing the PROBING ROT soft key.
- Position the ball tip at a starting position near the first touch point.
- Select the probe direction perpendicular to the angle reference axis: Select the axis by soft key.
- To probe the workpiece, press the machine START button.
- Position the ball tip at a starting position near the second touch point.
- To probe the workpiece, press the machine START button. The TNC determines the basic rotation and displays the angle after the dialog **`Rotation angle =`**
- Activate basic rotation: Press the SET BASIC ROTATION soft key
- Terminate the probe function by pressing the END soft key

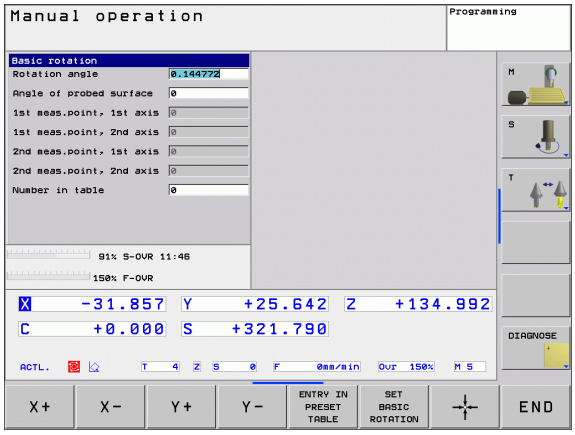
Saving the basic rotation in the preset table

- After the probing process, enter the preset number in which the TNC is to save the active basic rotation in the **Number in table:** input box.
- Press the ENTRY IN PRESET TABLE soft key to save the basic rotation in the preset table.

Displaying a basic rotation

The angle of the basic rotation appears after ROTATION ANGLE whenever PROBING ROT is selected. The TNC also displays the rotation angle in the additional status display (STATUS POS.).

In the status display a symbol is shown for a basic rotation whenever the TNC is moving the axes according to a basic rotation.



To cancel a basic rotation

- Select the probe function by pressing the PROBING ROT soft key.
- Enter rotation range **0**, confirm with SET BASIC ROTATION soft key
- Terminate the probe function by pressing the END soft key

2.4 Setting the Datum with a 3-D Touch Probe

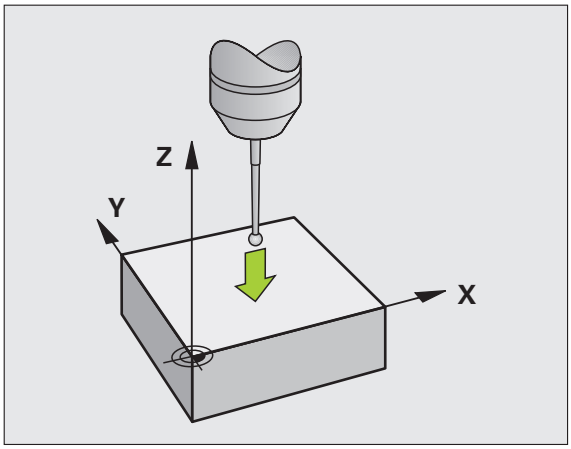
Introduction

The following functions are available for setting the datum on an aligned workpiece:

- Datum setting in any axis with PROBING POS
- Defining a corner as datum with PROBING P
- Setting the datum at a circle center with PROBING CC

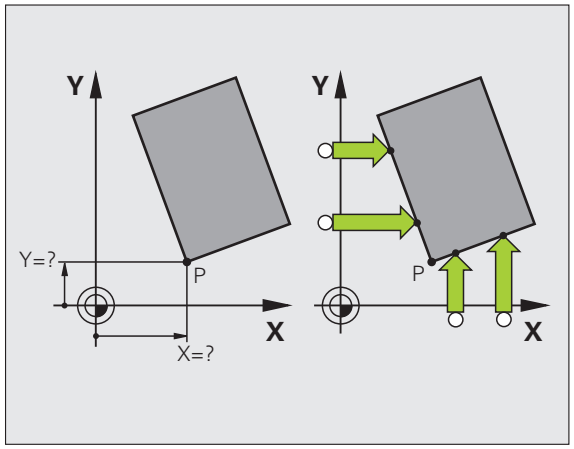
Datum setting in any axis

- Select the probe function by pressing the PROBING POS soft key.
- Move the touch probe to a starting position near the touch point.
- Select the probe axis and direction in which you wish to set the datum, such as Z in direction Z–. Selection is made via soft keys.
- To probe the workpiece, press the machine START button.
- **Datum:** Enter the nominal coordinate and confirm your entry with SET DATUM, or write the value to a table (see "Writing the measured values from touch probe cycles in datum tables," page 27, or see "Writing the measured values from touch probe cycles in the preset table," page 28).
- To terminate the probe function, press the END key.



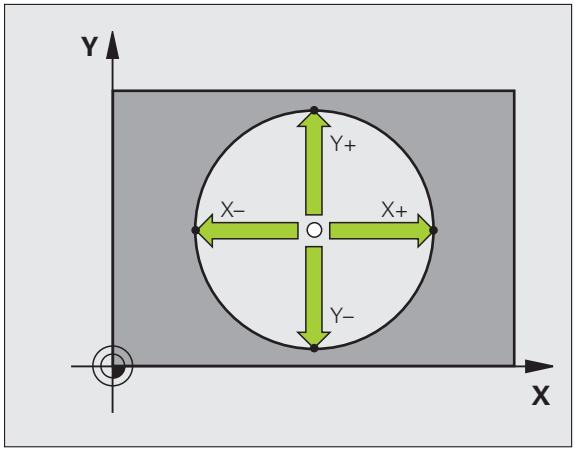
Corner as datum

- Select the probe function by pressing the PROBING P soft key.
- Position the touch probe near the first touch point on the same side of the workpiece.
- Select the probe direction by soft key.
- To probe the workpiece, press the machine START button.
- Position the touch probe near the second touch point on the same side.
- To probe the workpiece, press the machine START button.
- Position the touch probe near the first touch point on the second side of the workpiece.
- Select the probe direction by soft key.
- To probe the workpiece, press the machine START button.
- Position the touch probe near the second touch point on the same side.
- To probe the workpiece, press the machine START button.
- **Datum:** Enter both coordinates of the datum in the menu window, confirm with the SET DATUM soft key, or write the values to a table (see "Writing the measured values from touch probe cycles in datum tables," page 27, or see "Writing the measured values from touch probe cycles in the preset table," page 28).
- To terminate the probe function, press the END key.



Circle center as datum

With this function, you can set the datum at the center of bore holes, circular pockets, cylinders, studs, circular islands, etc.



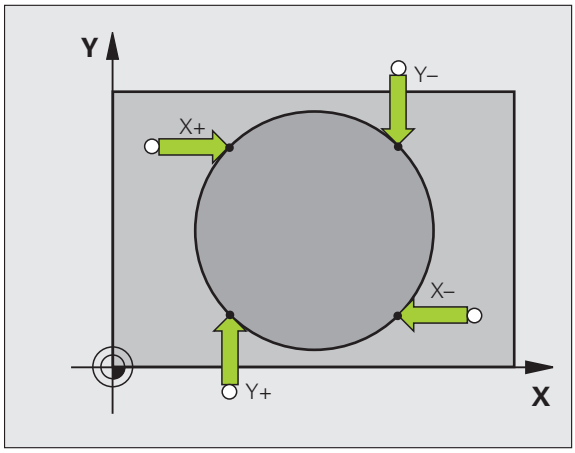
Inside circle:

The TNC automatically probes the inside wall in all four coordinate axis directions.

For incomplete circles (circular arcs) you can choose the appropriate probing direction.

- Position the touch probe approximately in the center of the circle.

- Select the probe function by pressing the PROBING CC soft key
- To probe the workpiece, press the machine START button four times. The touch probe touches four points on the inside of the circle.
- **Datum:** In the menu window, enter both coordinates of the circle center, confirm with the SET DATUM soft key, or write the values to a table (see "Writing the measured values from touch probe cycles in datum tables," page 27, or see "Writing the measured values from touch probe cycles in the preset table," page 28).
- To terminate the probe function, press the END key.



Outside circle

- Position the touch probe at the starting position for the first touch point outside of the circle.
- Select the probe direction by soft key.
- To probe the workpiece, press the machine START button.
- Repeat the probing process for the remaining three points. See figure at lower right.
- **Datum:** Enter the coordinates of the datum and confirm your entry with the SET DATUM soft key, or write the values to a table (see "Writing the measured values from touch probe cycles in datum tables," page 27, or see "Writing the measured values from touch probe cycles in the preset table," page 28).
- To terminate the probe function, press the END key.

After the probing procedure is completed, the TNC displays the coordinates of the circle center and the circle radius PR.

2.5 Measuring Workpieces with a 3-D Touch Probe

Introduction

You can also use the touch probe in the Manual and Electronic Handwheel operating modes to make simple measurements on the workpiece. Numerous programmable probing cycles are available for more complex measuring tasks (see "Automatic Workpiece Measurement" on page 105). With a 3-D touch probe you can determine:

- position coordinates, and from them,
- dimensions and angles on the workpiece.

To find the coordinate of a position on an aligned workpiece:

- Select the probe function by pressing the PROBING POS soft key.
- Move the touch probe to a starting position near the touch point.
- Select the probe direction and axis of the coordinate. Use the corresponding soft keys for selection.
- To probe the workpiece, press the machine START button.

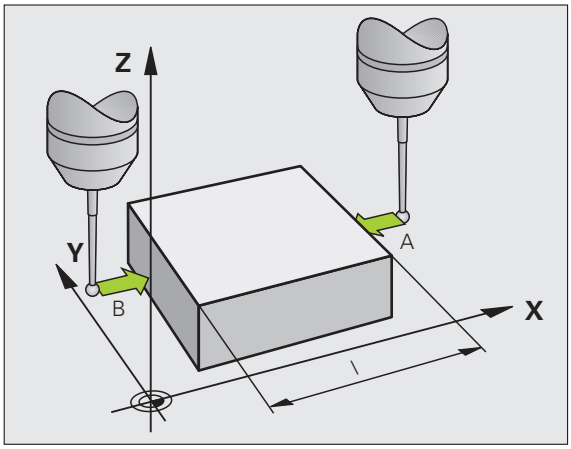
The TNC shows the coordinates of the touch point as reference point.

Finding the coordinates of a corner in the working plane

Find the coordinates of the corner point: See "Corner as datum," page 35. The TNC displays the coordinates of the probed corner as reference point.

To measure workpiece dimensions

- Select the probe function by pressing the PROBING POS soft key.
- Position the touch probe at a starting position near the first touch point A.
- Select the probing direction by soft key.
- To probe the workpiece, press the machine START button.
- If you will need the current datum later, write down the value that appears in the Datum display.
- Datum: Enter “0”.
- To terminate the dialog, press the END key.
- Select the probe function by pressing the PROBING POS soft key.
- Position the touch probe at a starting position near the second touch point B.
- Select the probe direction with the soft keys: Same axis but from the opposite direction.
- To probe the workpiece, press the machine START button.



The value displayed as datum is the distance between the two points on the coordinate axis.

To return to the datum that was active before the length measurement:

- Select the probe function by pressing the PROBING POS soft key.
- Probe the first touch point again.
- Set the datum to the value that you wrote down previously.
- To terminate the dialog, press the END key.

Measuring angles

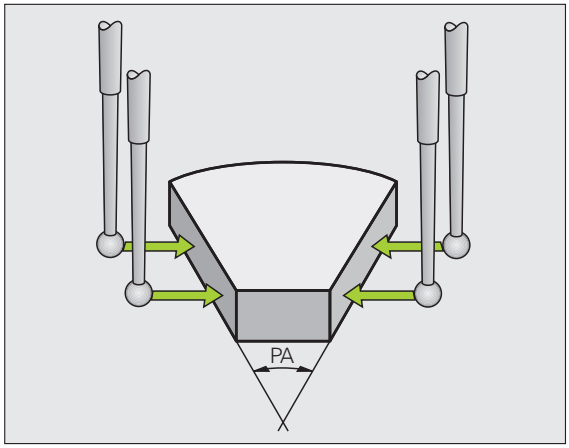
You can use the 3-D touch probe to measure angles in the working plane. You can measure

- the angle between the angle reference axis and a workpiece side, or
- the angle between two sides.

The measured angle is displayed as a value of maximum 90°.

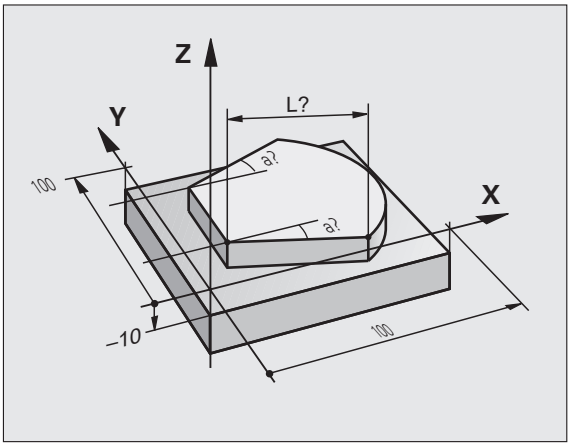
To find the angle between the angle reference axis and a side of the workpiece

- Select the probe function by pressing the PROBING ROT soft key.
- Rotation angle: If you will need the current basic rotation later, write down the value that appears under Rotation angle.
- Make a basic rotation with the side of the workpiece (see "Compensating Workpiece Misalignment" on page 32).
- Press the PROBING ROT soft key to display the angle between the angle reference axis and the side of the workpiece as the rotation angle.
- Cancel the basic rotation, or restore the previous basic rotation.
- This is done by setting the rotation angle to the value that you previously wrote down.



To measure the angle between two workpiece sides:

- Select the probe function by pressing the PROBING ROT soft key.
- Rotation angle: If you will need the current basic rotation later, write down the displayed rotation angle.
- Make a basic rotation with the side of the workpiece (see "Compensating Workpiece Misalignment" on page 32).
- Probe the second side as for a basic rotation, but do not set the rotation angle to zero!
- Press the PROBING ROT soft key to display the angle PA between the two sides as the rotation angle.
- Cancel the basic rotation, or restore the previous basic rotation by setting the rotation angle to the value that you wrote down previously.



3

Touch Probe Cycles for Automatic Workpiece Inspection

3.1 Measuring Workpiece Misalignment

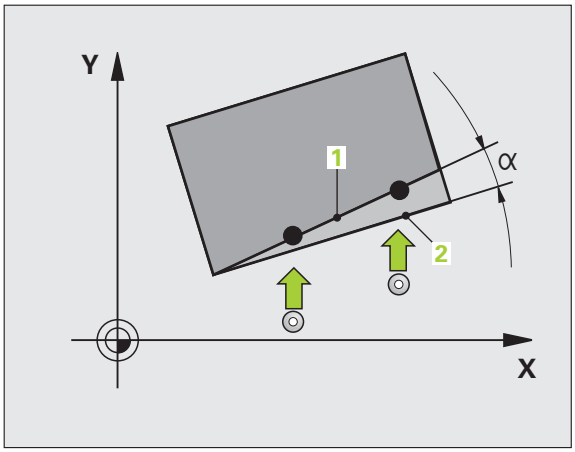
Overview

The TNC provides five cycles that enable you to measure and compensate workpiece misalignment. In addition, you can reset a basic rotation with Cycle 404:

Cycle	Soft key	Page
400 BASIC ROTATION Automatic measurement using two points. Compensation via basic rotation.		Page 44
401 ROT OF 2 HOLES Automatic measurement using two holes. Compensation via basic rotation.		Page 46
402 ROT OF 2 STUDS Automatic measurement using two studs. Compensation via basic rotation.		Page 49
403 ROT IN ROTARY AXIS Automatic measurement using two points. Compensation by turning the table.		Page 52
405 ROT IN C AXIS Automatic alignment of an angular offset between a hole center and the positive Y axis. Compensation via table rotation.		Page 57
404 SET BASIC ROTATION Setting any basic rotation.		Page 56

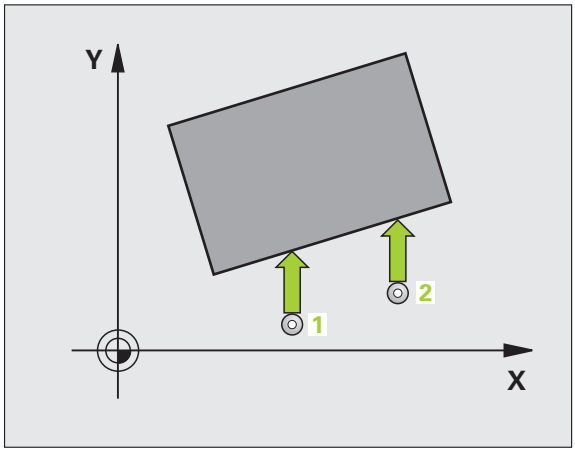
Characteristics common to all touch probe cycles for measuring workpiece misalignment

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



BASIC ROTATION (touch probe cycle 400, DIN/ISO: G400)

Touch probe cycle 400 determines a workpiece misalignment by measuring two points, which must lie on a straight surface. With the basic rotation function the TNC compensates the measured value (See also \gCompensating Workpiece Misalignment" on page 32).



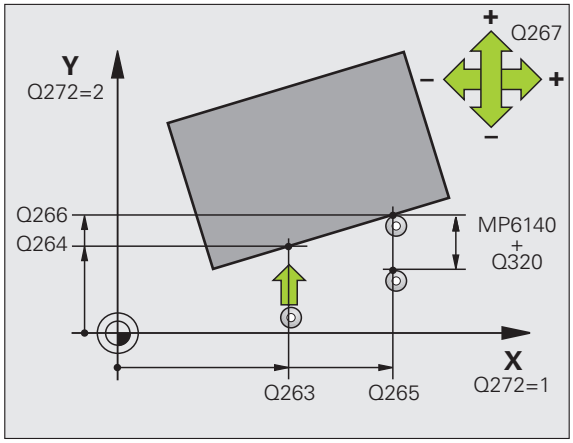
1. The TNC positions the touch probe to the starting points at rapid traverse (value from FMAX column) following the positioning logic (see "Running touch probe cycles" on page 21) to the programmed starting point **1**. The TNC offsets the touch probe by the safety clearance in the direction opposite the defined traverse direction.
2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (column F).
3. Then the touch probe moves to the next starting position **2** and probes the second position.
4. The TNC returns the touch probe to the clearance height and performs the basic rotation.

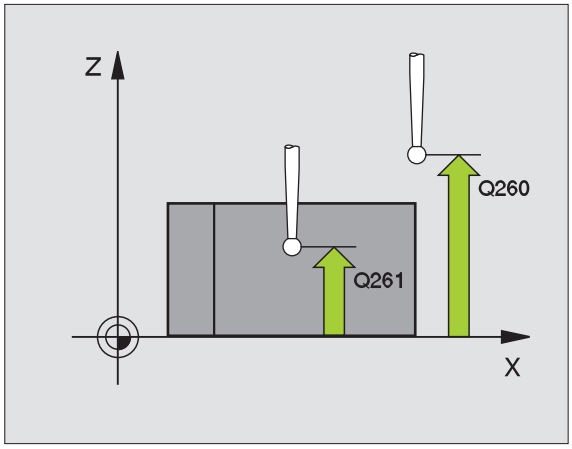
Before programming, note the following

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

The TNC will reset an active basic rotation at the beginning of the cycle.

- **1st measuring point in 1st axis** Q263 (absolute): coordinate of the first touch point in the reference axis of the working plane.
- **1st measuring point in 2nd axis** Q264 (absolute): coordinate of the first touch point in the minor axis of the working plane.
- **2nd measuring point in 1st axis** Q265 (absolute): coordinate of the second touch point in the reference axis of the working plane
- **2nd measuring point in 2nd axis** Q266 (absolute): coordinate of the second touch point in the minor axis of the working plane
- **Measuring axis** Q272: Axis in the working plane in which the measurement is to be made:
  1: Reference axis = measuring axis
  2:Minor axis = measuring axis
- **Traverse direction 1** Q267: Direction in which the probe is to approach the workpiece:
  -1:Negative traverse direction
  +1:Positive traverse direction
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Default setting for basic rotation** Q307 (absolute): If the misalignment is to be measured against a straight line other than the reference axis, enter the angle of this reference line. The TNC will then calculate the difference between the value measured and the angle of the reference line for the basic rotation.
- **Preset number in table** Q305: Enter the preset number in the table in which the TNC is to save the determined basic rotation. If you enter Q305=0, the TNC automatically places the determined basic rotation in the ROT menu of the Manual mode of operation.

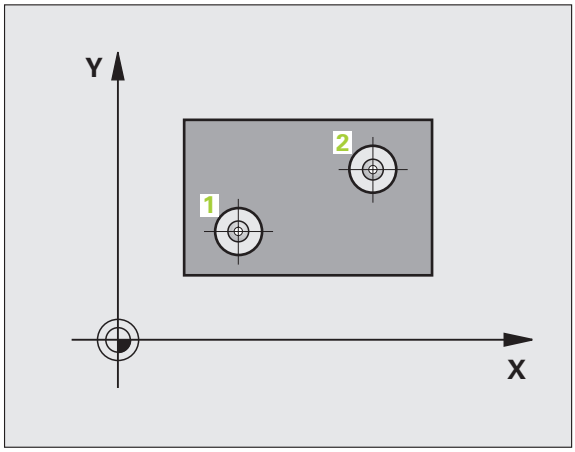




Example: NC blocks

```
5 TCH PROBE 400 BASIC ROTATION
    Q263=+10      ;1ST POINT 1ST AXIS
    Q264=+3.5     ;1ST POINT 2ND AXIS
    Q265=+25      ;2ND POINT 1ST AXIS
    Q266=+2       ;2ND POINT 2ND AXIS
    Q272=2        ;MEASURING AXIS
    Q267=+1       ;TRAVERSE DIRECTION
    Q261=-5       ;MEASURING HEIGHT
    Q320=0        ;SET-UP CLEARANCE
    Q260=+20      ;CLEARANCE HEIGHT
    Q301=0        ;MOVE TO CLEARANCE
    Q307=0        ;PRESET BASIC ROTATION
    Q305=0        ;NO. IN TABLE
```

BASIC ROTATION from two holes (touch probe cycle 401, DIN/ISO: G401)



The touch probe cycle 401 measures the centers of two holes. Then the TNC calculates the angle between the reference axis in the working plane and the line connecting the two hole centers. With the basic rotation function, the TNC compensates the calculated value (See also \gCompensating Workpiece Misalignment" on page 32). As an alternative, you can also compensate the determined misalignment by rotating the rotary table.

1. The TNC positions the touch probe at rapid traverse (value from column FMAX) and with positioning logic (see "Running touch probe cycles" on page 21) to the point entered as center of the first hole 1.
2. Then the probe moves to the entered measuring height and probes four points to find the first hole center.
3. The touch probe returns to the clearance height and then to the position entered as center of the second hole 2
4. The TNC moves the touch probe to the entered measuring height and probes four points to find the second hole center.
5. Then the TNC returns the touch probe to the clearance height and performs the basic rotation.

Before programming, note the following

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

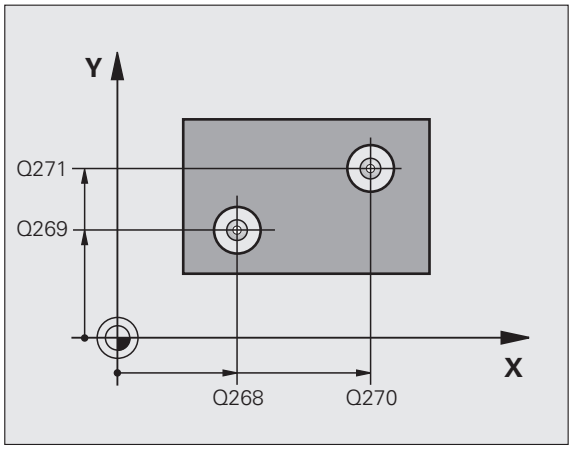
The TNC will reset an active basic rotation at the beginning of the cycle.

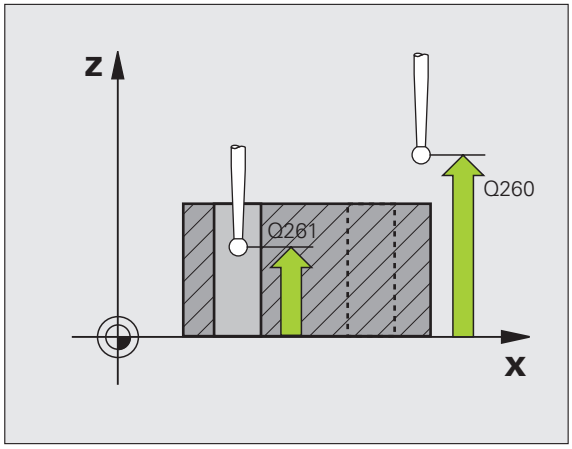
This touch probe cycle is not allowed when the tilted working plane function is active.

If you want to compensate the misalignment by rotating the rotary table, the TNC will automatically use the following rotary axes:

- C for tool axis Z
- B for tool axis Y
- A for tool axis X

- **First hole: Center in 1st axis** Q268 (absolute): center of the first hole in the reference axis of the working plane.
- **First hole: Center in 2nd axis** Q269 (absolute): center of the first hole in the minor axis of the working plane.
- **Second hole: Center in 1st axis** Q270 (absolute): center of the second hole in the reference axis of the working plane.
- **Second hole: Center in 2nd axis** Q271 (absolute): center of the second hole in the minor axis of the working plane.
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Default setting for basic rotation** Q307 (absolute): If the misalignment is to be measured against a straight line other than the reference axis, enter the angle of this reference line. The TNC will then calculate the difference between the value measured and the angle of the reference line for the basic rotation.



