HEIDENHAIN

User's Manual
Touch Probe Cycles

iTNC 530

English (en)
11/2007

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
iTNC 530	340 490-04
iTNC 530 E	340 491-04
iTNC 530	340 492-04
iTNC 530 E	340 493-04
iTNC 530 programming station	340 494-04

The suffix E indicates the export version of the TNC. The export version of the TNC has the following limitations:

■ Simultaneous linear movement in up to 4 axes

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 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 iTNC 530. Please contact HEIDENHAIN if you need a copy of this User's Manual. ID 533 190-xx

> **smarT.NC user documentation:**
>
> The smarT.NC operating mode is described in a separate Pilot. Please contact HEIDENHAIN if you require a copy of this Pilot. ID 533 191-xx.

Software options

The iTNC 530 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, 29 and 39)

Feed rate in mm/min on rotary axes: **M116**

Tilting the machining plane (Cycle 19, **PLANE** function and 3-D ROT soft key in the Manual operating mode)

Circle in 3 axes with tilted working plane

Software option 2

Block processing time 0.5 ms instead of 3.6 ms

5-axis interpolation

Spline interpolation

3-D machining:

- **M114:** Automatic compensation of machine geometry when working with tilted axes
- **M128:** Maintaining the position of the tool tip when positioning with tilted axes (TCPM)
- **FUNCTION TCPM:** Maintaining the position of the tool tip when positioning with tilted axes (TCPM) in selectable modes
- **M144:** Compensating the machine's kinematics configuration for ACTUAL/NOMINAL positions at end of block
- Additional **finishing/roughing** and **tolerance parameters for rotary axes** in Cycle 32 (G62)
- **LN** blocks (3-D compensation)

DCM collision software option

Function which dynamically monitors areas defined by the machine manufacturer to prevent collisions.

Additional conversational language software option

Function for enabling the conversational languages Slovenian, Slovak, Norwegian, Latvian, Estonian, Korean.

DXF converter software option

Extract contours from DXF files (R12 format).

Global Program Settings software option

Function for superimposing coordinate transformations in the Program Run modes.

AFC software option

Function for adaptive feed-rate control for optimizing the machining conditions during series production.

KinematicsOpt software option

Touch-probe cycles for inspecting and optimizing the machine accuracy.

TNC Model, Software and Features

i

Feature content level (upgrade functions)

Along with software options, significant further improvements of the TNC software are managed via the Feature Content Level **(FCL)** upgrade functions. Functions subject to the FCL are not available simply by updating the software on your 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.

FCL 4 functions	Description
Graphical depiction of the protected space when DCM collision monitoring is active	User's Manual
Handwheel superimposition in stopped condition when DCM collision monitoring is active	User's Manual
3-D rotation (set-up compensation)	Machine manual

FCL 3 functions	Description
Touch probe cycle for 3-D probing	Page 152
Touch probe cycles for automatic datum setting using the center point of a slot/ridge	Page 70
Feed-rate reduction for the machining of contour pockets with the tool being in full contact with the workpiece	User's Manual
PLANE function: Entry of axis angle	User's Manual
User documentation as a context-sensitive help system	User's Manual
smarT.NC: Programming of smarT.NC and machining can be carried out simultaneously	User's Manual
smarT.NC: Contour pocket on point pattern	smarT.NC Pilot
smarT.NC: Preview of contour programs in the file manager	smarT.NC Pilot
smarT.NC: Positioning strategy for machining point patterns	smarT.NC Pilot

FCL 2 functions	Description
3-D line graphics	User's Manual
Virtual tool axis	User's Manual
USB support of block devices (memory sticks, hard disks, CD-ROM drives)	User's Manual
Filtering of externally created contours	User's Manual
Possibility of assigning different depths to each subcontour in the contour formula	User's Manual
DHCP dynamic IP-address management	User's Manual
Touch-probe cycle for global setting of touch-probe parameters	Page 156
smarT.NC: Graphic support of block scan	smarT.NC Pilot
smarT.NC: Coordinate transformation	smarT.NC Pilot
smarT.NC: PLANE function	smarT.NC Pilot

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 of software 340 49x-02

■ New machine parameter for defining the positioning speed (see "Touch trigger probe, rapid traverse for positioning: MP6151" on page 25)

■ New machine parameter for consideration of basic rotation in Manual Operation (see "Consider a basic rotation in the Manual Operation mode: MP6166" on page 24)

■ Cycles 420 to 431 for automatic tool measurement were improved so that the measuring log can now also be displayed on the screen (see "Recording the results of measurement" on page 109)

■ A new cycle that enables you to set global touch probe parameters was introduced (see "FAST PROBING (touch probe cycle 441, DIN/ISO: G441, FCL 2 function)" on page 156)