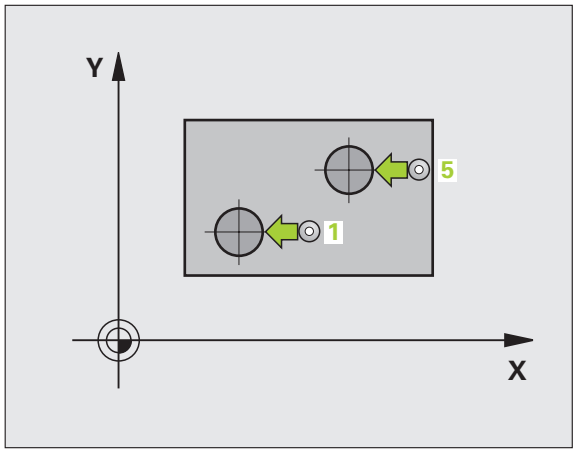


- **Preset number in table** Q305: Enter the preset number in the table in which the TNC is to save the determined basic rotation. If you enter Q305=0, the TNC automatically places the determined basic rotation in the ROT menu of the Manual mode of operation. The parameter has no effect if the misalignment is to be compensated by a rotation of the rotary table (**Q402=1**). In this case the misalignment is not saved as an angular value.
- **Basic rotation / alignment** Q402: Specify whether the TNC should compensate misalignment with a basic rotation, or by rotating the rotary table:
  0: Set basic rotation
  1: Rotate the rotary table
  When you select rotary table, the TNC does not save the measured alignment, not even when you have defined a table line in parameter **Q305.**
- **Set to zero after alignment** Q337: Definition of whether the TNC should set the display of the aligned rotary axis to zero:
  0: Do not reset the display of the rotary axis to 0 after alignment
  1: Reset the display of the rotary axis to 0 after alignment
  The TNC sets the display to 0 only if you have defined **Q402=1.**

Example: NC blocks

```
5 TCH PROBE 401 ROT OF 2 HOLES
    Q268=-37      ;1ST CENTER IN 1ST AXIS
    Q269=+12      ;1ST CENTER IN 2ND AXIS
    Q270=+75      ;2ND CENTER IN 1ST AXIS
    Q271=+20      ;2ND CENTER IN 2ND AXIS
    Q261=-5       ;MEASURING HEIGHT
    Q260=+20      ;CLEARANCE HEIGHT
    Q307=0        ;PRESET BASIC ROTATION
    Q305=0        ;NO. IN TABLE
    Q402=0        ;ALIGNMENT
    Q337=0        ;SET TO ZERO
```

BASIC ROTATION over two studs (touch probe cycle 402, DIN/ISO: G402)



The touch probe cycle 402 measures the centers of two studs. Then the TNC calculates the angle between the reference axis in the working plane and the line connecting the two stud centers. With the basic rotation function, the TNC compensates the calculated value (See also \gCompensating Workpiece Misalignment" on page 32). As an alternative, you can also compensate the determined misalignment by rotating the rotary table.

1. Following the positioning logic (see "Running touch probe cycles" on page 21), the TNC positions the touch probe in rapid traverse (value from column FMAX) to the starting point **1** of the first stud.
2. Then the probe moves to the entered **measuring height 1** and probes four points to find the center of the first stud. The touch probe moves on a circular arc between the touch points, each of which is offset by 90°.
3. The touch probe returns to the clearance height and then to the starting point for probing **5** the second stud.
4. The TNC moves the touch probe to the entered **measuring height 2** and probes four points to find the center of the second stud.
5. Then the TNC returns the touch probe to the clearance height and performs the basic rotation.

Before programming, note the following

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

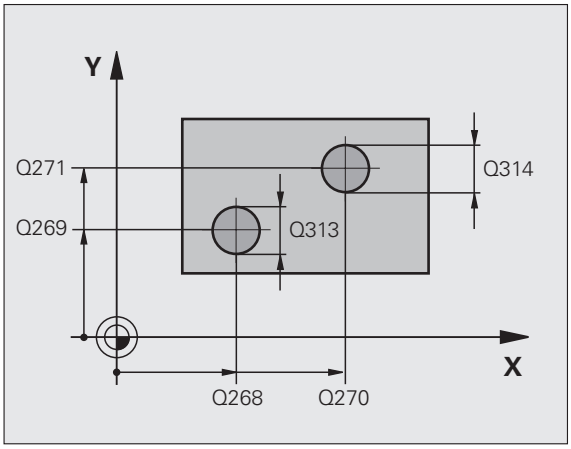
The TNC will reset an active basic rotation at the beginning of the cycle.

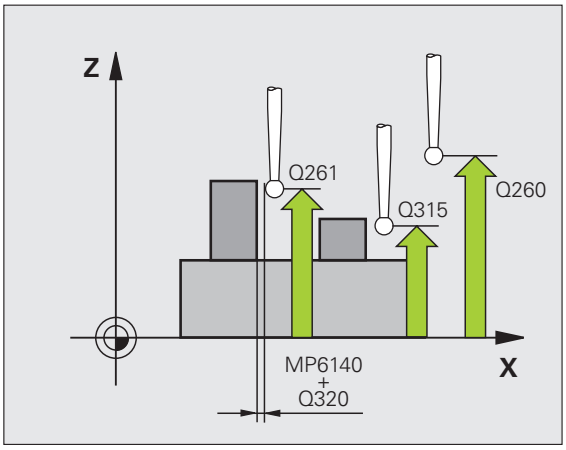
This touch probe cycle is not allowed when the tilted working plane function is active.

If you want to compensate the misalignment by rotating the rotary table, the TNC will automatically use the following rotary axes:

- C for tool axis Z
- B for tool axis Y
- A for tool axis X

- **First stud: Center in 1st axis** (absolute): center of the first stud in the reference axis of the working plane.
- **First stud: Center in 2nd axis** Q269 (absolute): center of the first stud in the minor axis of the working plane.
- **Diameter of stud 1** Q313: Approximate diameter of the 1st stud. Enter a value that is more likely to be too large than too small.
- **Measuring height 1 in the probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point in the touch probe axis) at which stud 1 is to be measured.
- **Second stud: Center in 1st axis** Q270 (absolute): center of the second stud in the reference axis of the working plane.
- **Second stud: Center in 2nd axis** Q271 (absolute): center of the second stud in the minor axis of the working plane.
- **Diameter of stud 2** Q314: Approximate diameter of the 2nd stud. Enter a value that is more likely to be too large than too small.
- **Measuring height 2 in the probe axis** Q315 (absolute): Coordinate of the ball tip center (= touch point in the touch probe axis) at which stud 2 is to be measured.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.





- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Default setting for basic rotation** Q307 (absolute): If the misalignment is to be measured against a straight line other than the reference axis, enter the angle of this reference line. The TNC will then calculate the difference between the value measured and the angle of the reference line for the basic rotation.
- **Preset number in table** Q305: Enter the preset number in the table in which the TNC is to save the determined basic rotation. If you enter Q305=0, the TNC automatically places the determined basic rotation in the ROT menu of the Manual mode of operation. The parameter has no effect if the misalignment is to be compensated by a rotation of the rotary table (**Q402=1**). In this case the misalignment is not saved as an angular value.
- **Basic rotation / alignment** Q402: Specify whether the TNC should compensate misalignment with a basic rotation, or by rotating the rotary table:
  0: Set basic rotation
  1: Rotate the rotary table
  When you select rotary table, the TNC does not save the measured alignment, not even when you have defined a table line in parameter **Q305.**
- **Set to zero after alignment** Q337: Definition of whether the TNC should set the display of the aligned rotary axis to zero:
  0: Do not reset the display of the rotary axis to 0 after alignment
  1: Reset the display of the rotary axis to 0 after alignment
  The TNC sets the display to 0 only if you have defined **Q402=1.**

Example: NC blocks

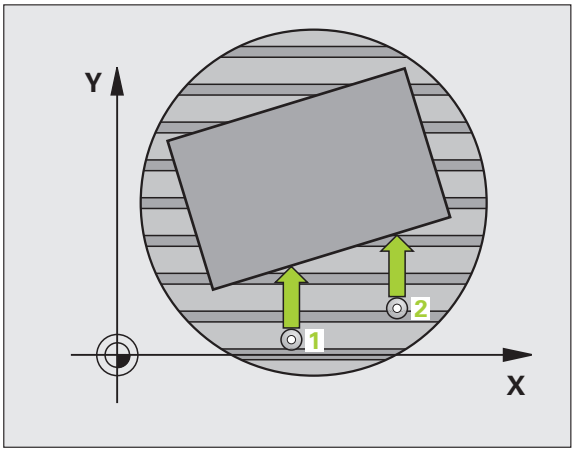
```
5 TCH PROBE 402 ROT OF 2 STUDS
    Q268=-37      ;1ST CENTER IN 1ST AXIS
    Q269=+12      ;1ST CENTER IN 2ND AXIS
    Q313=60       ;DIAMETER OF STUD 1
    Q261=-5       ;MEASURING HEIGHT 1
    Q270=+75      ;2ND CENTER IN 1ST AXIS
    Q271=+20      ;2ND CENTER IN 2ND AXIS
    Q314=60       ;DIAMETER OF STUD 2
    Q315=-5       ;MEASURING HEIGHT 2
    Q320=0        ;SET-UP CLEARANCE
    Q260=+20      ;CLEARANCE HEIGHT
    Q301=0        ;MOVE TO CLEARANCE
    Q307=0        ;PRESET BASIC ROTATION
    Q305=0        ;NO. IN TABLE
    Q402=0        ;ALIGNMENT
    Q337=0        ;SET TO ZERO
```

BASIC ROTATION compensation via rotary axis (touch probe cycles 403, DIN/ISO: G403)

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

The combinations of measuring axis (Cycle Parameter Q272) and compensation axis (Cycle Parameter Q312) listed below are permitted. Function for tilting the working plane:

Active TX axis	Measuring axis	Compensation axis
Z	X (Q272=1)	C (Q312=6)
Z	Y (Q272=2)	C (Q312=6)
Z	Z (Q272=3)	B (Q312=5) or A (Q312=4)
Y	Z (Q272=1)	B (Q312=5)
Y	X (Q272=2)	C (Q312=5)
Y	Y (Q272=3)	C (Q312=6) or A (Q312=4)
X	Y (Q272=1)	A (Q312=4)
X	Z (Q272=2)	A (Q312=4)
X	X (Q272=3)	B (Q312=5) or C (Q312=6)



1. The TNC positions the touch probe to the starting points at rapid traverse (value from FMAX column) following the positioning logic (see "Running touch probe cycles" on page 21) to the programmed starting point **1**. The TNC offsets the touch probe by the safety clearance in the direction opposite the defined traverse direction.
2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (column F).
3. Then the touch probe moves to the next starting position **2** and probes the second position.
4. The TNC returns the touch probe to the clearance height and moves the rotary axis, which was defined in the cycle, by the measured value. Optionally you can have the display set to 0 after alignment.

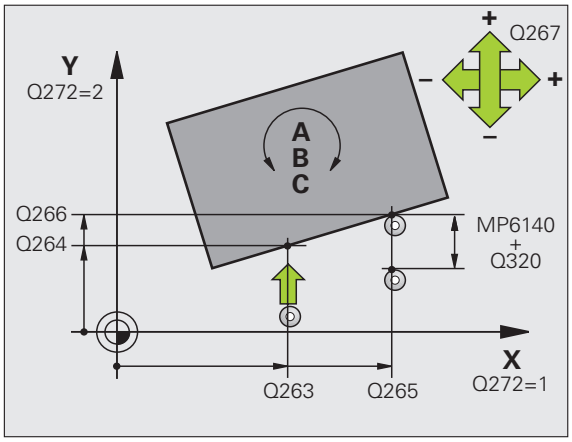
Before programming, note the following

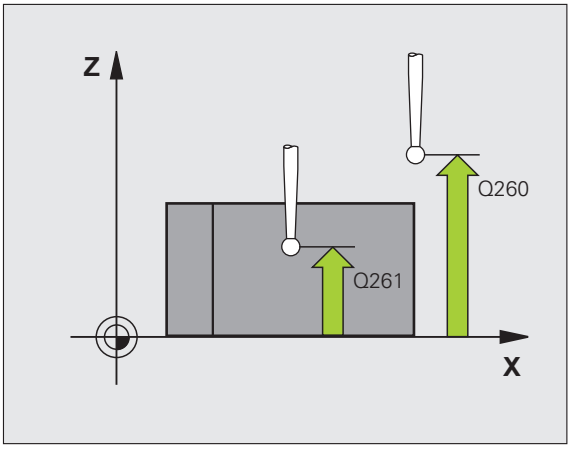
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

Cycle 403 must not be used if the "Tilt working plane" function is active.

The TNC stores the measured angle in parameter **Q150**.

- **1st measuring point in 1st axis** Q263 (absolute): coordinate of the first touch point in the reference axis of the working plane.
- **1st measuring point in 2nd axis** Q264 (absolute): coordinate of the first touch point in the minor axis of the working plane.
- **2nd measuring point in 1st axis** Q265 (absolute): coordinate of the second touch point in the reference axis of the working plane
- **2nd measuring point in 2nd axis** Q266 (absolute): coordinate of the second touch point in the minor axis of the working plane
- **Measuring axis** Q272: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
  3: Touch probe axis = measuring axis
- **Traverse direction 1** Q267: Direction in which the probe is to approach the workpiece:
  -1: Negative traverse direction
  +1: Positive traverse direction
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.





- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- Axis for compensation motion Q312: assignment of the rotary axis in which the TNC is to compensate the measured misalignment:
  4: Compensate misalignment with rotary axis A
  5: Compensate misalignment with rotary axis B
  6: Compensate misalignment with rotary axis C
- **Set to zero after alignment** Q337: Definition of whether the TNC should set the display of the aligned rotary axis to zero:
  0: Do not reset the display of the rotary axis to 0 after alignment
  1:Reset the display of the rotary axis to 0 after alignment
- **Number in table** Q305: Enter the number in the preset table/datum table in which the TNC is to set the rotary axis to zero. Only effective if Q337 is set to 1.
- **Measured-value transfer (0, 1)** Q303: Specify if the determined basic rotation is to be saved in the datum table or in the preset table:
  0: Write the measured basic rotation as a datum shift in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write the measured basic rotation into the preset table. The reference system is the machine coordinate system (REF system).
- **Reference angle? (0=ref. axis)** Q380: Angle with which the TNC is to align the probed straight line. Only effective if the rotary axis C is selected (Q312 = 6)

Example: NC blocks

```
5 TCH PROBE 403 ROT IN C-AXIS
    Q263=+0        ;1ST POINT 1ST AXIS
    Q264=+0        ;1ST POINT 2ND AXIS
    Q265=+20       ;2ND POINT 1ST AXIS
    Q266=+30       ;2ND POINT 2ND AXIS
    Q272=1         ;MEASURING AXIS
    Q267=-1        ;TRAVERSE DIRECTION
    Q261=-5        ;MEASURING HEIGHT
    Q320=0         ;SET-UP CLEARANCE
    Q260=+20       ;CLEARANCE HEIGHT
    Q301=0         ;MOVE TO CLEARANCE
    Q312=6         ;COMPENSATION AXIS
    Q337=0         ;SET TO ZERO
    Q305=1         ;NO. IN TABLE
    Q303=+1        ;MEAS. VALUE TRANSFER
    Q380=+90       ;REFERENCE ANGLE
```

Setting a BASIC ROTATION (touch probe cycle 404, DIN/ISO: G404)

With touch probe cycle 404, you can set any basic rotation automatically during program run. This cycle is intended primarily for resetting a previous basic rotation.

- **Preset value for basic rotation:** Angular value at which the basic rotation is to be set.

Example: NC blocks

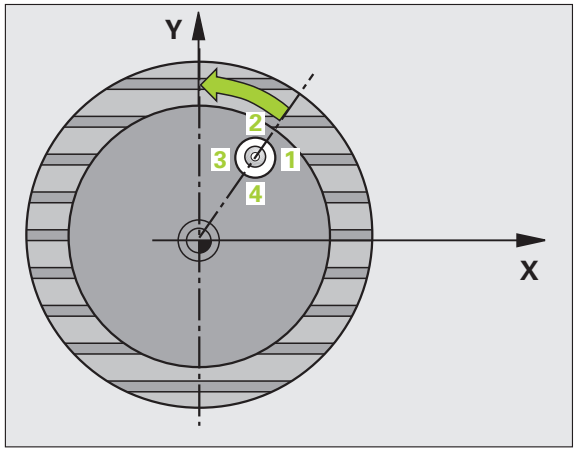
```
5 TCH PROBE 404 BASIC ROTATION
    Q307=+0       ;PRESET BASIC ROTATION
```

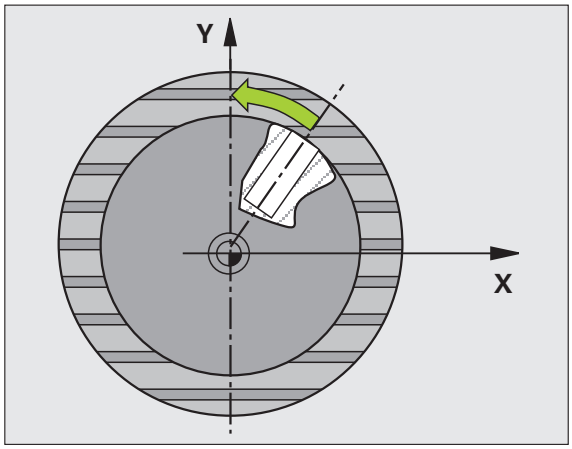
Compensating workpiece misalignment by rotating the C axis (touch probe cycle 405, DIN/ISO: G405)

With touch probe cycle 405, you can measure

- the angular offset between the positive Y axis of the active coordinate system and the center of a hole, or
- the angular offset between the nominal position and the actual position of a hole center.

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





1. The TNC positions the touch probe at rapid traverse (value from FMAX column) following the positioning logic (see "Running touch probe cycles" on page 21) to the starting point 1. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table
2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column F). The TNC derives the probing direction automatically from the programmed starting angle.
3. Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point 2 and probes the second touch point.
4. The TNC positions the probe to starting point 3 and then to starting point 4 to probe the third and fourth touch points and positions the touch probe on the hole centers measured.
5. Finally the TNC returns the touch probe to the clearance height and aligns the workpiece by rotating the table. The TNC rotates the rotary table so that the hole center after compensation lies in the direction of the positive Y axis, or on the nominal position of the hole center — both with a vertical and horizontal touch probe axis. The measured angular misalignment is also available in parameter Q150.

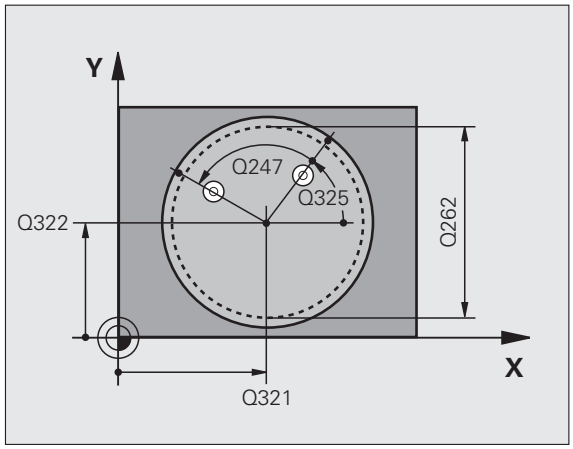
Before programming, note the following

To prevent a collision between the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole).

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

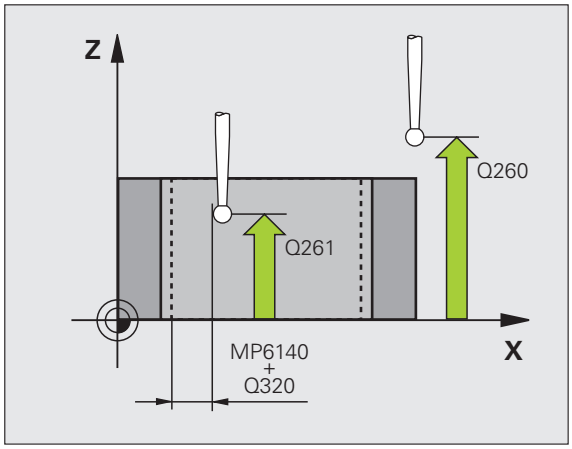
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

- **Center in 1st axis** Q321 (absolute value): Center of the hole in the reference axis of the working plane.
- **Center in 2nd axis** Q322 (absolute value): Center of the hole in the minor axis of the working plane. If you program Q322 = 0, the TNC aligns the hole center to the positive Y axis. If you program Q322 not equal to 0, then the TNC aligns the hole center to the nominal position (angle of the hole center).
- **Nominal diameter** Q262: Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large.
- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point.
- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (negative = clockwise) in which the touch probe moves to the next measuring point. If you wish to probe a circular arc instead of a complete circle, then program the stepping angle to be less than 90°.



The smaller the angle, the less accurately the TNC can calculate the circle center. Minimum input value: 5°.

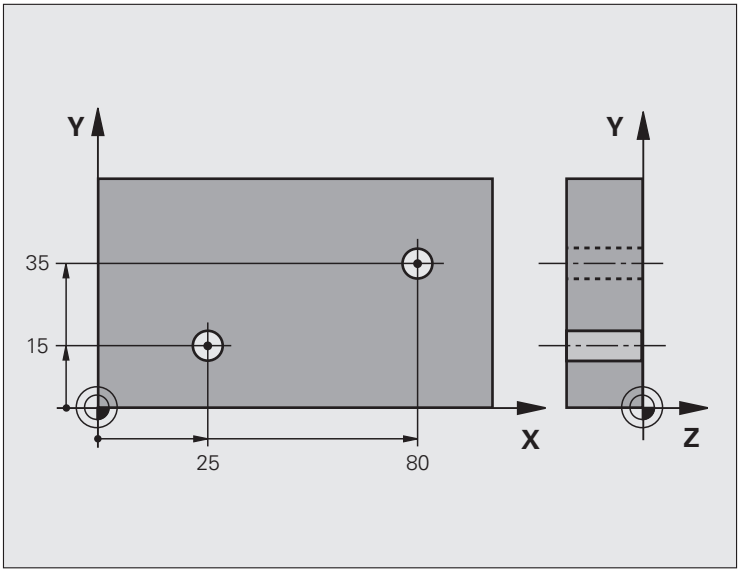
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Set to zero after alignment** Q337: Definition of whether the TNC should set the display of the C axis to zero, or write the angular offset in column C of the datum table:
  0: Set display of C to 0
  >0: Write the angular misalignment, including algebraic sign, in the datum table. Line number = value of Q337. If a C-axis shift is registered in the datum table, the TNC adds the measured angular misalignment.



Example: NC blocks

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

Example: Determining a basic rotation from two holes



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

3.2 Presetting automatically

Overview

The TNC offers twelve cycles for automatically finding reference points and handling them as follows:

- Setting the determined values directly as display values
- Entering the determined values in the preset table
- Entering the determined values in a datum table

Cycle	Soft key	Page
408 SLOT CENTER REF PT. Measuring the inside width of a slot, and defining the slot center as datum		Page 65
409 RIDGE CENTER REF PT. Measuring the outside width of a ridge, and defining the ridge center as datum		Page 68
410 DATUM INSIDE RECTAN. Measuring the inside length and width of a rectangle, and defining the center as datum		Page 71
411 DATUM OUTSIDE RECTAN. Measuring the outside length and width of a rectangle, and defining the center as datum		Page 74
412 DATUM INSIDE CIRCLE Measuring any four points on the inside of a circle, and defining the center as datum		Page 76
413 DATUM OUTSIDE CIRCLE Measuring any four points on the outside of a circle, and defining the center as datum		Page 81
414 DATUM OUTSIDE CORNER Measuring two lines from the outside of the angle, and defining the intersection as datum		Page 85
415 DATUM INSIDE CORNER Measuring two lines from within the angle, and defining the intersection as datum		Page 88
416 DATUM CIRCLE CENTER (2nd soft-key level) Measuring any three holes on a bolt hole circle, and defining the bolt-hole center as datum		Page 91

Cycle	Soft key	Page
417 DATUM IN TS AXIS (2nd soft-key level) Measuring any position in the touch probe axis and defining it as datum		Page 94
418 DATUM FROM 4 HOLES (2nd soft-key level) Measuring 4 holes crosswise and defining the intersection of the lines between them as datum		Page 96
419 DATUM IN ONE AXIS (2nd soft-key level) Measuring any position in any axis and defining it as datum		Page 99

Characteristics common to all touch probe cycles for datum setting

You can also run the touch probe cycles 408 to 419 during an active basic rotation.

The tilting the working plane function is not permitted in combination with cycles 408 to 419.

When running touch probe cycles, no cycles must be active for conversion of coordinates (Cycle 7 DATUM, Cycle 8 REFLECTION, Cycle 10 ROTATION, Cycles 11 and 26 SIZE FACTOR and Cycle 19 WORKING PLANE).

Datum point and touch probe axis

From the touch probe axis that you have defined in the measuring program the TNC determines the working plane for the datum:

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

Saving the calculated datum

In all cycles for datum setting you can use the input parameters Q303 and Q305 to define how the TNC is to save the calculated datum:

- **Q305 = 0, Q303 = any value**
  The TNC sets the calculated datum in the display. The new datum is active immediately.
- **Q305 not equal to 0, Q303 = -1**

This combination can only occur if you

- read in programs containing Cycles 410 to 418 created on a TNC 4xx
- read in programs containing Cycles 410 to 418 created with an older software version on an iTNC 530
- did not yourself define the measured-value transfer with parameter Q303 in the cycle definition

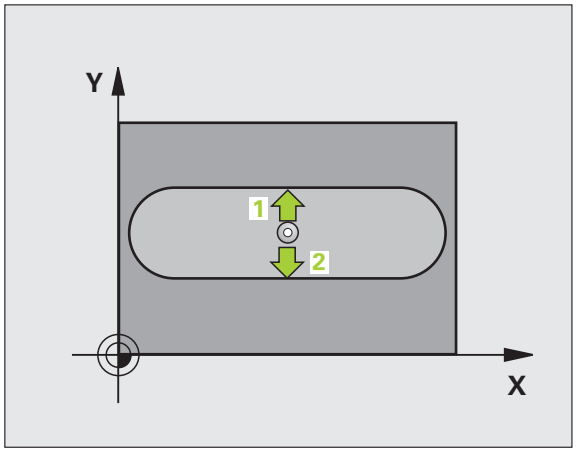
In these cases the TNC outputs an error message, since the complete handling of REF-referenced datum tables has changed. You must define a measured-value transfer yourself with parameter Q303.

- **Q305 not equal to 0, Q303 = 0**
  The TNC writes the calculated reference point in the active datum table. The reference system is the active workpiece coordinate system. The value of parameter Q305 determines the datum number. **Activate datum with Cycle 7 in the part program.**
- **Q305 not equal to 0, Q303 = 1**
  The TNC writes the calculated reference point in the preset table. The reference system is the machine coordinate system (REF coordinates). The value of parameter Q305 determines the preset number. **Activate preset with Cycle 247 in the part program.**

Measurement results in Q parameters

The TNC 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 program. Note the table of result parameters that are listed with every cycle description.

DATUM SLOT CENTER (touch probe Cycle 408, DIN/ISO: G408)



Touch probe cycle 408 finds the center of a slot and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.

1. The TNC positions the touch probe at rapid traverse (value from FMAX column) following the positioning logic (see "Running touch probe cycles" on page 21) to the starting point 1. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table
2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (column F).
3. Then the touch probe moves either paraxially at the measuring height or linearly at the clearance height to the next starting point 2 and probes the second touch point.
4. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 64) and saves the actual values in the Q parameters listed below
5. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

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

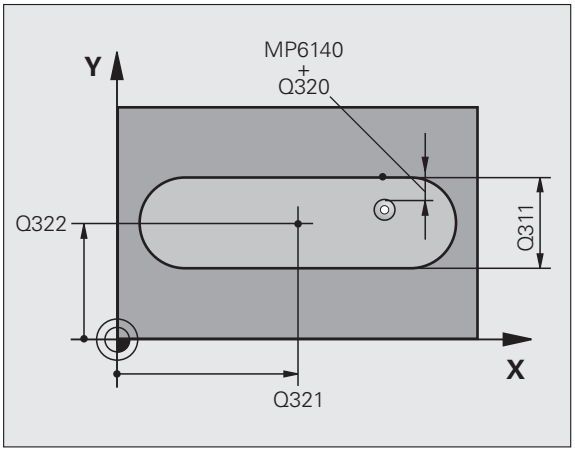
Before programming, note the following

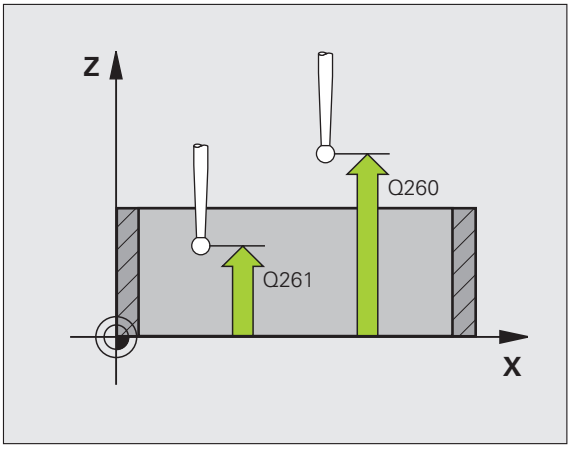
To prevent a collision between touch probe and workpiece, enter a **low** estimate for the slot width.

If the slot width and the safety clearance do not permit pre-positioning in the proximity of the touch points, the TNC always starts probing from the center of the slot. In this case the touch probe does not return to the clearance height between the two measuring points.

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

- **Center in 1st axis** Q321 (absolute value): Center of the slot in the reference axis of the working plane.
- **Center in 2nd axis** Q322 (absolute value): Center of the slot in the minor axis of the working plane.
- **Width of slot** Q311 (incremental): Width of the slot, regardless of its position in the working plane.
- **Measuring axis (1=1st axis / 2=2nd axis)** Q272: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the slot center. If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the slot center.
- **New datum** Q405 (absolute): Coordinate in the measuring axis at which the TNC should set the calculated slot center. Basic setting = 0





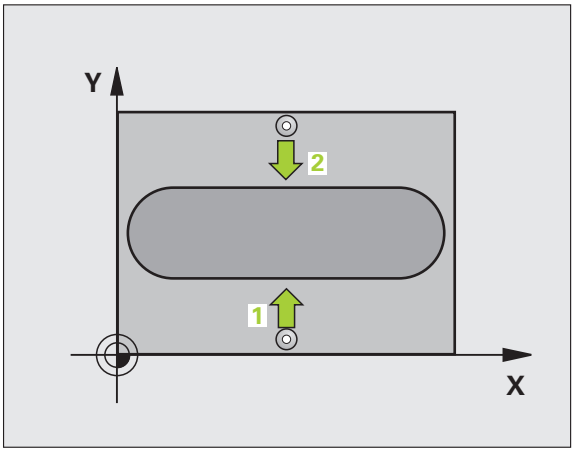
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set datum in the touch probe axis
- **Probe TS axis: Coord. 1st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the reference point is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 2nd axis** Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the reference point is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 3 rd axis** Q384 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **New datum for touch probe axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Basic setting = 0

Example: NC blocks

```
5 TCH PROBE 408 SLOT CENTER REF PT
     Q321=+50      ;CENTER 1ST AXIS
     Q322=+50      ;CENTER 2ND AXIS
     Q311=25       ;SLOT WIDTH
     Q272=1        ;MEASURING AXIS
     Q261=-5       ;MEASURING HEIGHT
     Q320=0        ;SET-UP CLEARANCE
     Q260=+20      ;CLEARANCE HEIGHT
     Q301=0        ;MOVE TO CLEARANCE
     Q305=10       ;NO. IN TABLE
     Q405=+0       ;DATUM
     Q303=+1       ;MEAS. VALUE TRANSFER
     Q381=1        ;PROBE IN TS AXIS
     Q382=+85      ;1ST CO. FOR TS AXIS
     Q383=+50      ;2ND CO. FOR TS AXIS
     Q384=+0       ;2RD CO. FOR TS AXIS
     Q333=+0       ;DATUM
```

DATUM RIDGE CENTER (touch probe cycle 409, DIN/ISO: G409)

Touch probe cycle 409 finds the center of a ridge and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.



1. The TNC positions the touch probe at rapid traverse (value from FMAX column) following the positioning logic (see "Running touch probe cycles" on page 21) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table
2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (column F).
3. Then the touch probe moves at clearance height to the next touch point **2** and probes the second touch point.
4. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 64) and saves the actual values in the Q parameters listed below
5. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

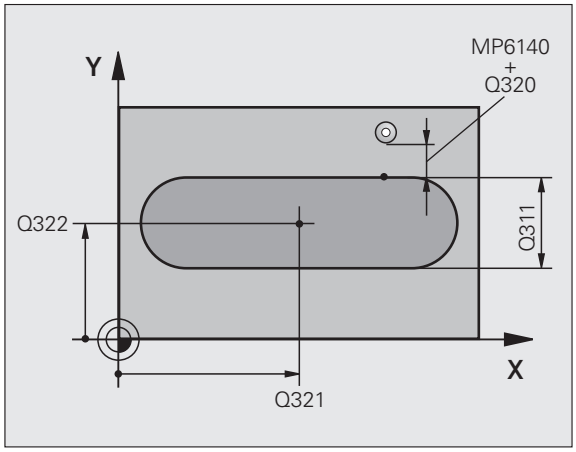
Parameter number	Meaning
Q166	Actual value of measured ridge width
Q157	Actual value of the centerline

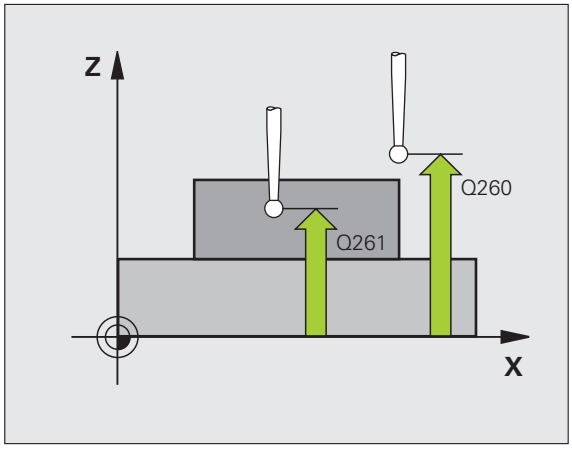
Before programming, note the following

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

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

- **Center in 1st axis** Q321 (absolute): Center of the ridge in the reference axis of the working plane.
- **Center in 2nd axis** Q322 (absolute): Center of the ridge in the minor axis of the working plane.
- **Width of ridge** Q311 (incremental): Width of the ridge, regardless of its position in the working plane.
- **Measuring axis (1=1st axis / 2=2nd axis)** Q272: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the ridge center. If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the slot center.
- **New datum** Q405 (absolute): Coordinate in the measuring axis at which the TNC should set the calculated ridge center. Basic setting = 0



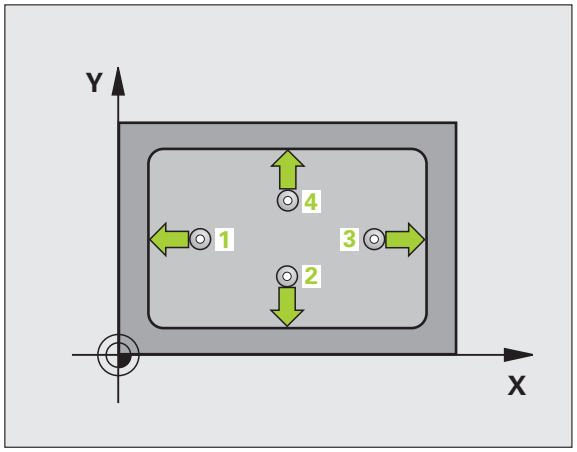


- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set datum in the touch probe axis
- **Probe TS axis: Coord. 1. st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 2. nd axis** Q383 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 3. rd axis** Q384 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **New datum for touch probe axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Basic setting = 0

Example: NC blocks

```
5 TCH PROBE 409 SLOT CENTER STUD
    Q321=+50      ;CENTER 1ST AXIS
    Q322=+50      ;CENTER 2ND AXIS
    Q311=25       ;RIDGE WIDTH
    Q272=1        ;MEASURING AXIS
    Q261=-5       ;MEASURING HEIGHT
    Q320=0        ;SET-UP CLEARANCE
    Q260=+20      ;CLEARANCE HEIGHT
    Q305=10       ;NO. IN TABLE
    Q405=+0       ;DATUM
    Q303=+1       ;MEAS. VALUE TRANSFER
    Q381=1        ;PROBE IN TS AXIS
    Q382=+85      ;1ST CO. FOR TS AXIS
    Q383=+50      ;2ND CO. FOR TS AXIS
    Q384=+0       ;2RD CO. FOR TS AXIS
    Q333=+0       ;DATUM
```

DATUM FROM INSIDE OF RECTANGLE (touch probe cycle 410, DIN/ISO: G410)



Touch probe cycle 410 finds the center of a rectangular pocket and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.

1. The TNC positions the touch probe at rapid traverse (value from FMAX column) following the positioning logic (see "Running touch probe cycles" on page 21) to the starting point **1.** The TNC calculates the probe starting points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table
2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (column F).
3. Then the touch probe moves either paraxially at the measuring height or linearly at the clearance height to the next starting point **2** and probes the second touch point.
4. The TNC positions the probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305. (see "Saving the calculated datum" on page 64)
6. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing and saves the actual values in the following Q parameters.

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

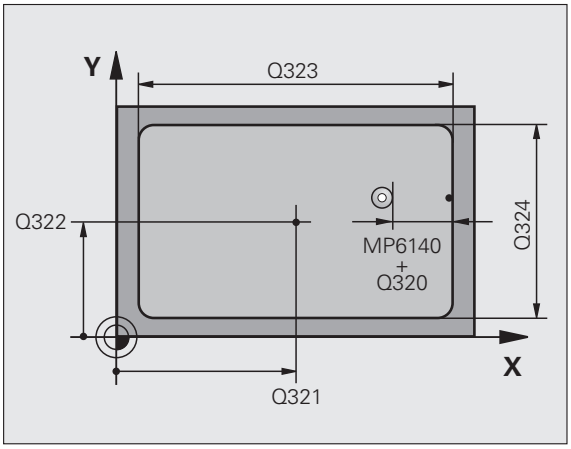
Before programming, note the following

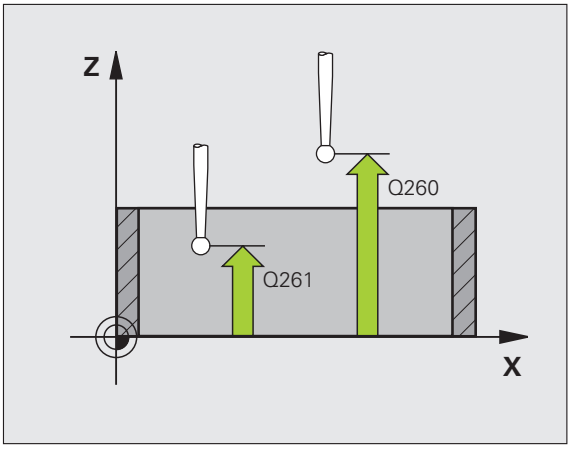
To prevent a collision between touch probe and workpiece, enter **low** estimates for the lengths of the 1st and 2nd sides.

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

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

- **Center in 1st axis** Q321 (absolute value): Center of the pocket in the reference axis of the working plane.
- **Center in 2nd axis** Q322 (absolute value): Center of the pocket in the minor axis of the working plane.
- **First side length** Q323 (incremental value): Pocket length, parallel to the reference axis of the working plane.
- **Second side length** Q324 (incremental value): Pocket length, parallel to the minor axis of the working plane
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Datum number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the pocket center. If you enter Q305=0, the TNC automatically sets the display so that the new datum is at the center of the pocket.
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the pocket center. Basic setting = 0
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the pocket center. Basic setting = 0



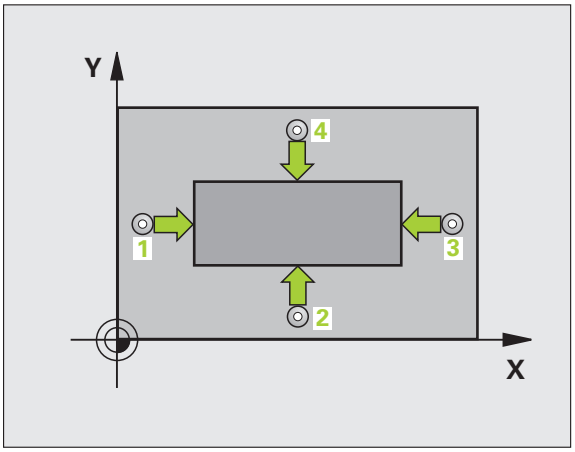


- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use. Is entered by the TNC when old programs are read in (see "Saving the calculated datum" on page 64).
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set datum in the touch probe axis
- **Probe TS axis: Coord. 1. st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 2. nd axis** Q383 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 3. rd axis** Q384 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **New datum for touch probe axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Basic setting = 0

Example: NC blocks

```
5 TCH PROBE 410 DATUM INSIDE RECTAN.
    Q321=+50      ;CENTER 1ST AXIS
    Q322=+50      ;CENTER 2ND AXIS
    Q323=60       ;FIRST SIDE LENGTH
    Q324=20       ;SECOND SIDE LENGTH
    Q261=-5       ;MEASURING HEIGHT
    Q320=0        ;SET-UP CLEARANCE
    Q260=+20      ;CLEARANCE HEIGHT
    Q301=0        ;MOVE TO CLEARANCE
    Q305=10       ;NO. IN TABLE
    Q331=+0       ;DATUM
    Q332=+0       ;DATUM
    Q303=+1       ;MEAS. VALUE TRANSFER
    Q381=1        ;PROBE IN TS AXIS
    Q382=+85      ;1ST CO. FOR TS AXIS
    Q383=+50      ;2ND CO. FOR TS AXIS
    Q384=+0       ;2RD CO. FOR TS AXIS
    Q333=+0       ;DATUM
```

DATUM FROM OUTSIDE OF RECTANGLE (touch probe cycle 411, DIN/ISO: G411)



Touch probe cycle 411 finds the center of a rectangular stud and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.

1. The TNC positions the touch probe at rapid traverse (value from FMAX column) following the positioning logic (see "Running touch probe cycles" on page 21) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table
2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (column F).
3. Then the touch probe moves either paraxially at the measuring height or linearly at the clearance height to the next starting point **2** and probes the second touch point.
4. The TNC positions the probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305. (see "Saving the calculated datum" on page 64)
6. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing and saves the actual values in the following Q parameters.

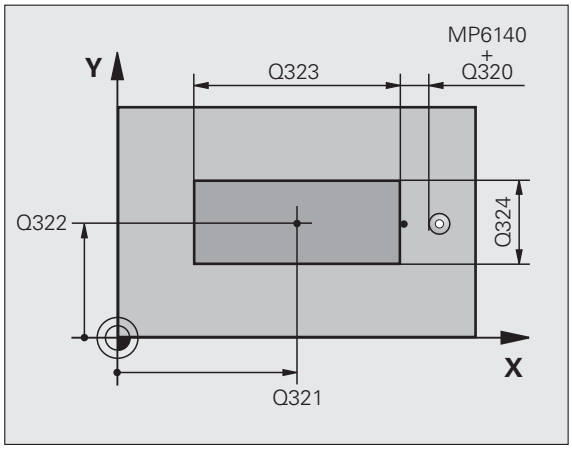
Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of length in the reference axis
Q155	Actual value of length in the minor axis

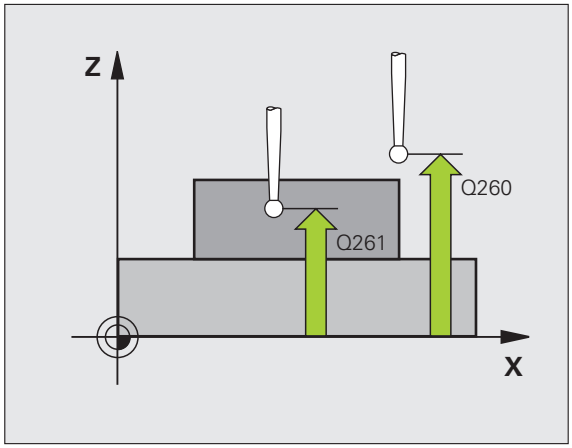
Before programming, note the following

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

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

- **Center in 1st axis** Q321 (absolute value): Center of the stud in the reference axis of the working plane.
- **Center in 2nd axis** Q322 (absolute value): Center of the stud in the minor axis of the working plane.
- **1st side length** Q323 (incremental value): Stud length, parallel to the reference axis of the working plane
- **2nd side length** Q324 (incremental value): Stud length, parallel to the secondary axis of the working plane.
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Datum number in table** Q305: Enter the datum number in the table in which the TNC is to save the coordinates of the pocket center. If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the stud center.
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the stud center. Basic setting = 0
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the stud center. Basic setting = 0





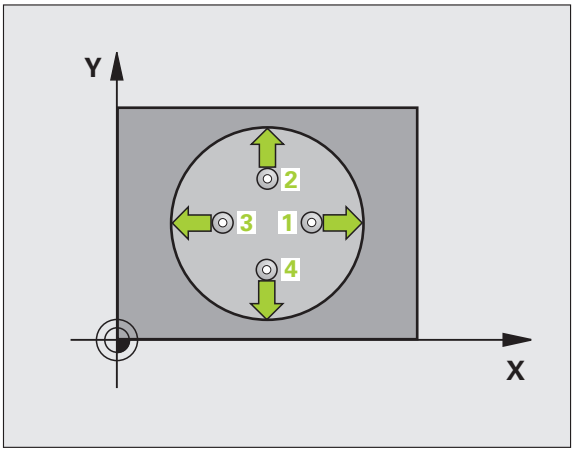
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use. Is entered by the TNC when old programs are read in (see "Saving the calculated datum" on page 64).
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set datum in the touch probe axis
- **Probe TS axis: Coord. 1. st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 2. nd axis** Q383 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 3. rd axis** Q384 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **New datum for touch probe axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Basic setting = 0

Example: NC blocks

```
5 TCH PROBE 411 DATUM OUTS. RECTAN.
    Q321=+50      ;CENTER 1ST AXIS
    Q322=+50      ;CENTER 2ND AXIS
    Q323=60       ;FIRST SIDE LENGTH
    Q324=20       ;SECOND SIDE LENGTH
    Q261=-5       ;MEASURING HEIGHT
    Q320=0        ;SET-UP CLEARANCE
    Q260=+20      ;CLEARANCE HEIGHT
    Q301=0        ;MOVE TO CLEARANCE
    Q305=0        ;NO. IN TABLE
    Q331=+0       ;DATUM
    Q332=+0       ;DATUM
    Q303=+1       ;MEAS. VALUE TRANSFER
    Q381=1        ;PROBE IN TS AXIS
    Q382=+85      ;1ST CO. FOR TS AXIS
    Q383=+50      ;2ND CO. FOR TS AXIS
    Q384=+0       ;2RD CO. FOR TS AXIS
    Q333=+0       ;DATUM
```

DATUM FROM INSIDE OF CIRCLE (touch probe cycle 412, DIN/ISO: G412)

Touch probe cycle 412 finds the center of a circular pocket (or of a hole) and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.



1. The TNC positions the touch probe at rapid traverse (value from FMAX column) following the positioning logic (see "Running touch probe cycles" on page 21) to the starting point 1. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table
2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column F). The TNC derives the probing direction automatically from the programmed starting angle.
3. Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point 2 and probes the second touch point.

4 The TNC positions the probe to starting point 3 and then to starting point 4 to probe the third and fourth touch points.

5 Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 64) and saves the actual values in the Q parameters listed below

6 If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

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

Before programming, note the following

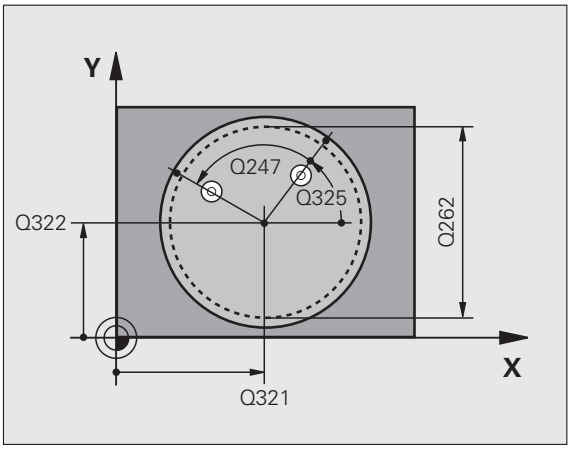
To prevent a collision between the touch probe and the workpiece, enter a **low** estimate for the nominal diameter of the pocket (or hole).

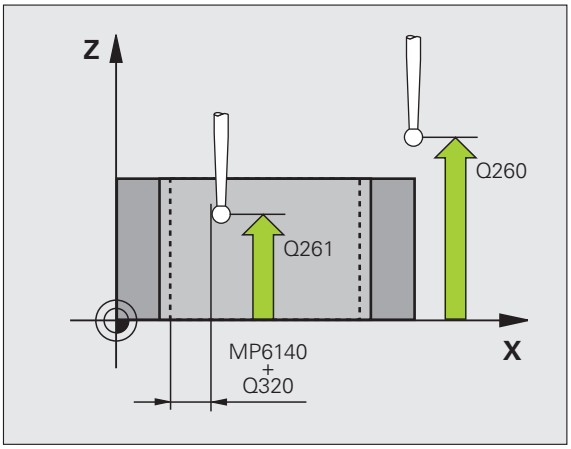
If the dimensions of the pocket and the safety clearance do not permit pre-positioning in the proximity of the touch points, the TNC always starts probing from the center of the pocket. In this case the touch probe does not return to the clearance height between the four measuring points.

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

- **Center in 1st axis** Q321 (absolute value): Center of the pocket in the reference axis of the working plane.
- **Center in 2nd axis** Q322 (absolute value): Center of the pocket in the minor axis of the working plane. If you program Q322 = 0, the TNC aligns the hole center to the positive Y axis. If you program Q322 not equal to 0, then the TNC aligns the hole center to the nominal position.
- **Nominal diameter** Q262: Approximate diameter of the circular pocket (or hole). Enter a value that is more likely to be too small than too large.
- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point.
- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (- = 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°.

The smaller the angle, the less accurately the TNC can calculate the datum. Minimum input value: 5°





- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Datum number in table** Q305: Enter the number in the datum/preset table in which the TNC is to save the coordinates of the pocket center. If you enter Q305=0, the TNC automatically sets the display so that the new datum is at the center of the pocket.

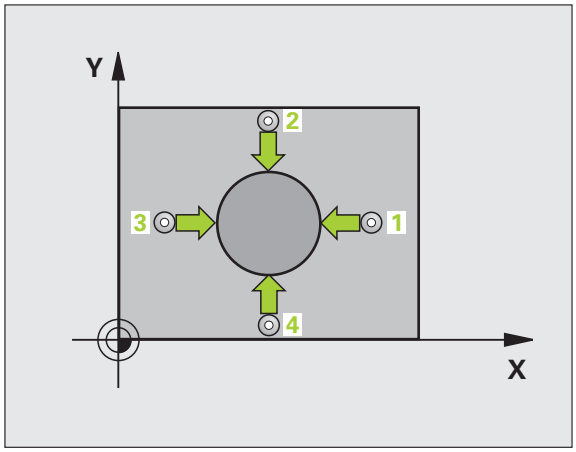
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the pocket center. Basic setting = 0
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the pocket center. Basic setting = 0
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use. Is entered by the TNC when old programs are read in (see "Saving the calculated datum" on page 64).
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set datum in the touch probe axis
- **Probe TS axis: Coord. 1. st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 2. nd axis** Q383 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 3. rd axis** Q384 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **New datum for touch probe axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Basic setting = 0
- **No. of measuring points (4/3)** Q423: Specify whether the TNC should measure the hole with 4 or 3 probing points:
  4: Use 4 measuring points (standard setting)
  3: Use 3 measuring points (standard setting)

Example: NC blocks

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

DATUM FROM OUTSIDE OF CIRCLE (touch probe cycle 413, DIN/ISO: G413)

Touch probe cycle 413 finds the center of a circular stud and defines it as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.