New functions of software 340 49x-03

■ New cycle for setting a datum in the center of a slot (see "SLOT CENTER REFERENCE POINT (touch probe cycle 408, DIN/ISO: G408, FCL 3 function)" on page 70)

■ New cycle for setting a datum in the center of a ridge (see "SLOT CENTER REFERENCE POINT (touch probe cycle 409, DIN/ISO: G409, FCL 3 function)" on page 73)

■ New 3-D probing cycle (see "MEASURING IN 3-D (touch probe cycle 4, FCL 3 function)" on page 152)

■ Cycle 401 now also allows you to compensate workpiece misalignment by rotating the rotary table (see "BASIC ROTATION from two holes (touch probe cycle 401, DIN/ISO: G401)" on page 52)

■ Cycle 402 now also allows you to compensate workpiece misalignment by rotating the rotary table (see "BASIC ROTATION over two studs (touch probe cycle 402, DIN/ISO: G402)" on page 55)

■ In the cycles for datum setting, the results of measurement are available in the Q parameters **Q15X** (see "Measurement results in Q parameters" on page 69)

New functions of software 340 49x-04

- New cycle for saving a machine's kinematic configuration (see "SAVE KINEMATICS (touch probe cycle 450, DIN/ISO: G450, option)" on page 160)
- New cycle for testing and optimizing a machine's kinematic configuration (see "MEASURE KINEMATICS (touch probe cycle 451, DIN/ISO: G451, option)" on page 162)
- Cycle 412: Number of measuring points selectable through parameter Q423 (see "DATUM FROM INSIDE OF CIRCLE (touch probe cycle 412, DIN/ISO: G412)" on page 82)
- Cycle 413: Number of measuring points selectable through parameter Q423 (see "DATUM FROM OUTSIDE OF CIRCLE (touch probe cycle 413, DIN/ISO: G413)" on page 85)
- Cycle 421: Number of measuring points selectable through parameter Q423 (see "MEASURE HOLE (touch probe cycle 421, DIN/ISO: G421)" on page 118)
- Cycle 422: Number of measuring points selectable through parameter Q423 (see "MEAS. CIRCLE OUTSIDE (touch probe cycle 422, DIN/ISO: G422)" on page 121)
- Cycle 3: Error message can be suppressed if the stylus is already deflected when a cycle starts (see "MEASURING (touch probe cycle 3)" on page 150)

Functions changed since the predecessor versions 340 422-xx/ 340 423-xx

■ The management of more than one block of calibration data was changed (see "Managing more than one block of calibrating data" on page 34)

Contents

Index

1 Working with Touch Probe Cycles 19

4 Touch Probe Cycles for Automatic Kinematic Measurement 157

5 Touch Probe Cycles for Automatic Tool Measurement 173

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.

> If you are carrying out measurements during program run, be sure that the tool data (length, radius) can be used from the calibrated data or from the last TOOL CALL block (selected with MP7411).

Method of function

Whenever the TNC runs a touch probe cycle, the 3-D touch probe approaches the workpiece in one linear axis. This is also true during an active basic rotation or with a tilted working plane. The machine tool builder determines the probing feed rate 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 distance defined in MP 6130, the TNC displays an error message.
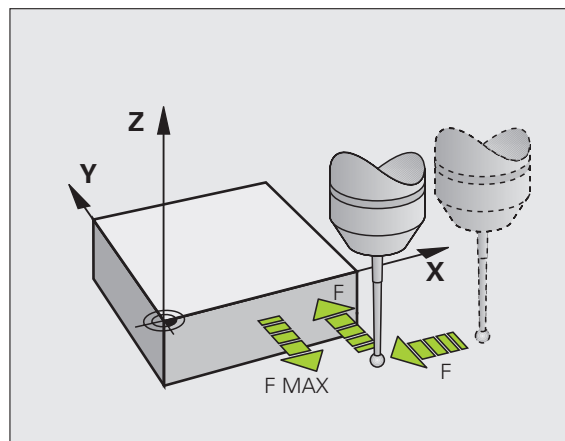
Touch probe cycles in the Manual and Electronic Handwheel modes

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

■ Set reference points

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 and Editing 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 (see figure at right).
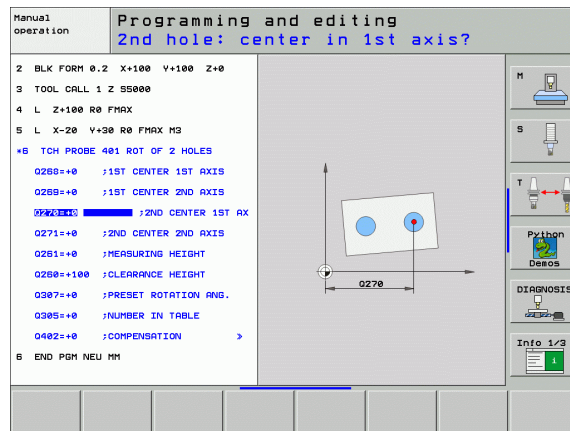
Defining the touch probe cycle in the Programming and Editing mode of operation