1. The TNC positions the touch probe at rapid traverse (value from FMAX column) following the positioning logic (see "Running touch probe cycles" on page 21) to the starting point 1. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table
2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column F). The TNC derives the probing direction automatically from the programmed starting angle.
3. Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point 2 and probes the second touch point.
4. The TNC positions the probe to starting point 3 and then to starting point 4 to probe the third and fourth touch points.
5. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 64) and saves the actual values in the Q parameters listed below
6. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

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

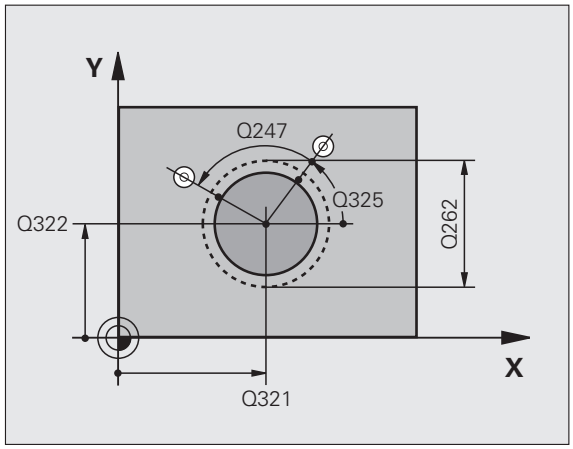
Before programming, note the following

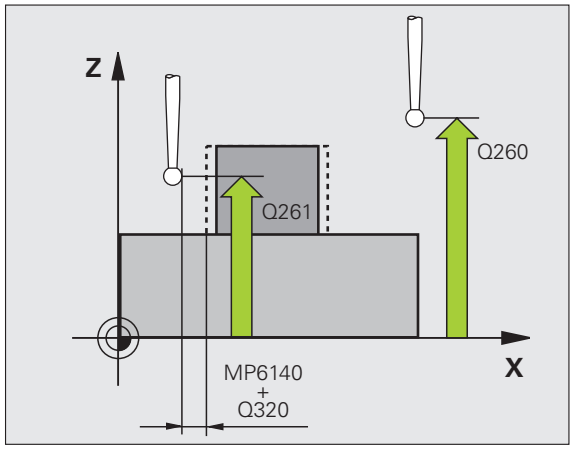
To prevent a collision between touch probe and workpiece, enter a **high** estimate for the nominal diameter of the stud.

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

- **Center in 1st axis** Q321 (absolute value): Center of the stud in the reference axis of the working plane.
- **Center in 2nd axis** Q322 (absolute value): Center of the stud in the minor axis of the working plane. If you program Q322 = 0, the TNC aligns the hole center to the positive Y axis. If you program Q322 not equal to 0, then the TNC aligns the hole center to the nominal position.
- **Nominal diameter** Q262: Approximate diameter of the stud. Enter a value that is more likely to be too large than too small.
- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point.
- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (- = 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°.

The smaller the angle, the less accurately the TNC can calculate the datum. Minimum input value: 5°.





- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Datum number in table** Q305: Enter the datum number in the table in which the TNC is to save the coordinates of the pocket center. If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the stud center.

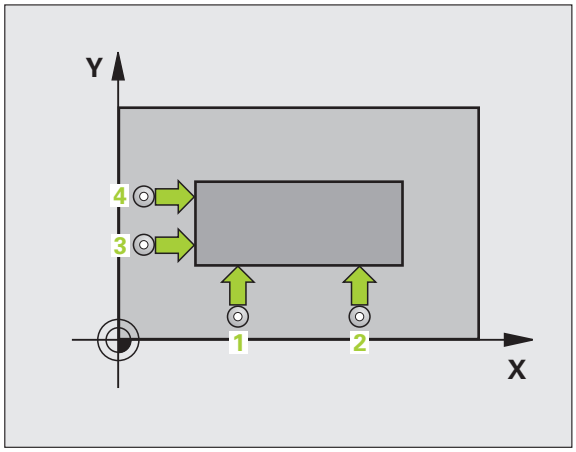
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the stud center. Basic setting = 0
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the stud center. Basic setting = 0
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use. Is entered by the TNC when old programs are read in (see "Saving the calculated datum" on page 64).
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set datum in the touch probe axis
- **Probe TS axis: Coord. 1. st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 2. nd axis** Q383 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 3. rd axis** Q384 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **New datum for touch probe axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Basic setting = 0
- **No. of measuring points (4/3)** Q423: Specify whether the TNC should measure the stud with 4 or 3 probing points:
  4: Use 4 measuring points (standard setting)
  3: Use 3 measuring points (standard setting)

Example: NC blocks

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

DATUM FROM OUTSIDE OF CORNER (touch probe cycle 414, DIN/ISO: G414)

Touch probe cycle 414 finds the intersection of two lines and defines it as the datum. If desired, the TNC can also enter the intersection into a datum table or preset table.



1 Following the positioning logic (see "Running touch probe cycles" on page 21), the TNC positions the touch probe at rapid traverse (value from FMAX column) to the first touch point 1 (see figure at upper right). The TNC offsets the touch probe by the safety clearance in the direction opposite the respective traverse direction.

2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column F). The TNC derives the probing direction automatically from the programmed 3rd measuring point.

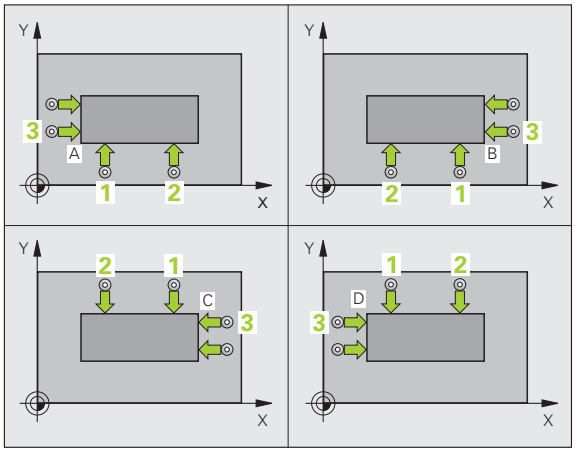
The TNC always measures the first line in the direction of the minor axis of the working plane.

3 Then the touch probe moves to the next starting position 2 and probes the second position.

4 The TNC positions the probe to starting point 3 and then to starting point 4 to probe the third and fourth touch points.

5 Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 64) and saves the coordinates of the determined corner in the Q parameters listed below

6 If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.



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

Before programming, note the following

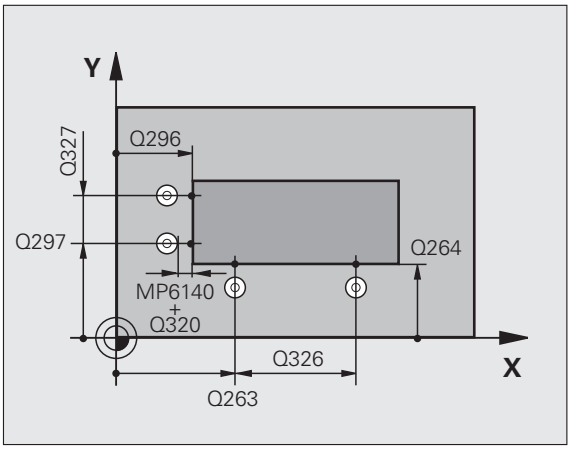
By defining the positions of the measuring points 1 and 3 you also determine the corner at which the TNC sets the datum (see figure at right and table at lower right).

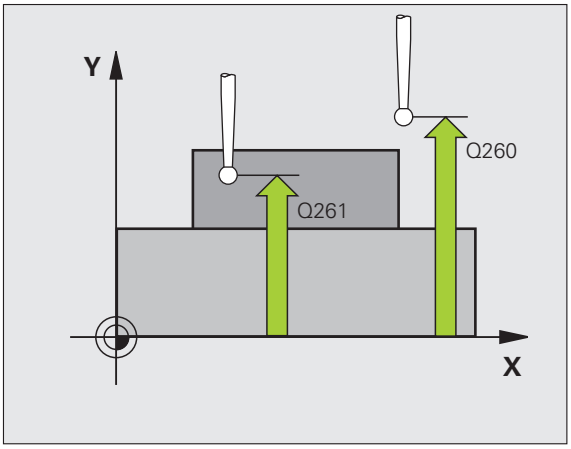
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

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

414

- **1st measuring point in 1st axis** Q263 (absolute): coordinate of the first touch point in the reference axis of the working plane.
- **1st measuring point in 2nd axis** Q264 (absolute): coordinate of the first touch point in the minor axis of the working plane.
- **Spacing in 1st axis** Q326 (incremental): Distance between the first and second measuring points in the reference axis of the working plane.
- **3rd measuring point in 1st axis** Q296 (absolute): coordinate of the third touch point in the reference axis of the working plane.
- **3rd measuring point in 2nd axis** Q297 (absolute): coordinate of the third touch point in the minor axis of the working plane.
- **Spacing in 2nd axis** Q327 (incremental): Distance between third and fourth measuring points in the minor axis of the working plane.
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points





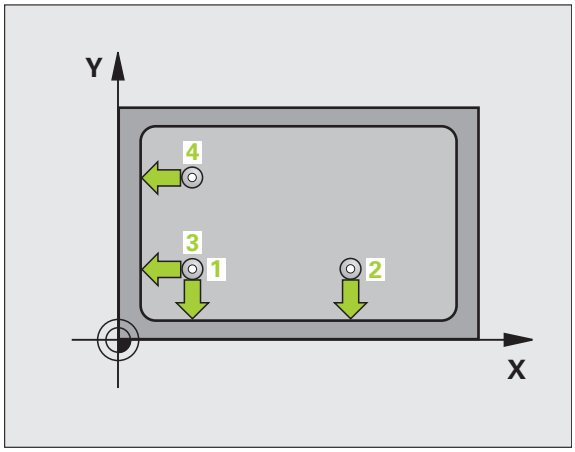
- **Execute basic rotation** Q304: Definition of whether the TNC should compensate workpiece misalignment with a basic rotation:
  0: No basic rotation
  1: Basic rotation
- **Datum number in table** Q305: Enter the datum number in the datum or preset table in which the TNC is to save the coordinates of the corner. If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the corner.
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the corner. Basic setting = 0
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the calculated corner. Basic setting = 0
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use. Is entered by the TNC when old programs are read in (see "Saving the calculated datum" on page 64).
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set datum in the touch probe axis
- **Probe TS axis: Coord. 1. st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 2. nd axis** Q383 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 3. rd axis** Q384 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **New datum for touch probe axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Basic setting = 0

Example: NC blocks

```
5 TCH PROBE 414 DATUM INSIDE CORNER
    Q263=+37      ;1ST POINT 1ST AXIS
    Q264=+7       ;1ST POINT 2ND AXIS
    Q326=50       ;SPACING IN 1ST AXIS
    Q296=+95      ;3RD POINT 1ST AXIS
    Q297=+25      ;3RD POINT 2ND AXIS
    Q327=45       ;SPACING IN 2ND AXIS
    Q261=-5       ;MEASURING HEIGHT
    Q320=0        ;SET-UP CLEARANCE
    Q260=+20      ;CLEARANCE HEIGHT
    Q301=0        ;MOVE TO CLEARANCE
    Q304=0        ;BASIC ROTATION
    Q305=7        ;NO. IN TABLE
    Q331=+0       ;DATUM
    Q332=+0       ;DATUM
    Q303=+1       ;MEAS. VALUE TRANSFER
    Q381=1        ;PROBE IN TS AXIS
    Q382=+85      ;1ST CO. FOR TS AXIS
    Q383=+50      ;2ND CO. FOR TS AXIS
    Q384=+0       ;2RD CO. FOR TS AXIS
    Q333=+0       ;DATUM
```

DATUM FROM INSIDE OF CORNER (touch probe cycle 415, DIN/ISO: G415)

Touch probe cycle 415 finds the intersection of two lines and defines it as the datum. If desired, the TNC can also enter the intersection into a datum table or preset table.



1. Following the positioning logic (see "Running touch probe cycles" on page 21), the TNC positions the touch probe at rapid traverse (value from FMAX column) to the first touch point **1** (see figure at upper right) that you have defined in the cycle. The TNC offsets the touch probe by the safety clearance in the direction opposite the respective traverse direction.
2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column F). The probing direction is derived from the number by which you identify the corner.

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

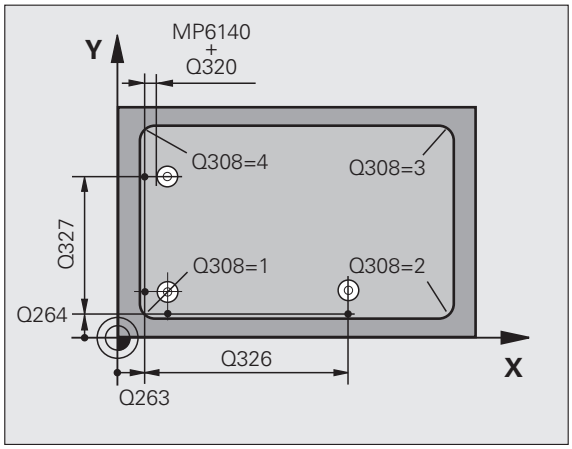
3. Then the touch probe moves to the next starting position **2** and probes the second position.
4. The TNC positions the probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 64) and saves the coordinates of the determined corner in the Q parameters listed below
6. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

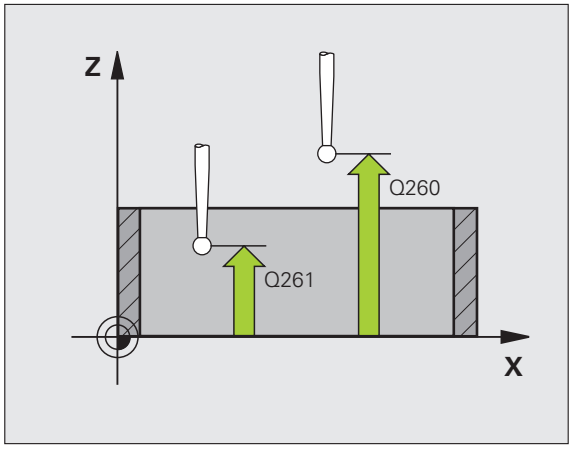
Parameter number	Meaning
Q151	Actual value of corner in reference axis
Q152	Actual value of corner in minor axis

Before programming, note the following

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

- **1st measuring point in 1st axis** Q263 (absolute): coordinate of the first touch point in the reference axis of the working plane.
- **1st measuring point in 2nd axis** Q264 (absolute): coordinate of the first touch point in the minor axis of the working plane.
- **Spacing in 1st axis** Q326 (incremental): Distance between the first and second measuring points in the reference axis of the working plane.
- **Spacing in 2nd axis** Q327 (incremental): Distance between third and fourth measuring points in the minor axis of the working plane.
- **Corner** Q308: Number identifying the corner which the TNC is to set as datum.
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Execute basic rotation** Q304: Definition of whether the TNC should compensate workpiece misalignment with a basic rotation:
  0: No basic rotation
  1: Basic rotation





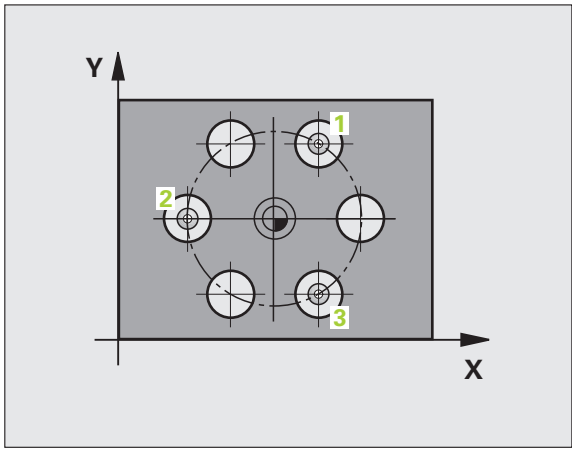
- **Datum number in table** Q305: Enter the datum number in the datum or preset table in which the TNC is to save the coordinates of the corner. If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the corner.
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the corner. Basic setting = 0
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the calculated corner. Basic setting = 0
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use. Is entered by the TNC when old programs are read in (see "Saving the calculated datum" on page 64).
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set datum in the touch probe axis
- **Probe TS axis: Coord. 1. st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 2. nd axis** Q383 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 3. rd axis** Q384 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **New datum for touch probe axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Basic setting = 0

Example: NC blocks

```
5 TCH PROBE 415 DATUM OUTSIDE CORNER
    Q263=+37      ;1ST POINT 1ST AXIS
    Q264=+7       ;1ST POINT 2ND AXIS
    Q326=50       ;SPACING IN 1ST AXIS
    Q296=+95      ;3RD POINT 1ST AXIS
    Q297=+25      ;3RD POINT 2ND AXIS
    Q327=45       ;SPACING IN 2ND AXIS
    Q261=-5       ;MEASURING HEIGHT
    Q320=0        ;SET-UP CLEARANCE
    Q260=+20      ;CLEARANCE HEIGHT
    Q301=0        ;MOVE TO CLEARANCE
    Q304=0        ;BASIC ROTATION
    Q305=7        ;NO. IN TABLE
    Q331=+0       ;DATUM
    Q332=+0       ;DATUM
    Q303=+1       ;MEAS. VALUE TRANSFER
    Q381=1        ;PROBE IN TS AXIS
    Q382=+85      ;1ST CO. FOR TS AXIS
    Q383=+50      ;2ND CO. FOR TS AXIS
    Q384=+0       ;2RD CO. FOR TS AXIS
    Q333=+0       ;DATUM
```

DATUM CIRCLE CENTER (touch probe cycle 416, DIN/ISO: G416)

Touch probe cycle 416 finds the center of a bolt hole circle and defines its center as datum. If desired, the TNC can also enter the coordinates into a datum table or the preset table.



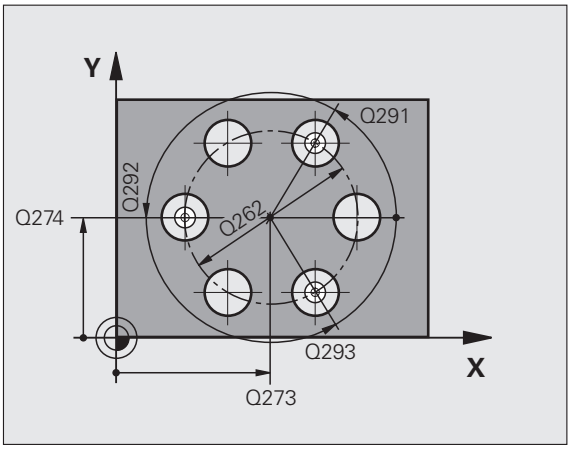
1. The TNC positions the touch probe at rapid traverse (value from column FMAX) and with positioning logic (see "Running touch probe cycles" on page 21) to the point entered as center of the first hole **1**.
2. Then the probe moves to the entered measuring height and probes four points to find the first hole center.
3. The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4. The TNC moves the touch probe to the entered measuring height and probes four points to find the second hole center.
5. The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
6. The TNC moves the touch probe to the entered measuring height and probes four points to find the third hole center.
7. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 64) and saves the actual values in the Q parameters listed below
8. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

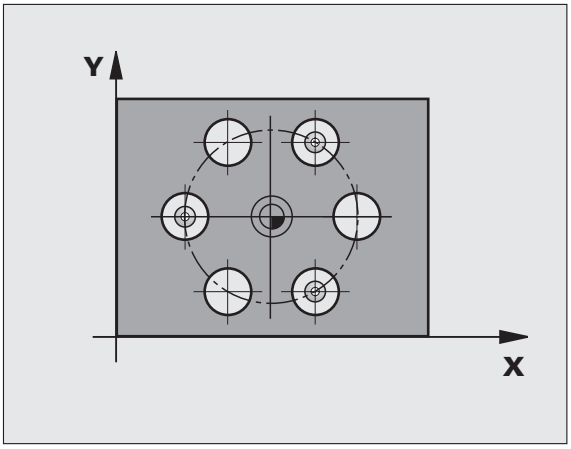
Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of bolt hole circle diameter

Before programming, note the following

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

- **Center of 1st axis** Q273 (absolute): Bolt hole circle center (nominal value) in the reference axis of the working plane.
- **Center in 2nd axis** Q274 (absolute): Bolt hole circle center (nominal value) in the minor axis of the working plane.
- **Nominal diameter** Q262: Enter the approximate bolt hole circle diameter. The smaller the hole diameter, the more exact the nominal diameter must be.
- **Angle of 1st hole** Q291 (absolute): Polar coordinate angle of the first hole center in the working plane.
- **Angle of 2nd hole** Q292 (absolute): Polar coordinate angle of the second hole center in the working plane.
- **Angle of 3rd hole** Q293 (absolute): Polar coordinate angle of the third hole center in the working plane.
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Datum number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinates of the bolt-hole circle center. If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the bolt hole center.
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the bolt-hole center.
  Basic setting = 0
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the bolt-hole center.
  Basic setting = 0





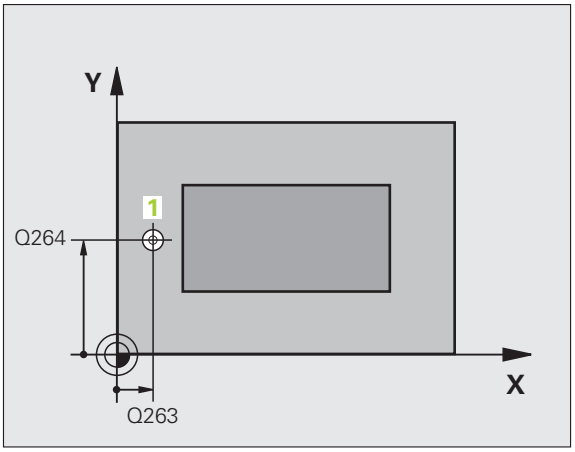
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use. Is entered by the TNC when old programs are read in (see "Saving the calculated datum" on page 64).
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set datum in the touch probe axis
- **Probe TS axis: Coord. 1. st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 2. nd axis** Q383 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 3. rd axis** Q384 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **New datum for touch probe axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Basic setting = 0

Example: NC blocks

```
5 TCH PROBE 416 DATUM CIRCLE CENTER
    Q273=+50     ;CENTER 1ST AXIS
    Q274=+50     ;CENTER 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      ;NO. IN TABLE
    Q331=+0      ;DATUM
    Q332=+0      ;DATUM
    Q303=+1      ;MEAS. VALUE TRANSFER
    Q381=1       ;PROBE IN TS AXIS
    Q382=+85     ;1ST CO. FOR TS AXIS
    Q383=+50     ;2ND CO. FOR TS AXIS
    Q384=+0      ;3RD CO. FOR TS AXIS
    Q333=+0      ;DATUM
```

DATUM IN TOUCH PROBE AXIS (touch probe cycle 417, DIN/ISO: G417)

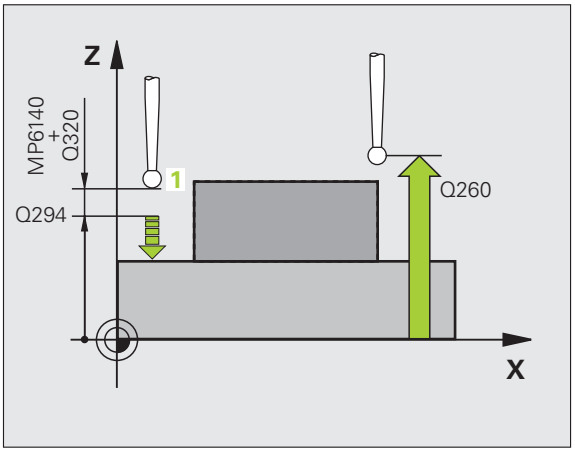
Touch probe cycle 417 measures any coordinate in the touch probe axis and defines it as datum. If desired, the TNC can also enter the measured coordinate in a datum table or preset table.



1 The TNC positions the touch probe to the starting points at rapid traverse (value from FMAX column) following the positioning logic (see "Running touch probe cycles" on page 21) to the programmed starting point 1. The TNC offsets the touch probe by the safety clearance in the positive direction of the touch probe axis.

2 Then the touch probe moves in its own axis to the coordinate entered as starting point 1 and measures the actual position with a simple probing movement.

3 Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 64) and saves the actual values in the Q parameters listed below



Parameter number	Meaning
Q160	Actual value of measured point

Before programming, note the following

Before a cycle definition you must have programmed a tool call to define the touch probe axis. The TNC then sets the datum in this axis.

- **1st measuring point in 1st axis** Q263 (absolute): coordinate of the first touch point in the reference axis of the working plane.
- **1st measuring point in 2nd axis** Q264 (absolute): coordinate of the first touch point in the minor axis of the working plane.
- **1st measuring point in 3rd axis** Q294 (absolute): coordinate of the first touch point in the touch probe axis.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.

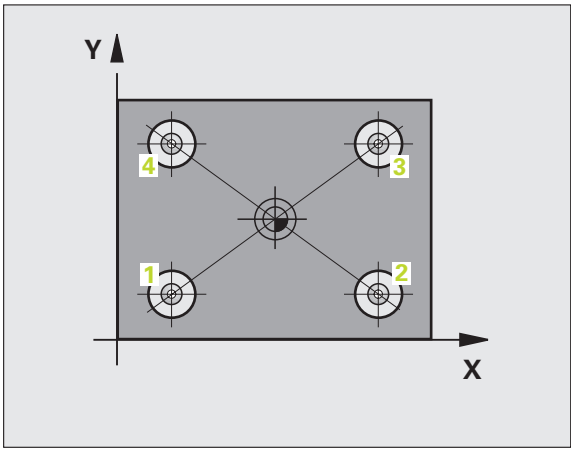
- **Datum number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinate. If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the probed surface.
- **New datum for touch probe axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Basic setting = 0
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use. Is entered by the TNC when old programs are read in (see "Saving the calculated datum" on page 64).
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).

Example: NC blocks

```
5 TCH PROBE 417 DATUM 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        ;NO. IN TABLE
    Q333=+0       ;DATUM
    Q303=+1       ;MEAS. VALUE TRANSFER
```

Datum at center of 4 holes (touch probe cycle 418, DIN/ISO: G418)

Touch probe cycle 418 calculates the intersection of the lines connecting opposite corners of a rectangle defined by four hole centers. If desired, the TNC can also enter the intersection into a datum table or preset table.



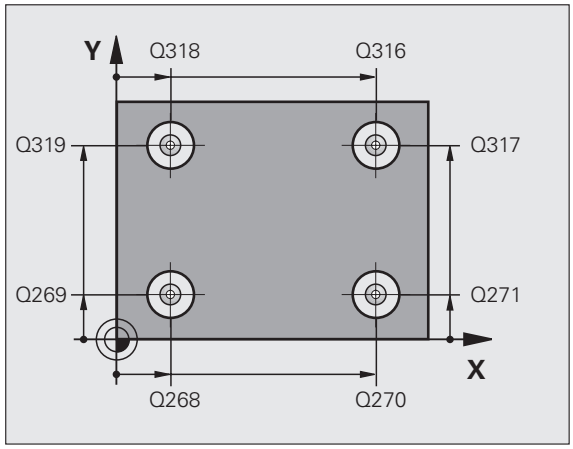
1. The TNC positions the touch probe at rapid traverse (value from column FMAX) and with positioning logic (see "Running touch probe cycles" on page 21) to the center of the first hole **1**.
2. Then the probe moves to the entered measuring height and probes four points to find the first hole center.
3. The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4. The TNC moves the touch probe to the entered measuring height and probes four points to find the second hole center.
5. The TNC repeats steps 3 and 4 for the holes **3** and **4**.
6. Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 (see "Saving the calculated datum" on page 64). The TNC calculates the datum as the intersection of the lines connecting the centers of holes **1**/**3** and **2**/**4** and saves the actual values in the Q parameters listed below.
7. If desired, the TNC subsequently measures the datum in the touch probe axis in a separate probing.

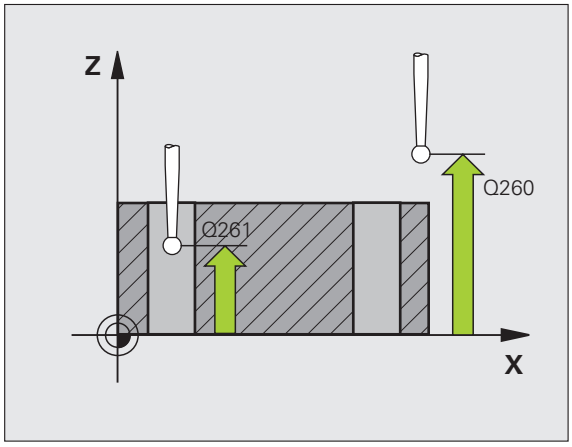
Parameter number	Meaning
Q151	Actual value of intersection point in reference axis
Q152	Actual value of intersection point in minor axis

Before programming, note the following

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

- **First center in 1st axis** Q268 (absolute): Center of the 1st hole in the reference axis of the working plane.
- **First center in 2nd axis** Q269 (absolute): Center of the 1st hole in the minor axis of the working plane.
- **Second center in 1st axis** Q270 (absolute): Center of the 2nd hole in the reference axis of the working plane.
- **Second center in 2nd axis** Q271 (absolute): Center of the 2nd hole in the minor axis of the working plane.
- **Third center in 1st axis** Q316 (absolute): Center of the 3rd hole in the reference axis of the working plane.
- **Third center in 2nd axis** Q317 (absolute): Center of the 3rd hole in the minor axis of the working plane.
- **Fourth center in 1st axis** Q318 (absolute): Center of the 4th hole in the reference axis of the working plane.
- **Fourth center in 2nd axis** Q319 (absolute): Center of the 4th hole in the minor axis of the working plane.
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.





- **Datum number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinates of the line intersection. If you enter Q305=0, the TNC automatically sets the display so that the new datum is at the intersection of the connecting lines.
- **New datum for reference axis** Q331 (absolute): Coordinate in the reference axis at which the TNC should set the calculated intersection of the connecting lines. Basic setting = 0
- **New datum for minor axis** Q332 (absolute): Coordinate in the minor axis at which the TNC should set the calculated intersection of the connecting lines. Basic setting = 0
- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use. Is entered by the TNC when old programs are read in (see "Saving the calculated datum" on page 64).
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).
- **Probe in TS axis** Q381: Specify whether the TNC should also set the datum in the touch probe axis:
  0: Do not set datum in the touch probe axis
  1: Set datum in the touch probe axis
- **Probe TS axis: Coord. 1. st axis** Q382 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 2. nd axis** Q383 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **Probe TS axis: Coord. 3. rd axis** Q384 (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.
- **New datum for touch probe axis** Q333 (absolute): Coordinate in the touch probe axis at which the TNC should set the datum. Basic setting = 0

Example: NC blocks

```
5 TCH PROBE 418 DATUM FROM 4 HOLES
     Q268=+20      ;1ST CENTER IN 1ST AXIS
     Q269=+25      ;1ST CENTER IN 2ND AXIS
     Q270=+150     ;2ND CENTER IN 1ST AXIS
     Q271=+25      ;2ND CENTER IN 2ND AXIS
     Q316=+150     ;3RD CENTER IN 1ST AXIS
     Q317=+85      ;3RD CENTER IN 2ND AXIS
     Q318=+22      ;4TH CENTER IN 1ST AXIS
     Q319=+80      ;4TH CENTER IN 2ND AXIS
     Q261=-5       ;MEASURING HEIGHT
     Q260=+10      ;CLEARANCE HEIGHT
     Q305=12       ;NO. IN TABLE
     Q331=+0       ;DATUM
     Q332=+0       ;DATUM
     Q303=+1       ;MEAS. VALUE TRANSFER
     Q381=1        ;PROBE IN TS AXIS
     Q382=+85      ;1ST CO. FOR TS AXIS
     Q383=+50      ;2ND CO. FOR TS AXIS
     Q384=+0       ;3RD CO. FOR TS AXIS
     Q333=+0       ;DATUM
```

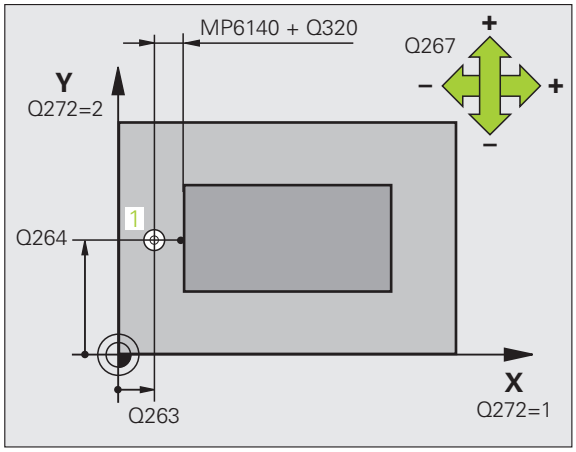
DATUM IN ONE AXIS (touch probe cycle 419, DIN/ISO: G419)

Touch probe cycle 419 measures any coordinate in any axis and defines it as datum. If desired, the TNC can also enter the measured coordinate in a datum table or preset table.

1 The TNC positions the touch probe to the starting points at rapid traverse (value from FMAX column) following the positioning logic (see "Running touch probe cycles" on page 21) to the programmed starting point 1. The TNC offsets the touch probe by the safety clearance in the direction opposite the programmed probing direction.

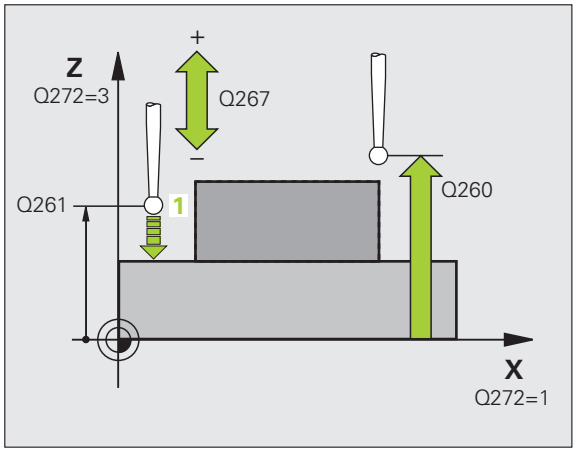
2 Then the touch probe moves to the programmed measuring height and measures the actual position with a simple probing movement.

3 Finally the TNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305. (see "Saving the calculated datum" on page 64)



Before programming, note the following

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



- **1st measuring point in 1st axis** Q263 (absolute): coordinate of the first touch point in the reference axis of the working plane.
- **1st measuring point in 2nd axis** Q264 (absolute): coordinate of the first touch point in the minor axis of the working plane.
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.

- **Measuring axis (1...3: 1=reference axis)** Q272: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
  3: Touch probe axis = measuring axis

Axis assignment

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

- **Traverse direction** Q267: Direction in which the probe is to approach the workpiece:
  -1: Negative traverse direction
  +1: Positive traverse direction

- **Datum number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinate. If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the probed surface.

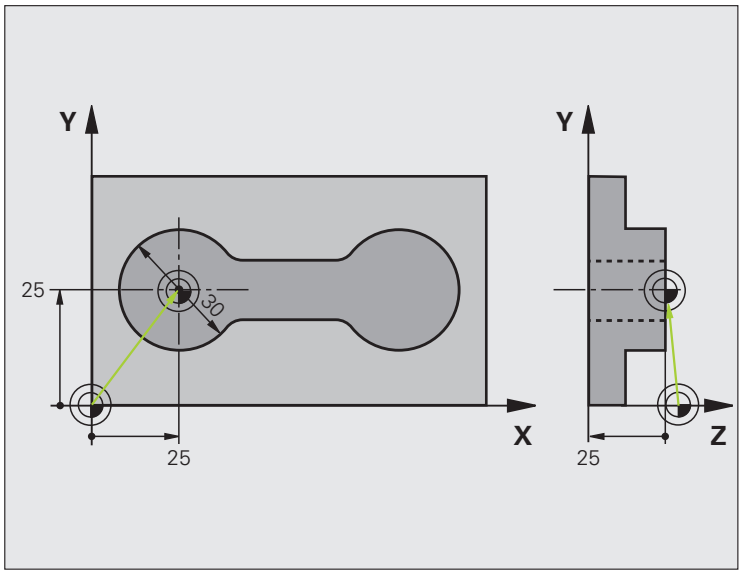
- **New datum** Q333 (absolute): Coordinate at which the TNC should set the datum. Basic setting = 0

- **Measured-value transfer (0, 1)** Q303: Specify whether the determined datum is to be saved in the datum table or in the preset table:
  -1: Do not use. See "Saving the calculated datum," page 64
  0: Write determined datum in the active datum table. The reference system is the active workpiece coordinate system.
  1: Write determined datum in the preset table. The reference system is the machine coordinate system (REF system).

Example: NC blocks

```
5 TCH PROBE 419 DATUM IN ONE AXIS
    Q263=+25      ;1ST POINT 1ST AXIS
    Q264=+25      ;1ST POINT 2ND AXIS
    Q261=+25      ;MEASURING HEIGHT
    Q320=0        ;SET-UP CLEARANCE
    Q260=+50      ;CLEARANCE HEIGHT
    Q272=+1       ;MEASURING AXIS
    Q267=+1       ;TRAVERSE DIRECTION
    Q305=0        ;NO. IN TABLE
    Q333=+0       ;DATUM
    Q303=+1       ;MEAS. VALUE TRANSFER
```

Example: Datum setting in center of a circular segment and on top surface of workpiece

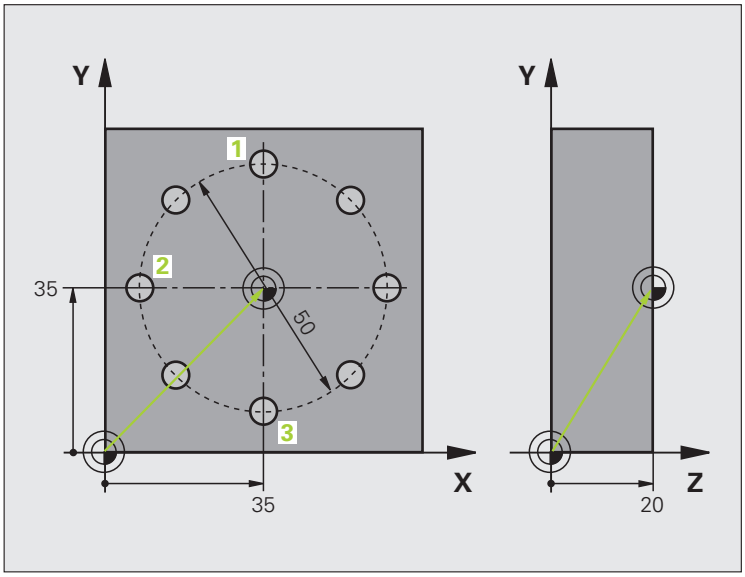


`0 BEGIN PGM CYC413 MM`	
`1 TOOL CALL 69 Z`	Call tool 0 to define the touch probe axis

2 TCH PROBE 413 DATUM OUTSIDE CIRCLE	
Q321=+25 ;CENTER 1ST AXIS	Center of circle: X coordinate
Q322=+25 ;CENTER 2ND AXIS	Center of circle: Y coordinate
Q262=30 ;NOMINAL DIAMETER	Diameter of circle
Q325=+90 ;STARTING ANGLE	Polar coordinate angle for 1st touch point
Q247=+45 ;STEPPING ANGLE	Stepping angle for calculating the starting points 2 to 4
Q261=-5 ;MEASURING HEIGHT	Coordinate in the touch probe axis in which the measurement is made
Q320=2 ;SET-UP CLEARANCE	Safety clearance in addition to SET_UP column
Q260=+10 ;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q301=0 ;MOVE TO CLEARANCE	Do not move to clearance height between measuring points
Q305=0 ;NO. IN TABLE	Set display
Q331=+0 ;DATUM	Set the display in X to 0
Q332=+10 ;DATUM	Set the display in Y to 10
Q303=+0 ;MEAS. VALUE TRANSFER	Without function, since display is to be set
Q381=1 ;PROBE IN TS AXIS	Also set datum in the touch probe axis
Q382=+25 ;1ST CO. FOR TS AXIS	X coordinate of touch point
Q383=+25 ;2ND CO. FOR TS AXIS	Y coordinate of touch point
Q384=+25 ;3RD CO. FOR TS AXIS	Z coordinate of touch point
Q333=+0 ;DATUM	Set the display in Z to 0
3 CALL PGM 1860	Part program call
4 END PGM CYC413 MM	

Example: Datum setting on top surface of workpiece and in center of a bolt hole circle

The measured bolt hole center shall be written in the preset table so that it may be used at a later time.



`0 BEGIN PGM CYC416 MM`	
`1 TOOL CALL 69 Z`	Call tool 0 to define the touch probe axis
`2 TCH PROBE 417 DATUM IN TS AXIS`	Cycle definition for datum setting in the touch probe axis
`Q263=+7.5 ;1ST POINT 1ST AXIS`	Touch point: X coordinate
`Q264=+7.5 ;1ST POINT 2ND AXIS`	Touch point: Y coordinate
`Q294=+25 ;1ST POINT 3RD AXIS`	Touch point: Z coordinate
`Q320=0 ;SET-UP CLEARANCE`	Safety clearance in addition to SET_UP column
`Q260=+50 ;CLEARANCE HEIGHT`	Height in the touch probe axis at which the probe can traverse without collision
`Q305=1 ;NO. IN TABLE`	Write Z coordinate in line 1
`Q333=+0 ;DATUM`	Set touch-probe axis to 0
`Q303=+1 ;MEAS. VALUE TRANSFER`	In the preset table PRESET.PR, save the calculated datum referenced to the machine-based coordinate system (REF system)

3 TCH PROBE 416 DATUM CIRCLE CENTER	
Q273=+35 ;CENTER 1ST AXIS	Center of the bolt hole circle: X coordinate
Q274=+35 ;CENTER 2ND AXIS	Center of the bolt hole circle: Y coordinate
Q262=50 ;NOMINAL DIAMETER	Diameter of the bolt hole circle
Q291=+90 ;ANGLE OF 1ST HOLE	Polar coordinate angle for 1st hole center **1**
Q292=+180 ;ANGLE OF 2ND HOLE	Polar coordinate angle for 2nd hole center **2**
Q293=+270 ;ANGLE OF 3RD HOLE	Polar coordinate angle for 3rd hole center **3**
Q261=+15 ;MEASURING HEIGHT	Coordinate in the touch probe axis in which the measurement is made
Q260=+10 ;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q305=1 ;NO. IN TABLE	Enter center of bolt hole circle (X and Y) in line 1
Q331=+0 ;DATUM	
Q332=+0 ;DATUM	
Q303=+1 ;MEAS. VALUE TRANSFER	In the preset table PRESET.PR, save the calculated datum referenced to the machine-based coordinate system (REF system)
Q381=0 ;PROBE IN TS AXIS	Do not set a datum in the touch probe axis
Q382=+0 ;1ST CO. FOR TS AXIS	No function
Q383=+0 ;2ND CO. FOR TS AXIS	No function
Q384=+0 ;2RD CO. FOR TS AXIS	No function
Q333=+0 ;DATUM	No function
4 CYCL DEF 247 DATUM SETTING	Activate new preset with Cycle 247
Q339=1 ;DATUM NUMBER	
6 CALL PGM 1860	Part program call
7 END PGM CYC416 MM	

3.3 Automatic Workpiece Measurement

Overview

The TNC offers twelve cycles for measuring workpieces automatically.

Cycle	Soft key	Page
0 REFERENCE PLANE Measuring a coordinate in a selectable axis		Page 110
1 POLAR DATUM PLANE Measuring a point in a probing direction		Page 111
420 MEASURE ANGLE Measuring an angle in the working plane		Page 112
421 MEASURE HOLE Measuring the position and diameter of a hole		Page 114
422 MEAS. CIRCLE OUTSIDE Measuring the position and diameter of a circular stud		Page 117
423 MEAS. RECTAN. INSIDE Measuring the position, length and width of a rectangular pocket		Page 120
424 MEAS. RECTAN. OUTSIDE Measuring the position, length and width of a rectangular stud		Page 123
425 MEASURE INSIDE WIDTH (2nd soft-key level) Measuring slot width		Page 126
426 MEASURE RIDGE WIDTH (2nd soft-key level) Measuring the width of a ridge		Page 128
427 MEASURE COORDINATE (2nd soft-key level) Measuring any coordinate in a selectable axis		Page 130
430 MEAS. BOLT HOLE CIRC. (2nd soft-key level) Measuring position and diameter of a bolt hole circle		Page 133
431 MEASURE PLANE (2nd soft-key level) Measuring the A and B axis angles of a plane		Page 136

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 TNC record the measurement results. In the respective probing cycle you can define if the TNC is to

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

If you want to save the measuring log to a file, the TNC, by default, saves the data as an ASCII file in the directory TNC:\..

All measured values listed in the log file are referenced to the datum active during the respective cycle you are running.

Use the HEIDENHAIN data transfer software TNCremo if you wish to output the measuring log via the data interface.

Example: Measuring log for touch probe cycle 421:

Measuring log for Probing Cycle 421 Hole Measuring

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

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

Given limit values: Maximum dimension 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

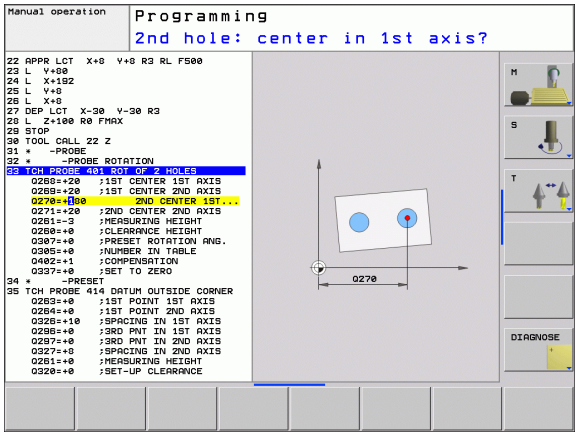
Further measuring results: Measuring height: -5.0000

End of measuring log

Measurement results in Q parameters

The TNC saves the measurement results of the respective touch probe cycle in the globally effective Q parameters Q150 to Q160. Deviations from the nominal value are saved in the parameters Q161 to Q166. Note the table of result parameters that are listed with every cycle description.

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



Classification of results

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

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

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

In Cycle 427 the TNC assumes 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 TNC also sets the status markers if you have not defined any tolerance values or maximum/minimum dimensions.

Tolerance monitoring

For most of the cycles for workpiece inspection you can have the TNC 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 0 (the default value) in the monitoring parameters.

Tool monitoring

For some cycles for workpiece inspection you can have the TNC perform tool monitoring. The TNC then monitors whether

- The tool radius should be compensated because of 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

This function works only:

- If the tool table is active.
- If tool monitoring is switched on in the cycle: **Q330** not equal to 0.

If you perform several compensation measurements, the TNC adds the respective measured deviation to the value stored in the tool table.

The TNC always compensates the tool radius in the DR column of the tool table, even if the measured deviation lies within the given tolerance. You can inquire whether re-working is necessary via Parameter Q181 in the NC program (Q181=1: must be reworked).

For Cycle 427:

- If an axis of the active working plane is defined as measuring axis (Q272 = 1 or 2), the TNC compensates the tool radius as described above. From the defined traversing direction (Q267) the TNC determines the direction of compensation.
- If the touch probe axis is defined as measuring axis (Q272 = 3), the TNC compensates the tool length.

Tool breakage monitoring

This function works only:

- If the tool table is active.
- If tool monitoring is switched on in the cycle (enter Q330 not equal to 0).
- If the breakage tolerance RBREAK for the tool number entered in the table is greater than 0 (see also the User's Manual, section 5.2 “Tool Data”).

The TNC 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

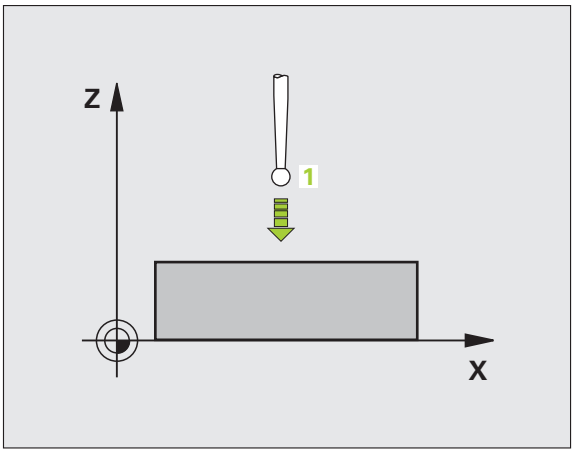
The TNC transfers all the measurement results to the result parameters and the protocol file in the active coordinate system, or as the case may be, the displaced coordinate system.

REFERENCE PLANE (touch probe cycle 0, DIN/ISO: G55)

1 The touch probe moves at rapid traverse (value from FMAX column) to the starting position 1 programmed in the cycle.

2 Then the touch probe runs the probing process at the probing feed rate (column F). The probing direction is to be defined in the cycle.

3 After the TNC has saved the position, the probe retracts to the starting point and saves the measured coordinate in a Q parameter. The TNC also stores the coordinates of the touch probe position at the time of the triggering signal in the parameters Q115 to Q119. For the values in these parameters the TNC does not account for the stylus length and radius.



Before programming, note the following

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

- **Parameter number for result:** Enter the number of the Q parameter to which you want to assign the coordinate.
- **Probing axis/Probing direction:** Enter the probing axis with the axis selection keys or ASCII keyboard and the algebraic sign for the probing direction. Confirm your entry with the ENT key.
- **Position value:** Use the axis selection keys or the ASCII keyboard to enter all coordinates of the nominal pre-positioning point values for the touch probe.
- To conclude the input, press the ENT key.

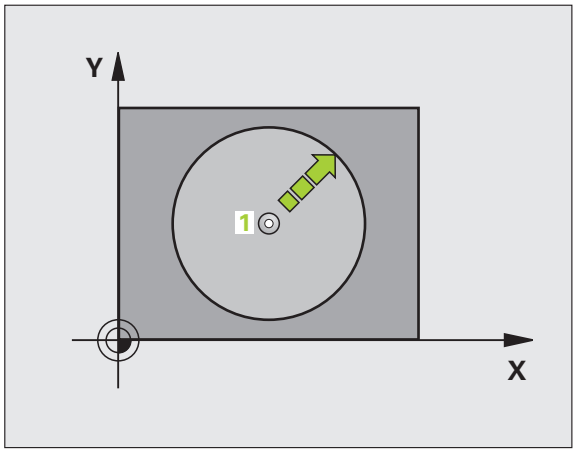
Example: NC blocks

```
67 TCH PROBE 0.0 REF. PLANE Q5 X-
68 TCH PROBE 0.1 X+5 Y+0 Z-5
```

DATUM PLANE (touch probe cycle 1)

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

1. The touch probe moves at rapid traverse (value from FMAX column) to the starting position 1 programmed in the cycle.
2. Then the touch probe runs the probing process at the probing feed rate (column F). During probing the TNC moves simultaneously in 2 axes (depending on the probing angle). The scanning direction is defined by the polar angle entered in the cycle.
3. After the TNC has saved the position, the probe returns to the starting point. The TNC also stores the coordinates of the touch probe position at the time of the triggering signal in parameters Q115 to Q119.



Before programming, note the following

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

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

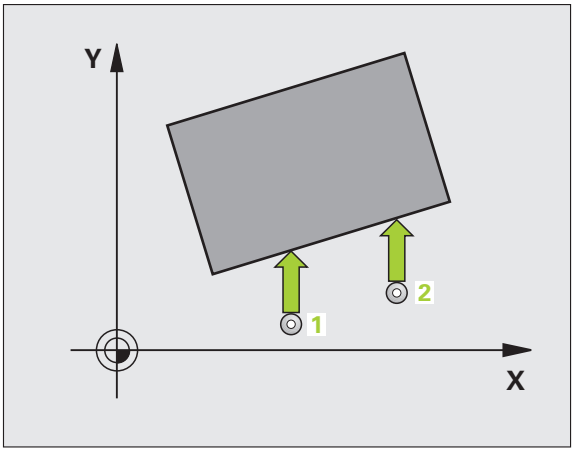
- **Probing axis:** Enter the probing axis with the axis selection keys or ASCII keyboard. Confirm your entry with the ENT key.
- **Probing angle:** Angle, measured from the probing axis, at which the touch probe is to move.
- **Position value:** Use the axis selection keys or the ASCII keyboard to enter all coordinates of the nominal pre-positioning point values for the touch probe.
- To conclude the input, press the ENT key.

Example: NC blocks

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

MEASURE ANGLE (touch probe cycle 420, DIN/ISO: G420)

Touch probe cycle 420 measures the angle that any straight surface on the workpiece describes with respect to the reference axis of the working plane.