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

▶ Select the desired probe cycle, for example datum setting. Digitizing cycles and 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 48
Cycles for automatic workpiece presetting		Page 66
Cycles for automatic workpiece inspection		Page 108
Calibration cycles, special cycles	SPECIAL CYCLES	Page 147
Cycles for automatic tool measurement (enabled by the machine tool builder)		Page 174

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, machine parameters enable you to determine the behavior common to all touch probe cycles:

Maximum traverse to touch point: MP6130

If the stylus is not deflected within the path defined in MP6130, the TNC outputs an error message.

Safety clearance to touch point: MP6140

In MP6140 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 6140.

Orient the infrared touch probe to the programmed probe direction: MP6165

To increase measuring accuracy, you can use MP 6165 = 1 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 MP6165, you must recalibrate the touch probe.
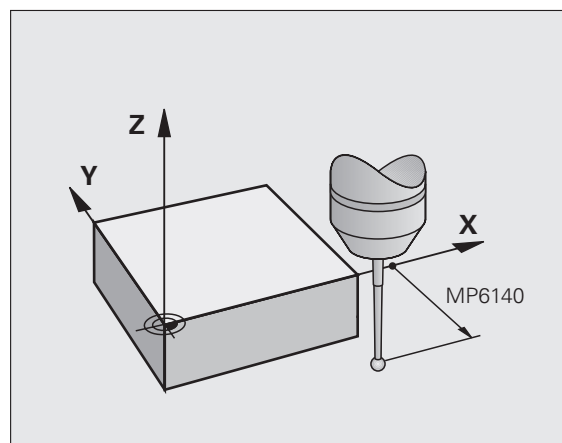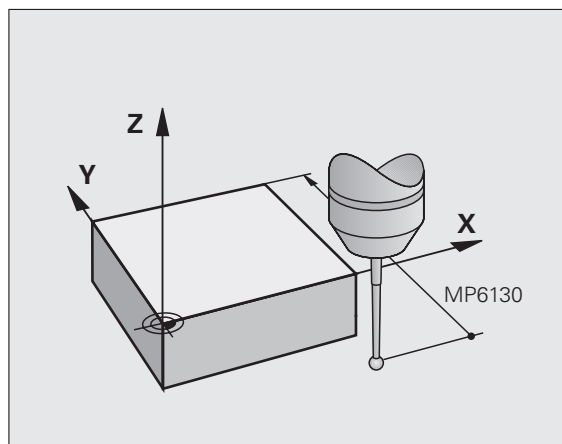
Consider a basic rotation in the Manual Operation mode: MP6166

Set MP 6166 = 1 for the TNC to consider an active basic rotation during the probing process (the workpiece is approached along an angular path if required) to ensure that the measuring accuracy for probing individual positions is also increased in Setup mode.

☞ This feature is not active during the following functions in the Manual Operation mode:

  - Calibrate length
  - Calibrate radius
  - Measure basic rotation

Multiple measurement: MP6170

To increase measuring certainty, the TNC can run each probing process up to three times in sequence. If the measured position values differ too greatly, the TNC outputs an error message (the limit value is defined in MP6171). 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 interval for multiple measurement: MP6171

In MP6171 you store the value by which the results may differ when you make multiple measurements. If the difference in the measured values exceeds the value in MP6171, the TNC outputs an error message.

Touch trigger probe, probing feed rate: MP6120

In MP6120 you define the feed rate at which the TNC is to probe the workpiece.

Touch trigger probe, rapid traverse for positioning: MP6150

In MP6150 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: MP6151

In MP6151 you define whether the TNC is to position the touch probe at the feed rate defined in MP6150 or at rapid traverse.

- Input value = 0: Position at feed rate from MP6150
- Input value = 1: Pre-position at rapid traverse

KinematicsOpt: Tolerance limit in Optimization mode: MP6600

In **MP6600** you define the tolerance limit starting from which the TNC displays a note in the Optimizing mode when the measured kinematic data is greater than this limit value. The default value is 0.05. The larger the machine, the greater these values should be.

- Input range: 0.001 to 0.999

KinematicsOpt, permissible deviation of the calibration ball radius: MP6601

In **MP6601** you define the maximum permissible deviation from the entered cycle parameter by the calibration ball radius measured in the cycles.

- Input range: 0.01 to 0.1

The TNC calculates the calibration ball radius twice at every measuring point for all 5 touch points. If the radius is greater than Q407 + MP6601 an error message appears because it could be contamination.