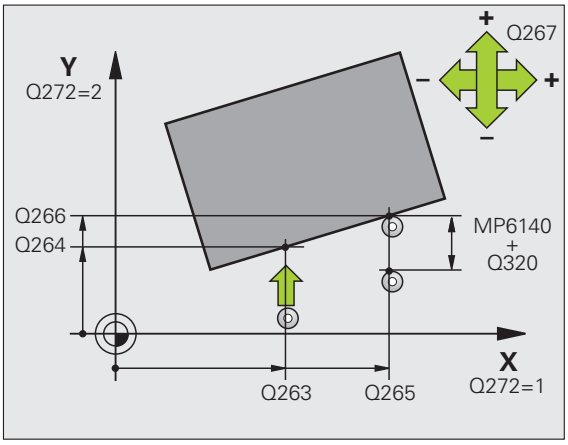
1 The TNC positions the touch probe to the starting points at rapid traverse (value from FMAX column) following the positioning logic (see "Running touch probe cycles" on page 21) to the programmed starting point 1. The TNC offsets the touch probe by the safety clearance in the direction opposite the defined traverse direction.

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

3 Then the touch probe moves to the next starting position 2 and probes the second position.

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

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



Before programming, note the following

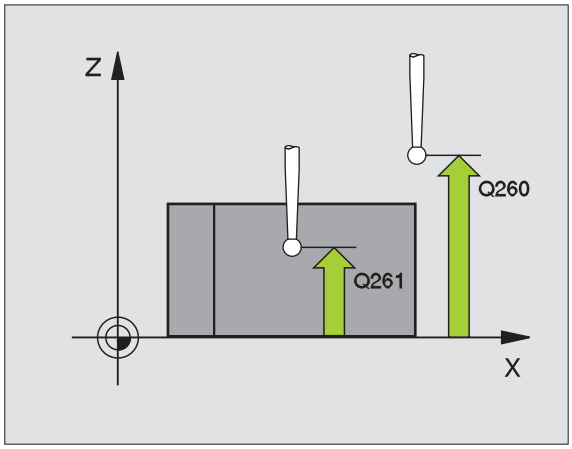
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

- **1st measuring point in 1st axis** Q263 (absolute): coordinate of the first touch point in the reference axis of the working plane.
- **1st measuring point in 2nd axis** Q264 (absolute): coordinate of the first touch point in the minor axis of the working plane.
- **2nd measuring point in 1st axis** Q265 (absolute): coordinate of the second touch point in the reference axis of the working plane
- **2nd measuring point in 2nd axis** Q266 (absolute): coordinate of the second touch point in the minor axis of the working plane
- **Measuring axis** Q272: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2: Minor axis = measuring axis
  3: Touch probe axis = measuring axis

If touch probe axis = measuring axis, then:

Set Q263 equal to Q265 if the angle about the A axis is to be measured; Set Q263 not equal to Q265 if the angle is to be measured about the B axis.

- **Traverse direction 1** Q267: Direction in which the probe is to approach the workpiece:
  -1: Negative traverse direction
  +1: Positive traverse direction
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Generate measuring log: the TNC saves the **log file TCHPR420.TXT** by default in the directory TNC:\
  2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.

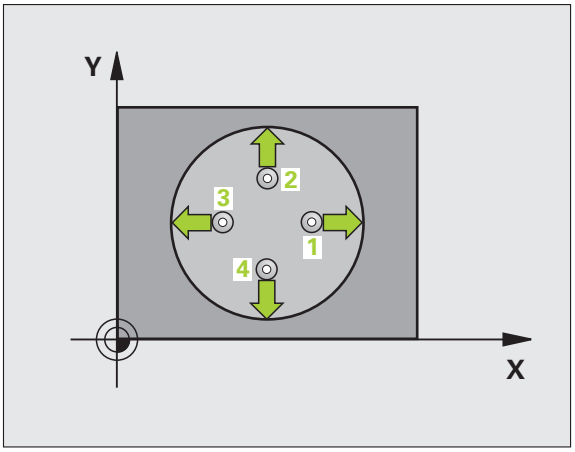


Example: NC blocks

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

MEASURE HOLE (touch probe cycle 421, DIN/ISO: G421)

Touch probe cycle 421 measures the center and diameter of a hole (or circular pocket). If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.



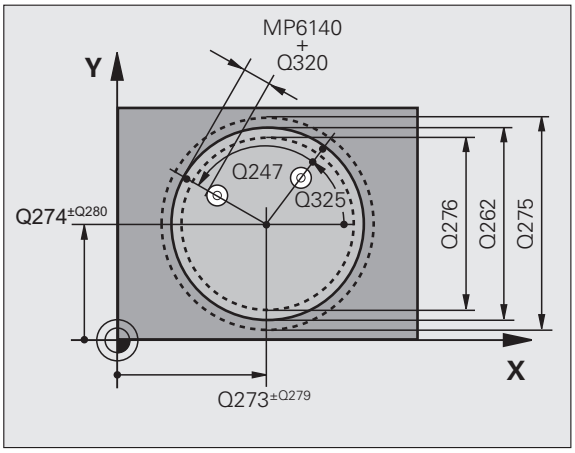
1. The TNC positions the touch probe at rapid traverse (value from FMAX column) following the positioning logic (see "Running touch probe cycles" on page 21) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table
2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column F). The TNC derives the probing direction automatically from the programmed starting angle.
3. Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point **2** and probes the second touch point.
4. The TNC positions the probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5. Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following Q parameters:

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

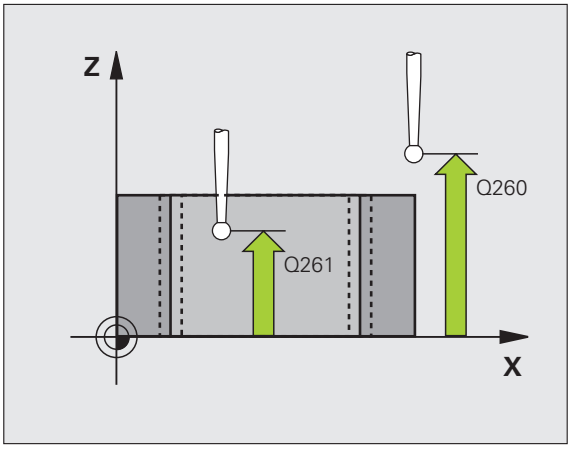
Before programming, note the following

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

- **Center in 1st axis** Q273 (absolute value): Center of the hole in the reference axis of the working plane.
- **Center in 2nd axis** Q274 (absolute value): Center of the hole in the minor axis of the working plane.
- **Nominal diameter** Q262: Enter the diameter of the hole.
- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point.
- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (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°.



The smaller the angle, the less accurately the TNC can calculate the hole dimensions. Minimum input value: 5°.



- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Maximum dimension of size for hole** Q275: Maximum permissible dimension for the hole (circular pocket).
- **Minimum dimension for hole** Q276: Minimum permissible dimension for the hole (circular pocket).
- **Tolerance value for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane.
- **Tolerance value for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane.

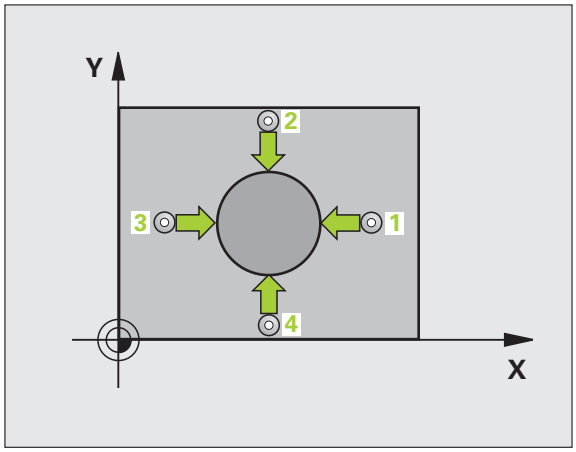
- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Generate measuring log: the TNC saves the **log file TCHPR421.TXT** by default in the directory TNC:\
  2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt the program run and output an error message:
  0: Do not interrupt program run, no error message
  1: Interrupt program run, output an error message
- **Tool number for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring" on page 108)
  0: Monitoring not active
  >0: Tool number in the tool table TOOL.T
- **No. of measuring points (4/3)** Q423: Specify whether the TNC should measure the stud with 4 or 3 probing points:
  4: Use 4 measuring points (standard setting)
  3: Use 3 measuring points (standard setting)

Example: NC blocks

```
5 TCH PROBE 421 MEASURE HOLE
    Q273=+50      ;CENTER 1ST AXIS
    Q274=+50      ;CENTER 2ND AXIS
    Q262=75       ;NOMINAL DIAMETER
    Q325=+0       ;STARTING ANGLE
    Q247=+60      ;STEPPING ANGLE
    Q261=-5       ;MEASURING HEIGHT
    Q320=0        ;SET-UP CLEARANCE
    Q260=+20      ;CLEARANCE HEIGHT
    Q301=1        ;MOVE TO CLEARANCE
    Q275=75.12    ;MAX. LIMIT
    Q276=74.95    ;MIN. LIMIT
    Q279=0.1      ;TOLERANCE 1ST CENTER
    Q280=0.1      ;TOLERANCE 2ND CENTER
    Q281=1        ;MEASURING LOG
    Q309=0        ;PGM STOP IF ERROR
    Q330=0        ;TOOL NUMBER
    Q423=4        ;NO. OF MEAS. POINTS
```

MEASURE OUTER WIDTH (touch probe cycle 422, DIN/ISO: G422)

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



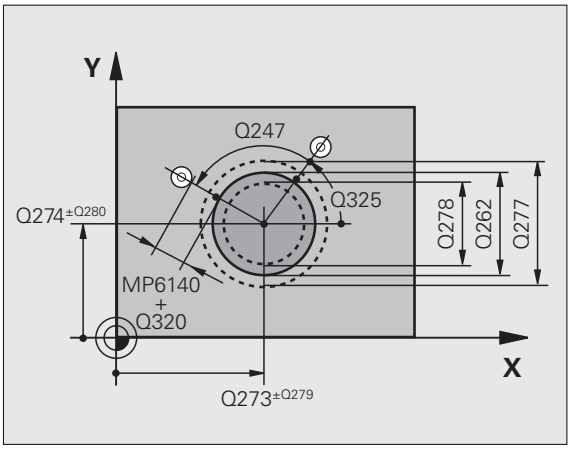
1. The TNC positions the touch probe at rapid traverse (value from FMAX column) following the positioning logic (see "Running touch probe cycles" on page 21) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table
2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column F). The TNC derives the probing direction automatically from the programmed starting angle.
3. Then the touch probe moves in a circular arc either at measuring height or at clearance height to the next starting point **2** and probes the second touch point.
4. The TNC positions the probe to starting point **3** and then to starting point **4** to probe the third and fourth touch points.
5. Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following Q parameters:

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

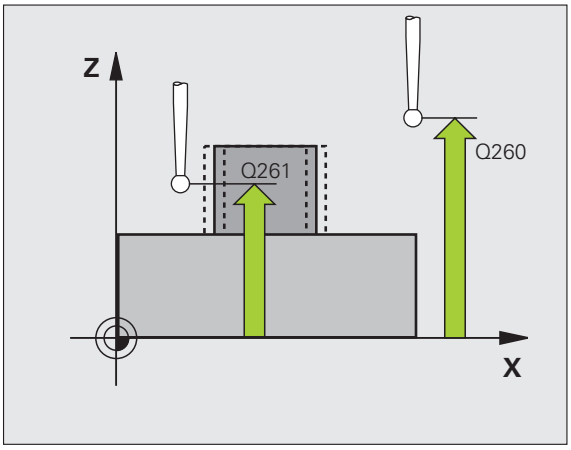
Before programming, note the following

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

- **Center in 1st axis** Q273 (absolute value): Center of the stud in the reference axis of the working plane.
- **Center in 2nd axis** Q274 (absolute value): Center of the stud in the minor axis of the working plane.
- **Nominal diameter** Q262: Enter the diameter of the stud.
- **Starting angle** Q325 (absolute): Angle between the reference axis of the working plane and the first touch point.
- **Stepping angle** Q247 (incremental): Angle between two measuring points. The algebraic sign of the stepping angle determines the direction of rotation (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°.



The smaller the angle, the less accurately the TNC can calculate the dimensions of the stud. Minimum input value: 5°



- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Maximum dimension of size for stud** Q277: Maximum permissible dimension for the stud.
- **Minimum dimension of size for the stud** Q278: Minimum permissible dimension for the stud.
- **Tolerance value for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane.
- **Tolerance value for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane.

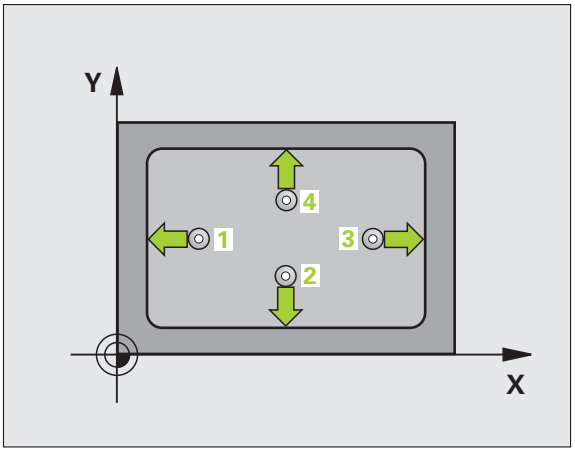
- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Generate measuring log: the TNC saves the **log file TCHPR422.TXT** by default in the directory TNC:\
  2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt the program run and output an error message:
  0: Do not interrupt program run, no error message
  1: Interrupt program run, output an error message
- **Tool number for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring" on page 108):
  0: Monitoring not active
  >0: Tool number in the tool table TOOL.T
- **No. of measuring points (4/3)** Q423: Specify whether the TNC should measure the stud with 4 or 3 probing points:
  4: Use 4 measuring points (standard setting)
  3: Use 3 measuring points (standard setting)

Example: NC blocks

```
5 TCH PROBE 422 MEAS. CIRCLE OUTSIDE
    Q273=+50      ;CENTER 1ST AXIS
    Q274=+50      ;CENTER 2ND AXIS
    Q262=75       ;NOMINAL DIAMETER
    Q325=+90      ;STARTING ANGLE
    Q247=+30      ;STEPPING ANGLE
    Q261=-5       ;MEASURING HEIGHT
    Q320=0        ;SET-UP CLEARANCE
    Q260=+10      ;CLEARANCE HEIGHT
    Q301=0        ;MOVE TO CLEARANCE
    Q275=35.15    ;MAX. LIMIT
    Q276=34.9     ;MIN. LIMIT
    Q279=0.05     ;TOLERANCE 1ST CENTER
    Q280=0.05     ;TOLERANCE 2ND CENTER
    Q281=1        ;MEASURING LOG
    Q309=0        ;PGM STOP IF ERROR
    Q330=0        ;TOOL NUMBER
    Q423=4        ;NO. OF MEAS. POINTS
```

MEASURE INSIDE RECTANGLE (touch probe cycle 423, DIN/ISO: G423)

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 TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.



1. The TNC positions the touch probe at rapid traverse (value from FMAX column) following the positioning logic (see "Running touch probe cycles" on page 21) to the starting point 1. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table
2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (column F).
3. Then the touch probe moves either paraxially at the measuring height or linearly at the clearance height to the next starting point 2 and probes the second touch point.
4. The TNC positions the probe to starting point 3 and then to starting point 4 to probe the third and fourth touch points.
5. Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following Q parameters:

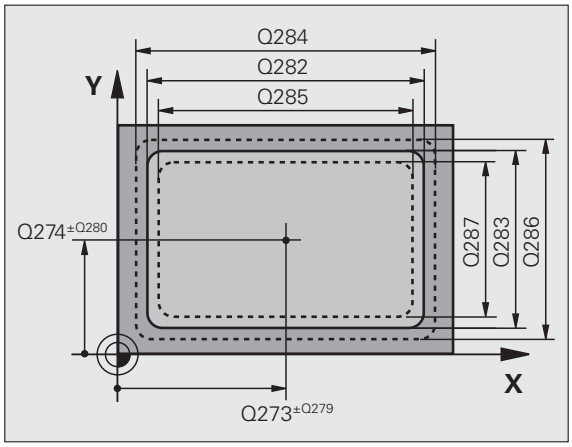
Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of length in the reference axis
Q155	Actual value of length in the minor axis
Q161	Deviation from center of reference axis
Q162	Deviation from center of minor axis
Q164	Deviation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

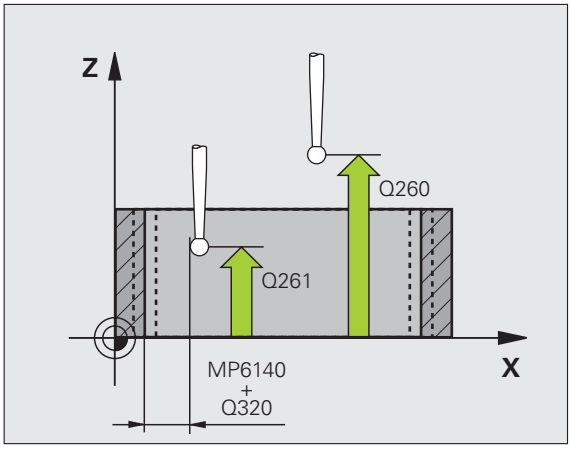
Before programming, note the following

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

If the dimensions of the pocket and the safety clearance do not permit pre-positioning in the proximity of the touch points, the TNC 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.

- **Center in 1st axis** Q273 (absolute value): Center of the pocket in the reference axis of the working plane.
- **Center in 2nd axis** Q274 (absolute value): Center of the pocket in the minor axis of the working plane.
- **First side length** Q282: pocket length, parallel to the reference axis of the working plane.
- **Second side length** Q283: pocket length, parallel to the secondary axis of the working plane.
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Max. size limit 1st side length** Q284: Maximum permissible length of the pocket.
- **Min. size limit 1st side length** Q285: Minimum permissible length of the pocket.
- **Max. size limit 2nd side length** Q286: Maximum permissible width of the pocket.
- **Min. size limit 2nd side length** Q287: Minimum permissible width of the pocket.
- **Tolerance value for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane.
- **Tolerance value for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane.





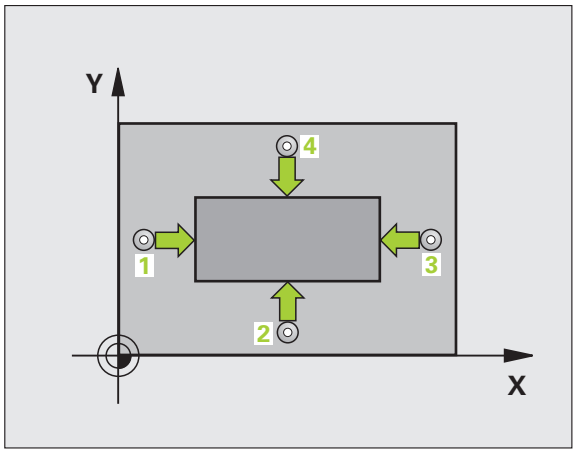
- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Generate measuring log: the TNC saves the **log file TCHPR423.TXT** by default in the directory TNC:\
  2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt the program run and output an error message:
  0: Do not interrupt program run, no error message
  1: Interrupt program run, output an error message
- **Tool number for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring" on page 108)
  0: Monitoring not active
  >0: Tool number in the tool table TOOL.T

Example: NC blocks

```
5 TCH PROBE 423 MEAS. RECTAN. INSIDE
    Q273=+50      ;CENTER 1ST AXIS
    Q274=+50      ;CENTER 2ND AXIS
    Q282=80       ;FIRST SIDE LENGTH
    Q283=60       ;SECOND SIDE LENGTH
    Q261=-5       ;MEASURING HEIGHT
    Q320=0        ;SET-UP CLEARANCE
    Q260=+10      ;CLEARANCE HEIGHT
    Q301=1        ;MOVE TO CLEARANCE
    Q284=0        ;MAX. LIMIT 1ST SIDE
    Q285=0        ;MIN. LIMIT 1ST SIDE
    Q286=0        ;MAX. LIMIT 2ND SIDE
    Q287=0        ;MIN. LIMIT 2ND SIDE
    Q279=0        ;TOLERANCE 1ST CENTER
    Q280=0        ;TOLERANCE 2ND CENTER
    Q281=1        ;MEASURING LOG
    Q309=0        ;PGM STOP IF ERROR
    Q330=0        ;TOOL NUMBER
```

MEASURE RECTANGLE FROM OUTSIDE (touch probe cycle 424, ISO: G424)

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 TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.



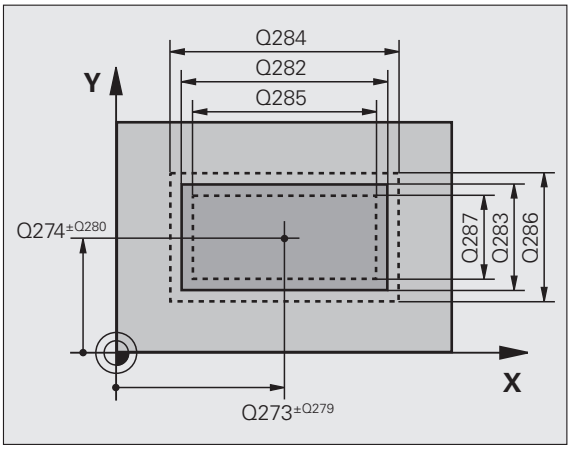
1. The TNC positions the touch probe at rapid traverse (value from FMAX column) following the positioning logic (see "Running touch probe cycles" on page 21) to the starting point 1. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table
2. Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (column F).
3. Then the touch probe moves either paraxially at the measuring height or linearly at the clearance height to the next starting point 2 and probes the second touch point.
4. The TNC positions the probe to starting point 3 and then to starting point 4 to probe the third and fourth touch points.
5. Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following Q parameters:

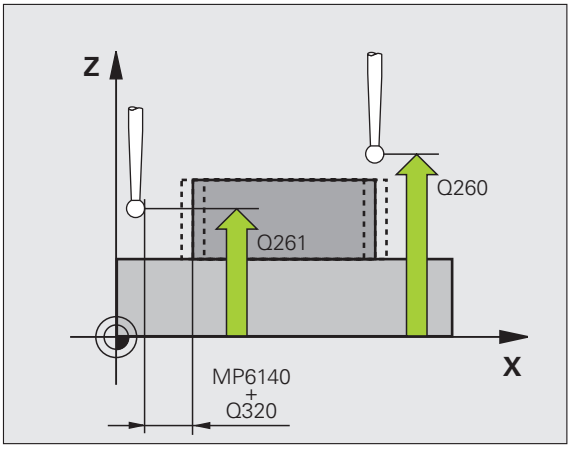
Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q154	Actual value of length in the reference axis
Q155	Actual value of length in the minor axis
Q161	Deviation from center of reference axis
Q162	Deviation from center of minor axis
Q164	Deviation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

Before programming, note the following

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

- **Center in 1st axis** Q273 (absolute value): Center of the stud in the reference axis of the working plane.
- **Center in 2nd axis** Q274 (absolute value): Center of the stud in the minor axis of the working plane.
- **First side length** Q282: stud length, parallel to the reference axis of the working plane
- **Second side length** Q283: stud length, parallel to the secondary axis of the working plane
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Traversing to clearance height** Q301: Definition of how the touch probe is to move between the measuring points:
  0: Move at measuring height between measuring points
  1: Move at clearance height between measuring points
- **Max. size limit 1st side length** Q284: Maximum permissible length of the stud.
- **Min. size limit 1st side length** Q285: Minimum permissible length of the stud.
- **Max. size limit 2nd side length** Q286: Maximum permissible width of the stud.
- **Min. size limit 2nd side length** Q287: Minimum permissible width of the stud.
- **Tolerance value for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane.
- **Tolerance value for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane.





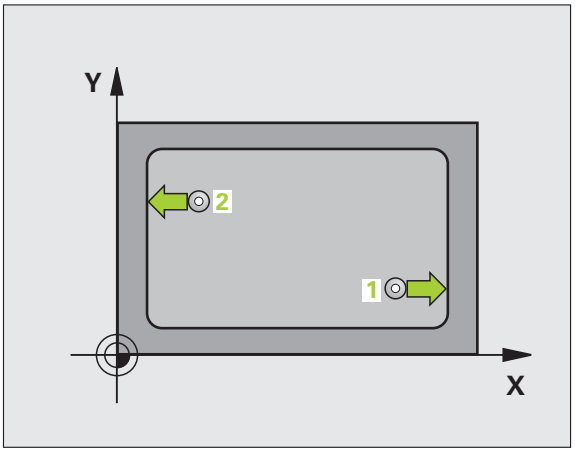
- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Generate measuring log: the TNC saves the **log file TCHPR424.TXT** by default in the directory TNC:\
  2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt the program run and output an error message:
  0: Do not interrupt program run, no error message
  1: Interrupt program run, output an error message
- **Tool number for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring" on page 108):
  0: Monitoring not active
  >0: Tool number in the tool table TOOL.T

Example: NC blocks

```
5 TCH PROBE 424 MEAS. RECTAN. OUTS.
    Q273=+50      ;CENTER 1ST AXIS
    Q274=+50      ;CENTER 2ND AXIS
    Q282=75       ;FIRST SIDE LENGTH
    Q283=35       ;SECOND SIDE LENGTH
    Q261=-5       ;MEASURING HEIGHT
    Q320=0        ;SET-UP CLEARANCE
    Q260=+20      ;CLEARANCE HEIGHT
    Q301=0        ;MOVE TO CLEARANCE
    Q284=75.1     ;MAX. LIMIT 1ST SIDE
    Q285=74.9     ;MIN. LIMIT 1ST SIDE
    Q286=35       ;MAX. LIMIT 2ND SIDE
    Q287=34.95    ;MIN. LIMIT 2ND SIDE
    Q279=0.1      ;TOLERANCE 1ST CENTER
    Q280=0.1      ;TOLERANCE 2ND CENTER
    Q281=1        ;MEASURING LOG
    Q309=0        ;PGM STOP IF ERROR
    Q330=0        ;TOOL NUMBER
```

MEASURE INSIDE WIDTH (touch probe cycle 425, DIN/ISO: G425)

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 TNC makes a nominal-to-actual value comparison and saves the deviation value in a system parameter.



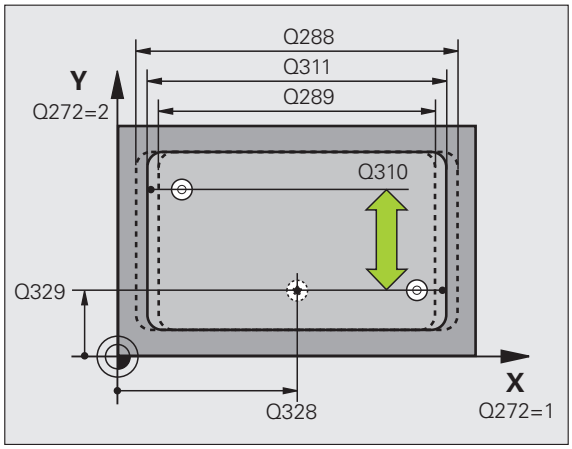
1. The TNC positions the touch probe at rapid traverse (value from FMAX column) following the positioning logic (see "Running touch probe cycles" on page 21) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table
2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column F). 1. The first probing is always in the positive direction of the programmed axis.
3. If you enter an offset for the second measurement, the TNC then moves the touch probe paraxially to the next starting point **2** and probes the second touch point. If you do not enter an offset, the TNC measures the width in the exact opposite direction.
4. Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviation in the following Q parameters:

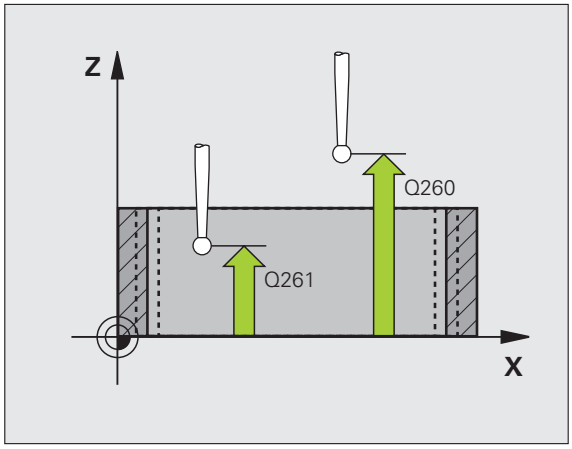
Parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

Before programming, note the following

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

- **Starting point in 1st axis** Q328 (absolute): Starting point for probing in the reference axis of the working plane.
- **Starting point in 2nd axis** Q329 (absolute): Starting point for probing in the minor axis of the working plane.
- **Offset for 2nd measurement** Q310 (incremental): Distance by which the touch probe is displaced before the second measurement. If you enter 0, the TNC does not displace the touch probe.
- **Measuring axis** Q272: Axis in the working plane in which the measurement is to be made:
  1: Reference axis = measuring axis
  2:Minor axis = measuring axis
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Nominal length** Q311: Nominal value of the length to be measured.
- **Maximum dimension** Q288: Maximum permissible length.
- **Minimum dimension** Q289: Minimum permissible length.
- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Generate measuring log: the TNC saves the **log file TCHPR425.TXT** by default in the directory TNC:\
  2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt the program run and output an error message:
  0: Do not interrupt program run, no error message
  1: Interrupt program run, output an error message
- **Tool number for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring" on page 108):
  0: Monitoring not active
  >0: Tool number in the tool table TOOL.T



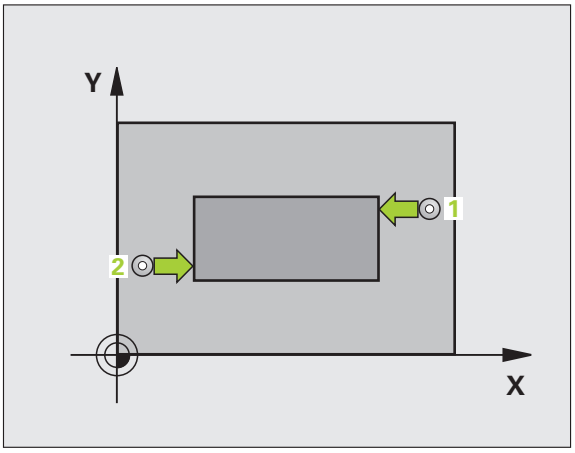


Example: NC blocks

```
5 TCH PROBE 425 MEASURE INSIDE WIDTH
    Q328=+75      ;STARTNG PNT 1ST AXIS
    Q329=-12.5    ;STARTNG PNT 2ND AXIS
    Q310=+0       ;OFFS. 2ND MEASUREMNT
    Q272=1        ;MEASURING AXIS
    Q261=-5       ;MEASURING HEIGHT
    Q260=+10      ;CLEARANCE HEIGHT
    Q311=25       ;NOMINAL LENGTH
    Q288=25.05    ;MAX. LIMIT
    Q289=25       ;MIN. LIMIT
    Q281=1        ;MEASURING LOG
    Q309=0        ;PGM STOP IF ERROR
    Q330=0        ;TOOL NUMBER
```

MEASURE OUTER RIDGE (touch probe cycle 426, DIN/ISO: G426)

Touch probe cycle 426 measures the position and width of a ridge. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.



1. The TNC positions the touch probe at rapid traverse (value from FMAX column) following the positioning logic (see "Running touch probe cycles" on page 21) to the starting point 1. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table
2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column F). 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 starting position and probes the second touch point.
4. Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviation in the following Q parameters:

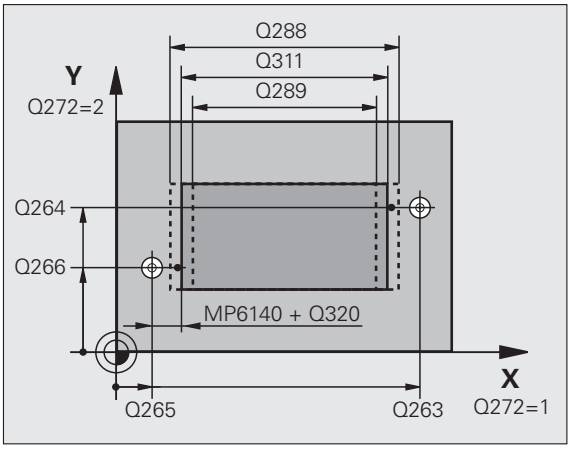
Parameter number	Meaning
Q156	Actual value of measured length
Q157	Actual value of the centerline
Q166	Deviation of the measured length

Before programming, note the following

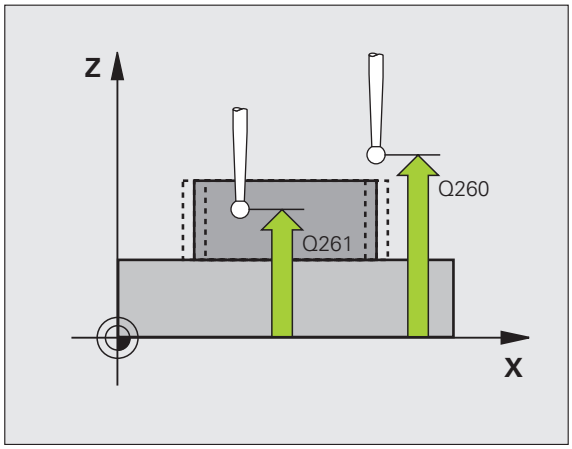
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

426

- **First measuring point in the 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane.
- **First measuring point in the 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane.
- **Second measuring point in the 1st axis** Q265 (absolute): Coordinate of the second touch point in the reference axis of the working plane.
- **Second measuring point in the 2nd axis** Q266 (absolute): Coordinate of the second touch point in the minor axis of the working plane.



- **Measuring axis** Q272: Axis in the working plane in which the measurement is to be made:
  1:Reference axis = measuring axis
  2:Minor axis = measuring axis
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Nominal length** Q311: Nominal value of the length to be measured.
- **Maximum dimension** Q288: Maximum permissible length.
- **Minimum dimension** Q289: Minimum permissible length.
- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Generate measuring log: The TNC saves the **log file TCHPR426.TXT** by default in the directory TNC:\
  2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt the program run and output an error message:
  0: Do not interrupt program run, no error message
  1: Interrupt program run, output an error message
- **Tool number for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring" on page 108)
  0: Monitoring not active
  >0: Tool number in the tool table TOOL.T

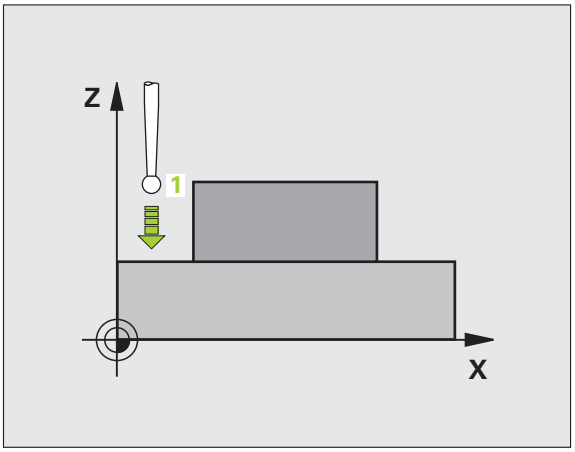


Example: NC blocks

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

MEASURE COORDINATE (touch probe cycle 427, DIN/ISO: G427)

Touch probe cycle 427 finds a coordinate in a selectable axis and saves the value in a system parameter. If you define the corresponding tolerance values in the cycle, the TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.



1 The TNC positions the touch probe at rapid traverse (value from FMAX column) following the positioning logic (see "Running touch probe cycles" on page 21) to the starting point 1. The TNC offsets the touch probe by the safety clearance in the direction opposite the defined traverse direction.

2 Then the TNC positions the touch probe to the entered touch point 1 in the working plane and measures the actual value in the selected axis.

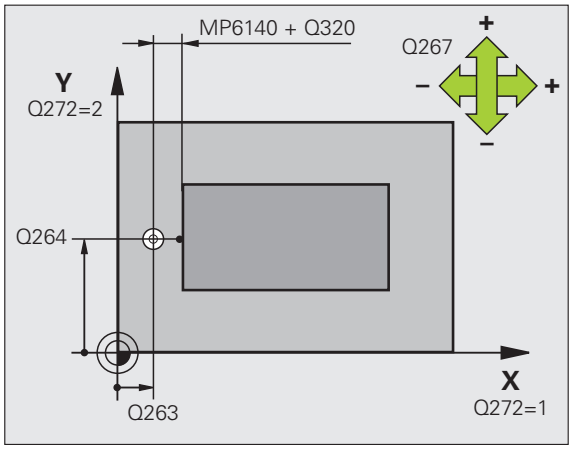
3 Finally the TNC returns the touch probe to the clearance height and saves the measured coordinate in the following Q parameter:

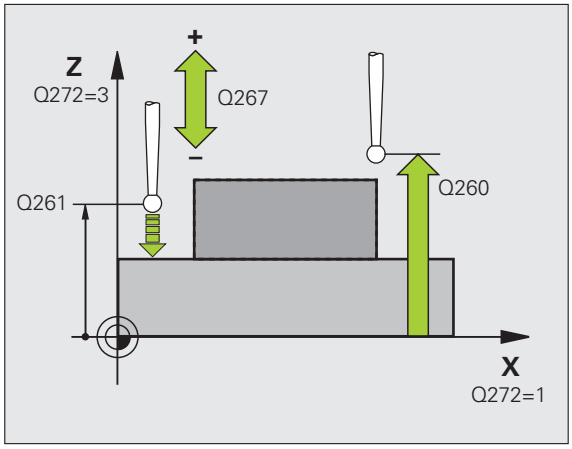
Parameter number	Meaning
Q160	Measured coordinate

Before programming, note the following

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

- **First measuring point in the 1st axis** Q263 (absolute): Coordinate of the first touch point in the reference axis of the working plane.
- **First measuring point in the 2nd axis** Q264 (absolute): Coordinate of the first touch point in the minor axis of the working plane.
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Measuring axis (1..3: 1=reference axis)** Q272: Axis in which the measurement is to be made:
  1: Reference axis = measuring axis
  2:Minor axis = measuring axis
  3: Touch probe axis = measuring axis
- **Traverse direction 1** Q267: Direction in which the probe is to approach the workpiece:
  -1: Negative traverse direction
  +1: Positive traverse direction
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.





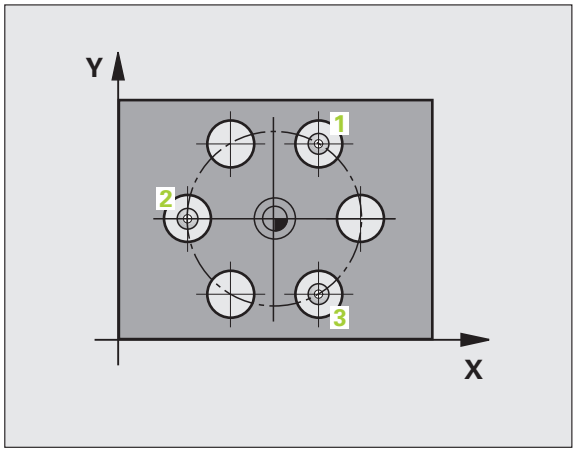
- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Generate measuring log: the TNC saves the **log file TCHPR427.TXT** by default in the directory TNC:\
  2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.
- **Maximum dimension** Q288: Maximum permissible measured value.
- **Minimum dimension** Q289: Minimum permissible measured value.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt the program run and output an error message:
  0: Do not interrupt program run, no error message
  1: Interrupt program run, output an error message
- **Tool number for monitoring** Q330: Definition of whether the TNC is to monitor the tool (see "Tool monitoring" on page 108):
  0: Monitoring not active
  >0: Tool number in the tool table TOOL.T

Example: NC blocks

```
5 TCH PROBE 427 MEASURE COORDINATE
    Q263=+35      ;1ST POINT 1ST AXIS
    Q264=+45      ;1ST POINT 2ND AXIS
    Q261=+5       ;MEASURING HEIGHT
    Q320=0        ;SET-UP CLEARANCE
    Q272=3        ;MEASURING AXIS
    Q267=-1       ;TRAVERSE DIRECTION
    Q260=+20      ;CLEARANCE HEIGHT
    Q281=1        ;MEASURING LOG
    Q288=5.1      ;MAX. LIMIT
    Q289=4.95     ;MIN. LIMIT
    Q309=0        ;PGM STOP IF ERROR
    Q330=0        ;TOOL NUMBER
```

MEAS. BOLT HOLE CIRC (touch probe cycle 430, DIN/ISO: G430)

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 TNC makes a nominal-to-actual value comparison and saves the deviation value in system parameters.



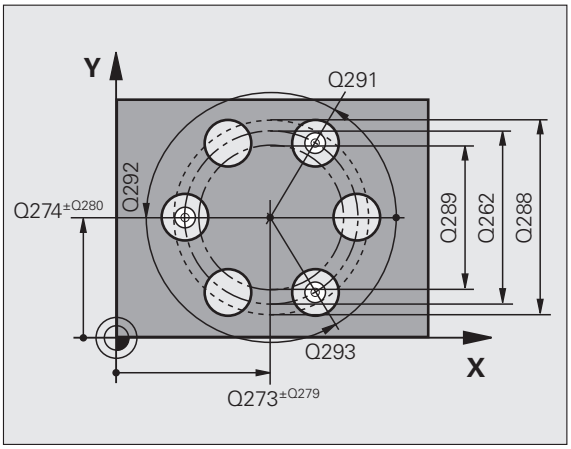
1. The TNC positions the touch probe at rapid traverse (value from column FMAX) and with positioning logic (see "Running touch probe cycles" on page 21) to the point entered as center of the first hole **1**.
2. Then the probe moves to the entered measuring height and probes four points to find the first hole center.
3. The touch probe returns to the clearance height and then to the position entered as center of the second hole **2**.
4. The TNC moves the touch probe to the entered measuring height and probes four points to find the second hole center.
5. The touch probe returns to the clearance height and then to the position entered as center of the third hole **3**.
6. The TNC moves the touch probe to the entered measuring height and probes four points to find the third hole center.
7. Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviations in the following Q parameters:

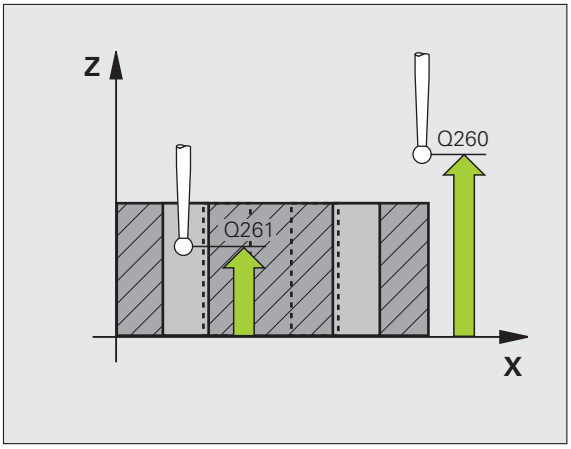
Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of bolt hole circle diameter
Q161	Deviation from center of reference axis
Q162	Deviation from center of minor axis
Q163	Deviation of bolt hole circle diameter

Before programming, note the following

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

- **Center of 1st axis** Q273 (absolute): Bolt hole circle center (nominal value) in the reference axis of the working plane.
- **Center in 2nd axis** Q274 (absolute): Bolt hole circle center (nominal value) in the minor axis of the working plane.
- **Nominal diameter** Q262: Enter the bolt hole circle diameter.
- **Angle of 1st hole** Q291 (absolute): Polar coordinate angle of the first hole center in the working plane.
- **Angle of 2nd hole** Q292 (absolute): Polar coordinate angle of the second hole center in the working plane.
- **Angle of 3rd hole** Q293 (absolute): Polar coordinate angle of the third hole center in the working plane.
- **Measuring height in the touch probe axis** Q261 (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Maximum dimension** Q288: Maximum permissible diameter of bolt hole circle.
- **Minimum dimension** Q289: Minimum permissible diameter of bolt hole circle.
- **Tolerance value for center 1st axis** Q279: Permissible position deviation in the reference axis of the working plane.
- **Tolerance value for center 2nd axis** Q280: Permissible position deviation in the minor axis of the working plane.





- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Generate measuring log: the TNC saves the **log file TCHPR430.TXT** by default in the directory TNC:\
  2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.
- **PGM stop if tolerance error** Q309: Definition of whether in the event of a violation of tolerance limits the TNC is to interrupt the program run and output an error message:
  0: Do not interrupt program run, no error message
  1: Interrupt program run, output an error message
- **Tool number for monitoring** Q330: Definition of whether the TNC is to monitor for tool breakage (see "Tool monitoring" on page 108):
  0: Monitoring not active
  >0: Tool number in the tool table TOOL.T

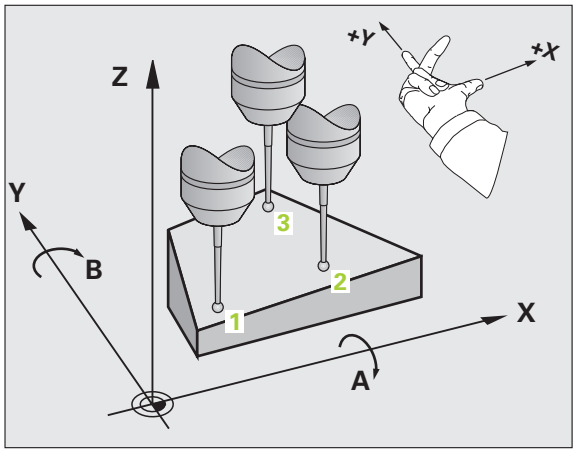
Note: Only tool breakage monitoring active, no automatic tool compensation.

Example: NC blocks

```
5 TCH PROBE 430 MEAS. BOLT HOLE CIRC
    Q273=+50      ;CENTER 1ST AXIS
    Q274=+50      ;CENTER 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     ;MAX. LIMIT
    Q289=79.9     ;MIN. LIMIT
    Q279=0.15     ;TOLERANCE 1ST CENTER
    Q280=0.15     ;TOLERANCE 2ND CENTER
    Q281=1        ;MEASURING LOG
    Q309=0        ;PGM STOP IF ERROR
    Q330=0        ;TOOL NUMBER
```

MEASURE PLANE (touch probe cycle 431, DIN/ISO: G431)

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



1. Following the positioning logic (see "Running touch probe cycles" on page 21), the TNC positions the touch probe at rapid traverse (value from FMAX column) to the programmed starting point **1** and measures the first touch point of the plane. The TNC offsets the touch probe by the safety clearance in the direction opposite to the direction of probing.
2. The touch probe returns to the clearance height and then moves in the working plane to starting point **2** and measures the actual value of the second touch point of the plane.
3. The touch probe returns to the clearance height and then moves in the working plane to starting point **3** and measures the actual value of the third touch point.
4. Finally the TNC returns the touch probe to the clearance height and saves the measured angle values in the following Q parameters:

Parameter number	Meaning
Q158	Projection angle of the A axis
Q159	Projection angle of the B axis
Q170	Spatial angle A
Q171	Spatial angle B
Q172	Spatial angle C
Q173	Measured value in the touch probe axis

Before programming, note the following

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

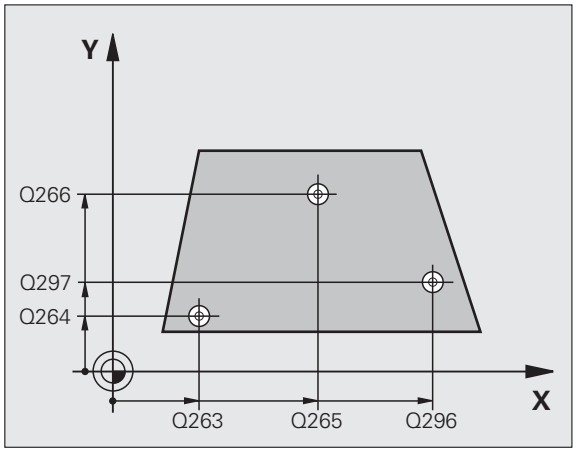
For the TNC to be able to calculate the angular values, the three measuring points must not be positioned on one straight line.

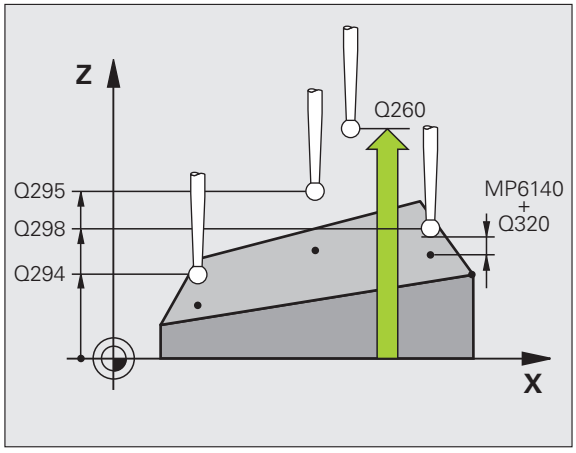
The spatial angles that are needed for tilting the working plane are saved in parameters Q170 – Q172. With the first two measuring points you also specify the direction of the reference axis when tilting the working plane.

The third measuring point determines the direction of the tool axis. Define the third measuring point in the direction of the positive Y axis to ensure that the position of the tool axis in a clockwise coordinate system is correct (see figure).

If you run the cycle while a tilted working plane is active, the spatial angle is measured with respect to the tilted coordinate system. In these cases use the determined spatial angle via incremental input with the tilting the working plane function.

- **1st measuring point in 1st axis** Q263 (absolute): coordinate of the first touch point in the reference axis of the working plane.
- **1st measuring point in 2nd axis** Q264 (absolute): coordinate of the first touch point in the minor axis of the working plane.
- **1st measuring point in 3rd axis** Q294 (absolute): coordinate of the first touch point in the touch probe axis.
- **2nd measuring point in 1st axis** Q265 (absolute): coordinate of the second touch point in the reference axis of the working plane
- **2nd measuring point in 2nd axis** Q266 (absolute): coordinate of the second touch point in the minor axis of the working plane
- **2nd measuring point in 3rd axis** Q295 (absolute): coordinate of the second touch point in the touch probe axis.
- **3rd measuring point in 1st axis** Q296 (absolute): coordinate of the third touch point in the reference axis of the working plane.
- **3rd measuring point in 2nd axis** Q297 (absolute): coordinate of the third touch point in the minor axis of the working plane.
- **3rd measuring point in 3rd axis** Q298 (absolute): coordinate of the third touch point in the touch probe axis.
- **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to column SET_UP.
- **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
- **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
  0: No measuring log
  1: Generate measuring log: the TNC saves the **log file TCHPR431.TXT** by default in the directory TNC:\
  2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.





Example: NC blocks

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

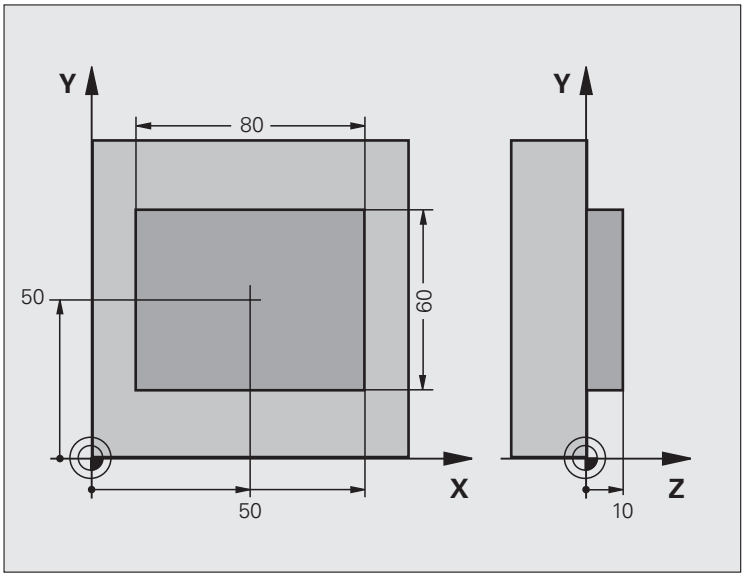
Example: Measuring and reworking a rectangular stud

Program sequence:

- Roughing with 0.5 mm finishing allowance

- Measuring

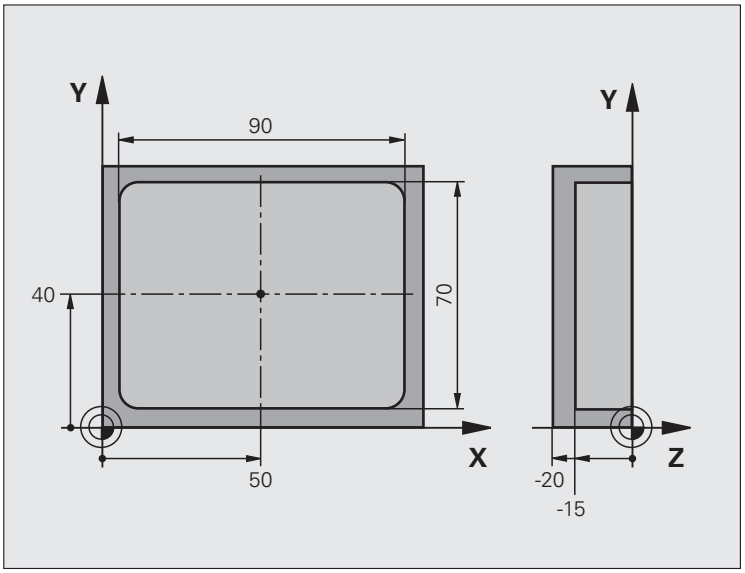
- Rectangular stud finishing in accordance with the measured values



`0 BEGIN PGM BEAMS MM`	
`1 TOOL CALL 0 Z`	Prepare tool call
`2 L Z+100 R0 FMAX`	Retract the tool.
`3 FN 0: Q1 = +81`	Pocket length in X (roughing dimension)
`4 FN 0: Q2 = +61`	Pocket length in Y (roughing dimension)
`5 CALL LBL 1`	Call subprogram for machining
`6 L Z+100 R0 FMAX`	Retract the tool, change the tool
`7 TOOL CALL 99 Z`	Call the touch probe
`8 TCH PROBE 424 MEAS. RECTAN. OUTS.`	Measure the rough-milled rectangle
`Q273=+50 ;CENTER 1ST AXIS`	
`Q274=+50 ;CENTER 2ND AXIS`	
`Q282=80 ;FIRST SIDE LENGTH`	Nominal length in X (final dimension)
`Q283=60 ;SECOND SIDE LENGTH`	Nominal length in Y (final dimension)
`Q261=-5 ;MEASURING HEIGHT`	
`Q320=0 ;SET-UP CLEARANCE`	
`Q260=+30 ;CLEARANCE HEIGHT`	
`Q301=0 ;MOVE TO CLEARANCE`	
`Q284=0 ;MAX. LIMIT 1ST SIDE`	Input values for tolerance checking not required
`Q285=0 ;MIN. LIMIT 1ST SIDE`	
`Q286=0 ;MAX. LIMIT 2ND SIDE`	

Q287=0 ;MIN. LIMIT 2ND SIDE	
Q279=0 ;TOLERANCE 1ST CENTER	
Q280=0 ;TOLERANCE 2ND CENTER	
Q281=0 ;MEASURING LOG	No measuring log transmission
Q309=0 ;PGM STOP IF ERROR	Do not output an error message
Q330=0 ;TOOL NUMBER	No tool monitoring
9 FN 2: Q1 = +Q1 - +Q164	Calculate length in X including the measured deviation
10 FN 2: Q2 = +Q2 - +Q165	Calculate length in Y including the measured deviation
11 L Z+100 R0 FMAX	Retract the touch probe, change the tool
12 TOOL CALL 1 Z S5000	Tool call for finishing
13 CALL LBL 1	Call subprogram for machining
14 L Z+100 R0 FMAX M2	Retract in the tool axis, end program
15 LBL 1	Subprogram with fixed cycle for rectangular studs
16 CYCL DEF 213 STUD FINISHING	
Q200=20 ;SET-UP CLEARANCE	
Q201=-10 ;DEPTH	
Q206=150 ;FEED RATE FOR PLUNGING	
Q202=5 ;PLUNGING DEPTH	
Q207=500 ;FEED RATE FOR MILLING	
Q203=+10 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q216=+50 ;CENTER 1ST AXIS	
Q217=+50 ;CENTER 2ND AXIS	
Q218=Q1 ;FIRST SIDE LENGTH	Length in X variable for roughing and finishing
Q219=Q2 ;2ND SIDE LENGTH	Length in Y variable for roughing and finishing
Q220=0 ;CORNER RADIUS	
Q221=0 ;ALLOWANCE IN 1ST AXS	
17 CYCL CALL M3	Cycle call
18 LBL 0	End of subprogram
19 END PGM BEAMS MM	

Example: Measuring a rectangular pocket and recording the results



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

`4 L Z+100 R0 FMAX M2`	Retract in the tool axis, end program
`5 END PGM BSMESS MM`	

3.4 Special Cycles

Overview

The TNC provides a cycle for the following special purpose:

Cycle	Soft key	Page
3 MEASURING Cycle for defining OEM cycles		Page 144

MEASURING (touch probe cycle 3)

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

Touch probe cycle 3 measures any position on the workpiece in a selectable direction. Unlike other measuring cycles, Cycle 3 enables you to enter the measuring path **DIST**and feed rate **F** directly. Also, the touch probe retracts by a definable value after determining the measured value **MB**.

1. The touch probe moves from the current position at the entered feed rate in the defined probing direction. The probing direction must be defined in the cycle as a polar angle.
2. After the TNC has saved the position, the touch probe stops. The TNC saves the X, Y, Z coordinates of the probe-tip center in three successive Q parameters. The TNC does not conduct any length or radius compensations. You define the number of the first result parameter in the cycle.
3. Finally, the TNC moves the touch probe back by that value against the probing direction that you defined in the parameter **MB.**

Before programming, note the following

The machine parameters **DIST** (maximum traverse to touch point) and **F** (probing feed rate), which are effective in other measuring cycles, do not apply in touch probe cycle 3.

Remember that the TNC always writes to 4 successive Q parameters.

If the TNC was not able to determine a valid touch point, the program is run without error message. In this case the TNC assigns the value –1 to the 4th result parameter so that you can deal with the error yourself.

The TNC retracts the touch probe by no more that the retraction distance **MB** and does not pass the starting point of the measurement. This rules out any collision during retraction.

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

- **Parameter number for result:** Enter the number of the Q parameter to which you want the TNC to assign the first measured coordinate (X). The values Y and Z are in the immediately following Q parameters.
- **Probing angle:** Enter the angle in whose direction the probe is to move and confirm with the ENT key.
- **Probing angle:** Angle, measured from the **probing axis** at which the touch probe is to move. Confirm with ENT
- **Maximum measuring path:** Enter the maximum distance from the starting point by which the touch probe is to move. Confirm with ENT.
- **Feed rate:** Enter the measuring feed rate in mm/min.
- **Maximum retraction path:** Traverse path in the direction opposite the probing direction, after the stylus was deflected. The TNC returns the touch probe to a point no farther than the starting point, so that there can be no collision.
- **REFERENCE SYSTEM (0=ACT/1=REF):** Specify whether the result of measurement is to be saved in the actual coordinate system (ACT, can therefore be shifted or rotated), or with respect to the machine coordinate system (REF).
- **Error mode (0=OFF/1=ON):** Specify whether the TNC is to issue an error message if the stylus is deflected at cycle start (**0**) or not (**1**). If you select mode **1,** the TNC continues to save the value **2.0** in the 4th result parameter and continues the cycle.
- To conclude the input, press the ENT key.

Example: NC blocks

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

4

Touch Probe Cycles for Automatic Tool Measurement

4.1 Tool Measurement with the TT Tool Touch Probe

Overview

The TNC and the machine tool must be set up by the machine tool builder for use of the TT touch probe.

Some cycles and functions may not be provided on your machine tool. Refer to your machine manual.

In conjunction with the TNC's tool measurement cycles, the tool touch probe enables you to measure tools automatically. The compensation values for tool length and radius can be stored in the central tool file TOOL.T and are accounted for at the end of the touch probe cycle. The following types of tool measurement are provided:

- Tool measurement while the tool is at standstill.
- Tool measurement while the tool is rotating.
- Measuring individual teeth.

Setting the machine parameters

Before you start work with the TT cycles, inspect all machine parameters defined in **ProbSettings** > **CfgToolMeasurement** and **CfgTTRoundStylus**.

The TNC uses the feed rate for probing defined in **probingFeed** when measuring a tool at standstill.

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

The spindle speed is calculated as follows:

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

n	Spindle speed (rpm)
maxPeriphSpeedMeas	Maximum permissible cutting speed in m/min
r	Active tool radius in mm

The feed rate for probing is calculated from:

v = meas. tolerance • n where

v	Feed rate for probing in mm/min
Measuring tolerance	Measuring tolerance [mm], depending on **maxPeriphSpeedMeas**
n	Speed in rpm

probingFeedCalc determines the calculation of the probing feed rate:

probingFeedCalc = **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 smaller you set the maximum permissible rotational speed (**maxPeriphSpeedMeas**) and the permissible tolerance (**measureTolerance1**), the sooner you will encounter this effect.

probingFeedCalc = **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 TNC adjusts the measuring tolerance according to the following table:

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

probingFeedCalc = **ConstantFeed**:

The feed rate for probing remains constant, the error of measurement, however, rises linearly with the increase in tool radius:

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

r	Active tool radius in mm
measureTolerance1	Maximum permissible error of measurement

Entries in the tool table TOOL.T

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

Input examples for common tool types

Tool type	CUT	R-OFFS	L-OFFS
Drill	– (no function)	0 (no offset required because tool tip is to be measured)	
End mill with diameter of < 19 mm	4 (4 teeth)	0 (no offset required because tool diameter is smaller than the contact plate diameter of the TT)	0 (no additional offset required for radius calibration; offset from **offsetToolAxis** is used.)
End mill with diameter of > 19 mm	4 (4 teeth)	R (offset required because tool diameter is larger that the contact plate diameter of the TT)	0 (no additional offset required for radius calibration; offset from **offsetToolAxis** is used.)
Radius cutter	4 (4 teeth)	0 (no offset required because the south pole of the ball is to be measured)	5 (always define the tool radius as the offset so that the diameter is not measured in the radius)

4.2 Available Cycles

Overview

You can program the cycles for tool measurement in the Programming mode of operation via the TOUCH PROBE key. The following cycles are available:

Cycle	Old format	New format
Calibrating the TT		
Measuring the tool length		
Measuring the tool radius		
Measuring tool length and radius		

The measuring cycles can be used only when the central tool file TOOL.T is active.

Before working with the measuring cycles, you must first enter all the required data into the central tool file and call the tool to be measured with TOOL CALL.

Differences between Cycles 31 to 33 and Cycles 481 to 483

The features and the operating sequences are absolutely identical. There are only two differences between Cycles 31 to 33 and Cycles 481 to 483:

- Cycles 481 to 483 are also available in TNCs for ISO programming under G481 to G483.
- Instead of a selectable parameter for the status of the measurement, the new cycles use the fixed parameter Q199.

Calibrating the TT (touch probe cycle 30 or 480, DIN/ISO: G480)

The method of function of the calibration cycle is determined by the machine manufacturer. Refer to your machine manual.

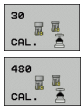
Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the tool table TOOL.T.

The position of the TT within the machine working space must be defined by setting the Machine Parameters **centerPos** > **[0]** to **[2]**.

If you change the setting of any of the Machine Parameters **centerPos** > **[0]** to **[2]**, you must recalibrate.

The TT is calibrated with the measuring cycle TCH PROBE 30 or TCH PROBE 480 (see also "Differences between Cycles 31 to 33 and Cycles 481 to 483" on page 152). The calibration process is automatic. The TNC also measures the center misalignment of the calibrating tool automatically by rotating the spindle by 180° after the first half of the calibration cycle.

The calibrating tool must be a precisely cylindrical part, for example a cylinder pin. The resulting calibration values are stored in the TNC memory and are accounted for during subsequent tool measurement.



- **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 datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from **safetyDistStylus**).

Example: NC blocks in old format

```
6 TOOL CALL 1 Z
7 TCH PROBE 30.0 CALIBRATE TT
8 TCH PROBE 30.1 HEIGHT: +90
```

Example: NC blocks in new format

```
6 TOOL CALL 1 Z
7 TCH PROBE 480 CALIBRATE TT
    Q260=+100     ;CLEARANCE HEIGHT
```

Measuring the tool length (touch probe cycle 31 or 481, DIN/ISO: G481)

Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

To measure the tool length, program the measuring cycle TCH PROBE 31 or TCH PROBE 480 (see also "Differences between Cycles 31 to 33 and Cycles 481 to 483" on page 152). Depending on the input parameters, you can measure the length of a tool by three methods:

- If the tool diameter is larger than the diameter of the measuring surface of the TT, you can 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 can measure the tool while it is at standstill.
- If the tool diameter is larger than the diameter of the measuring surface of the TT, you can measure the individual teeth of the tool while it is at standstill.

Measuring cycle for measuring a tool during rotation

The TNC 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 system and then moving it toward the measuring surface until it contacts the surface. The offset is programmed in the tool table under Tool offset: Radius (**R-OFFS**).

Measuring cycle for measuring a tool during standstill (e.g. for drills)

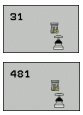
The TNC positions the tool to be measured over the center of the measuring surface. It then moves the non-rotating tool toward the measuring surface of the TT until it touches the surface. To activate this function, enter zero for the Tool offset: Radius (**R-OFFS**) in the tool table.

Measuring cycle for measuring individual teeth

The TNC 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**. You can enter an additional offset with Tool offset: Length (**L-OFFS**) in the tool table. The TNC probes the tool radially during rotation to determine the starting angle for measuring the individual teeth. It then measures the length of each tooth by changing the corresponding angle of spindle orientation. To activate this function, program TCH PROBE 31 = 1 for CUTTER MEASUREMENT.

You can run an individual tooth measurement of tools with up to 20 teeth.

Define cycle



- **Measure tool=0 / Check tool=1:** Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool length L in the central tool file TOOL.T by the delta value DL = 0. If you wish to inspect a tool, the TNC compares the measured length with the tool length L that is stored in TOOL.T. It then calculates the positive or negative deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation can also be used for Q parameter Q115. If the delta value is greater than the permissible tool length tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T).
- **Parameter number for result?:** Parameter number in which the TNC stores the status of the measurement:
  0.0: Tool is within the tolerance
  1.0: Tool is worn (**LTOL** exceeded)
  2.0: Tool is broken (**LBREAK** exceeded). If you do not wish to use the result of measurement within the program, answer the dialog prompt with NO ENT.
- **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 datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from **safetyDistStylus**).
- **Cutter measurement? 0=No / 1=Yes:** Choose whether the TNC is to measure the individual teeth (maximum of 20 teeth)

Example: Measuring a rotating tool for the first time; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 31.0 TOOL LENGTH
8 TCH PROBE 31.1 CHECK: 0
9 TCH PROBE 31.2 HEIGHT: +120
10 TCH PROBE 31.3 PROBING THE TEETH: 0
```

Example: Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 31.0 TOOL LENGTH
8 TCH PROBE 31.1 CHECK: 1 Q5
9 TCH PROBE 31.2 HEIGHT: +120
10 TCH PROBE 31.3 PROBING THE TEETH: 1
```

Example: NC blocks in new format

```
6 TOOL CALL 12 Z
7 TCH PROBE 481 TOOL LENGTH
    Q340=1        ;CHECK
    Q260=+100     ;CLEARANCE HEIGHT
    Q341=1        ;PROBING THE TEETH
```

Measuring the tool radius (touch probe cycle 32 or 482, ISO: G482)

Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

To measure the tool radius, program the cycle TCH PROBE 32 or TCH PROBE 482 (see also "Differences between Cycles 31 to 33 and Cycles 481 to 483" on page 152). Via input parameters you can measure the radius of a tool by two methods:

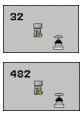
- Measuring the tool while it is rotating.
- Measuring the tool while it is rotating and subsequently measuring the individual teeth.

Cylindrical tools with diamond surfaces can be measured with stationary spindle. To do so, define the number of teeth (CUT) with 0 in the tool table and adjust the machine parameter **CfgToolMeasurement**. Refer to your machine manual.

Measuring cycle

The TNC 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 milling tool to the upper edge of the touch probe head is defined in **offsetToolAxis**. The TNC probes the tool radially while it is rotating. If you have programmed a subsequent measurement of individual teeth, the TNC measures the radius of each tooth with the aid of oriented spindle stops.

Define cycle



- **Measure tool=0 / Check tool=1:** Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool radius R in the central tool file TOOL.T by the delta value DR = 0. If you wish to inspect a tool, the TNC compares the measured radius with the tool radius R that is stored in TOOL.T. It then calculates the positive or negative deviation from the stored value and enters it into TOOL.T as the delta value DR. The deviation can also be used for Q parameter Q116. If the delta value is greater than the permissible tool radius tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T).
- **Parameter number for result?:** Parameter number in which the TNC stores the status of the measurement:
  0.0: Tool is within the tolerance
  1.0: Tool is worn (**RTOL** exceeded)
  2.0: Tool is broken (**RBREAK** exceeded). If you do not wish to use the result of measurement within the program, answer the dialog prompt with NO ENT.
- **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 datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from **safetyDistStylus**).
- **Cutter measurement? 0=No / 1=Yes:** Choose whether the TNC is also to measure the individual teeth (maximum of 20 teeth)

Example: Measuring a rotating tool for the first time; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 32.0 TOOL RADIUS
8 TCH PROBE 32.1 CHECK: 0
9 TCH PROBE 32.2 HEIGHT: +120
10 TCH PROBE 32.3 PROBING THE TEETH: 0
```

Example: Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 32.0 TOOL RADIUS
8 TCH PROBE 32.1 CHECK: 1 Q5
9 TCH PROBE 32.2 HEIGHT: +120
10 TCH PROBE 32.3 PROBING THE TEETH: 1
```

Example: NC blocks in new format

```
6 TOOL CALL 12 Z
7 TCH PROBE 482 TOOL RADIUS
    Q340=1       ;CHECK
    Q260=+100    ;CLEARANCE HEIGHT
    Q341=1       ;PROBING THE TEETH
```

Measuring the tool radius (touch probe cycle 33 or 483, ISO: G483)

Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

To measure both the length and radius of a tool, program the measuring cycles TCH PROBE 33 or TCH PROBE 482 (see also "Differences between Cycles 31 to 33 and Cycles 481 to 483" on page 152). This cycle is particularly suitable for the first measurement of tools, as it saves time when compared with individual measurement of length and radius. In input parameters you can select the desired type of measurement:

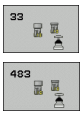
- Measuring the tool while it is rotating.
- Measuring the tool while it is rotating and subsequently measuring the individual teeth.

Cylindrical tools with diamond surfaces can be measured with stationary spindle. To do so, define the number of teeth (CUT) with 0 in the tool table and adjust the machine parameter **CfgToolMeasurement**. Refer to your machine manual.

Measuring cycle

The TNC 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 measuring cycles 31 and 32.

Define cycle



- **Measure tool=0 / Check tool=1:** Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool radius R and the tool length L in the central tool file TOOL.T by the delta values DR = 0 and DL = 0. If you wish to inspect a tool, the TNC compares the measured data with the tool data stored in TOOL.T. The TNC calculates the deviations and enters them as positive or negative delta values DR and DL in TOOL.T. The deviations are also available in the Q parameters Q115 and Q116. If the delta values are greater than the permissible tool tolerances for wear or break detection, the TNC will lock the tool (status L in TOOL.T).
- **Parameter number for result?:** Parameter number in which the TNC stores the status of the measurement:
  0.0: Tool is within the tolerance
  1.0: Tool is worn (**LTOL** or/and **RTOL** exceeded)
  2.0: Tool is broken (**LBREAK** or/and **RBREAK** exceeded).
  If you do not wish to use the result of measurement within the program, answer the dialog prompt with NO ENT.
- **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 datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from **safetyDistStylus**).
- **Cutter measurement? 0=No / 1=Yes:** Choose whether the TNC is also to measure the individual teeth (maximum of 20 teeth)

Example: Measuring a rotating tool for the first time; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 33.0 MEASURE TOOL
8 TCH PROBE 33.1 CHECK: 0
9 TCH PROBE 33.2 HEIGHT: +120
10 TCH PROBE 33.3 PROBING THE TEETH: 0
```

Example: Inspecting a tool and measuring the individual teeth and saving the status in Q5; old format

```
6 TOOL CALL 12 Z
7 TCH PROBE 33.0 MEASURE TOOL
8 TCH PROBE 33.1 CHECK: 1 Q5
9 TCH PROBE 33.2 HEIGHT: +120
10 TCH PROBE 33.3 PROBING THE TEETH: 1
```

Example: NC blocks in new format

```
6 TOOL CALL 12 Z
7 TCH PROBE 483 MEASURE TOOL
    Q340=1        ;CHECK
    Q260=+100     ;CLEARANCE HEIGHT
    Q341=1        ;PROBING THE TEETH
```

Symbole
3-D touch probes ... 16
Calibrating
Triggering ... 29

A
Angle of a plane, measuring ... 136
Automatic tool measurement ... 150

B
Basic rotation
Measuring during program run ... 42
Measuring in the Manual Operation mode ... 32
Bolt hole circle, measuring ... 133

C
Calibrating 3-D touch probes

Circle, measuring from inside ... 114
Circle, measuring from outside ... 117
Classification of results ... 107
Compensating workpiece misalignment
By measuring two points of a line ... 44
by measuring two points of a straight surface ... 32
Over two holes ... 46
Over two studs ... 49
Via rotary axis ... 52, 57
Confidence range ... 20
Consider basic rotation ... 16
Coordinate, measuring a single ... 130

D
Datum
Save in a datum table ... 64
Save in the preset table ... 64
Datum setting, manual
Circle center as datum ... 36
Corner as datum ... 35
In any axis ... 34
Datum table
Confirming probed values ... 27

F
FCL function ... 4
Feature content level ... 4
For automatic tool measurement, see tool measurement

H
Hole, measuring ... 114

M
Machine parameters for 3-D touch probes ... 19
Measurement results in Q parameters ... 64, 107
Measuring angles ... 112
Measuring the angle of a plane ... 136
Multiple measurements ... 20

P
Positioning logic ... 21
Preset table ... 64
Confirming probed values ... 28
Presetting automatically ... 61
Center of 4 holes ... 96
Center of bolt hole circle ... 91
Center of circular pocket (or hole) ... 76
Center of circular stud ... 81
Center of rectangular pocket ... 71
Center of rectangular stud ... 74
In any axis ... 99
In inside corner ... 88
In the touch probe axis ... 94
Outside corner ... 85
Ridge center ... 68
Slot center ... 65
Probing cycles
Manual operation mode ... 26
Probing feed rate ... 20

R
Recording the results of measurement ... 106
Rectangular pocket measurement ... 123
Rectangular stud, measuring ... 120
Result parameters ... 64, 107
Ridge, measuring from outside ... 128

S
Setting basic rotation ... 56
Slot width, measuring ... 126

T
Tolerance monitoring ... 108
Tool compensation ... 108
Tool measurement ... 150
Calibrating the TT ... 153
Machine parameters ... 148
Measuring tool length and radius ... 159
Overview ... 152
Tool length ... 154
Tool radius ... 157
Tool monitoring ... 108
Touch probe cycles
Touch probe cycles for automatic operation ... 18
Touch probe data ... 23
Touch probe table ... 22

W
Width, measuring from inside ... 126
Width, measuring from outside ... 128
Workpiece measurement ... 37, 105
Writing probed values in datum tables ... 27
Writing probed values in preset table ... 28

Overview

Touch probe cycles

Cycle number	Cycle designation	DEF-active	CALL-active	Page
0	Reference plane	■		Page 110
1	Polar datum	■		Page 111
3	Measuring	■		Page 144
30	Calibrating the TT	■		Page 153
31	Measure/Inspect the tool length	■		Page 154
32	Measure/Inspect the tool radius	■		Page 157
33	Measure/Inspect the tool length and the tool radius	■		Page 159
400	Basic rotation using two points	■		Page 44
401	Basic rotation from two holes	■		Page 46
402	Basic rotation from two studs	■		Page 49
403	Compensate misalignment with rotary axis	■		Page 52
404	Set basic rotation	■		Page 56
405	Compensate misalignment with the C axis	■		Page 57
408	Reference point at slot center (FCL 3 function)	■		Page 65
409	Reference point at ridge center (FCL 3 function)	■		Page 68
410	Datum from inside of rectangle	■		Page 71
411	Datum from outside of rectangle	■		Page 74
412	Datum from inside of circle (hole)	■		Page 76
413	Datum from outside of circle (stud)	■		Page 81
414	Datum from outside of corner	■		Page 85
415	Datum from inside of corner	■		Page 88
416	Datum from circle center	■		Page 91
417	Datum in touch probe axis	■		Page 94
418	Datum at center between four holes	■		Page 96
419	Datum in any one axis	■		Page 99
420	Workpiece—measure angle	■		Page 112

Cycle number	Cycle designation	DEF-active	CALL-active	Page
421	Workpiece—measure hole (center and diameter of hole)	■		Page 114
422	Workpiece—measure circle from outside (diameter of circular stud)	■		Page 117
423	Workpiece—measure rectangle from inside	■		Page 120
424	Workpiece—measure rectangle from outside	■		Page 123
425	Workpiece—measure inside width (slot)	■		Page 126
426	Workpiece—measure outside width (ridge)	■		Page 128
427	Workpiece—measure in any selectable axis	■		Page 130
430	Workpiece—measure bolt hole circle	■		Page 133
431	Workpiece—measure plane	■		Page 136
480	Calibrating the TT	■		Page 153
481	Measure/Inspect the tool length	■		Page 154
482	Measure/Inspect the tool radius	■		Page 157
483	Measure/Inspect the tool length and the tool radius	■		Page 159

HEIDENHAIN

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

Technical support FAX +49 (8669) 32-1000
Measuring systems ☏ +49 (8669) 31-3104
E-Mail: service.ms-support@heidenhain.de
TNC support ☏ +49 (8669) 31-3101
E-Mail: service.nc-support@heidenhain.de
NC programming ☏ +49 (8669) 31-3103
E-Mail: service.nc-pgm@heidenhain.de
PLC programming ☏ +49 (8669) 31-3102
E-Mail: service.plc@heidenhain.de
Lathe controls ☏ +49 (8669) 31-3105
E-Mail: service.lathe-support@heidenhain.de

www.heidenhain.de

3-D Touch Probe Systems from HEIDENHAIN

help you to reduce non-cutting time:

For example in

- workpiece alignment
- datum setting
- workpiece measurement
- digitizing 3-D surfaces

with the workpiece touch probes
TS 220 with cable
TS 640 with infrared transmission

- tool measurement
- wear monitoring
- tool breakage monitoring

with the tool touch probe
TT 140

Ve 00
661 873-20 · SW03 · 3 · 7/2008 · H · Printed in Germany · Subject to change without notice