If the radius found by the TNC is less than 5 * (Q407 - MP6601), the TNC also issues an error message.
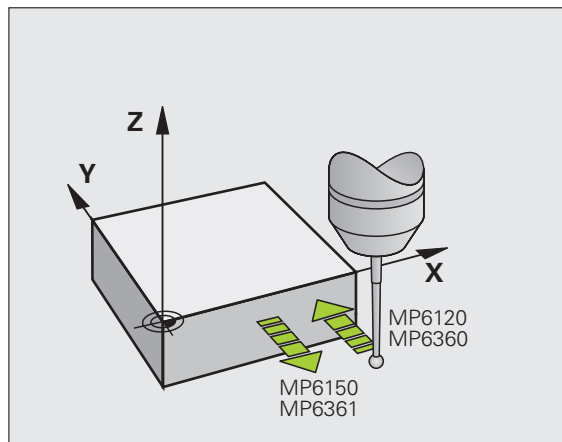
Running touch probe cycles

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.

☞ Make sure that at the beginning of the cycle the compensation data (length, radius) from the calibrated data or from the last TOOL CALL block are active (selection via MP7411, see the User's Manual of the iTNC 530, "General User Parameters").

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.

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	CAL L	Page 32
Calibrate the effective radius	PROBING	Page 33
Measure a basic rotation using a line	ROTATION	Page 35
Set the reference point in any axis	PROBING POS	Page 37
Set a corner as datum	PROBING P	Page 38
Set a circle center as datum	PROBING CC	Page 39
Set a center line as datum	PROBING	Page 40
Measure a basic rotation using two holes/cylindrical studs	PROBING ROT	Page 41
Set the datum using four holes/cylindrical studs	PROBING P	Page 41
Set a circle center using three holes/cylindrical studs	PROBING CC	Page 41

Selecting probe cycles

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

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

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

Recording measured values from the touch probe cycles

> The TNC must be specially prepared by the machine tool builder for use of this function. The machine tool manual provides further information.

After executing any selected probe cycle, the TNC displays the soft key PRINT. If you press this soft key, the TNC will record the current values determined in the active touch probe cycle. You can then use the PRINT function in the menu for setting the data interface (see the User's Manual Chapter 12, "MOD Functions, Setting the Data Interfaces") to define whether the TNC is to

- print the measuring result,
- store the measuring results on the TNC's hard disk, or
- store the measuring results on a PC.

If you store the measuring results, the TNC creates the ASCII file %TCHPRNT.A. Unless you define a specific path and interface in the interface configuration menu, the TNC will store the %TCHPRNT file in the main directory TNC:\.

> When you press the PRINT soft key, the %TCHPRNT.A file must not be active in the **Programming and Editing** mode of operation. The TNC will otherwise display an error message.
>
> The TNC stores the measured data in the %TCHPRNT.A file only. If you execute several touch probe cycles in succession and want to store the resulting measured data, you must make a backup of the contents stored in %TCHPRNT.A between the individual cycles by copying or renaming the file.
>
> Format and contents of the %TCHPRNT file are preset by the machine tool builder.
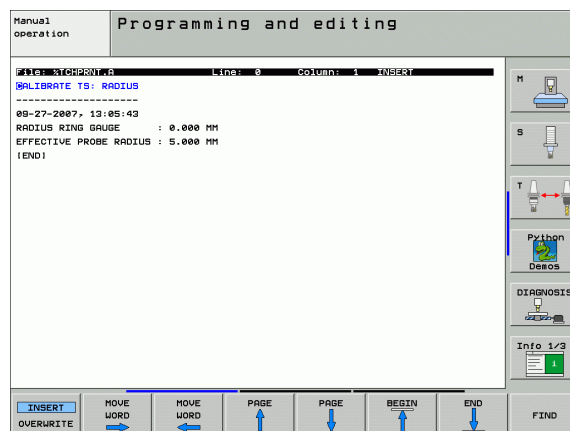
Writing the measured values from touch probe cycles in datum tables

☞ This function is active only if you have datum tables active on your TNC (bit 3 in Machine Parameter 7224.0 =0).

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 31).

With the ENTER IN DATUM TABLE soft key, the TNC can write the values measured during a touch probe cycle in a datum table:

🖑 Note that during an active datum shift the TNC always bases the probed value on the active preset (or on the reference point most recently set in the Manual operating mode), although the datum shift is included in the position display.

▶ 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 30).

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:\.

☝ Note that during an active datum shift the TNC always bases the probed value on the active preset (or on the reference point most recently set in the Manual operating mode), although the datum shift is included in the position display.

▶ 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.

☝ If you overwrite the active datum, the TNC shows a warning. If you really want to overwrite it, press the ENT key. If not, press the NO ENT key.

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

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.

CAL   L

▶ 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 boxes.

▶ Enter the tool axis (with the axis key).

▶ Datum: Enter the height of the ring gauge.

▶ The menu items 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.
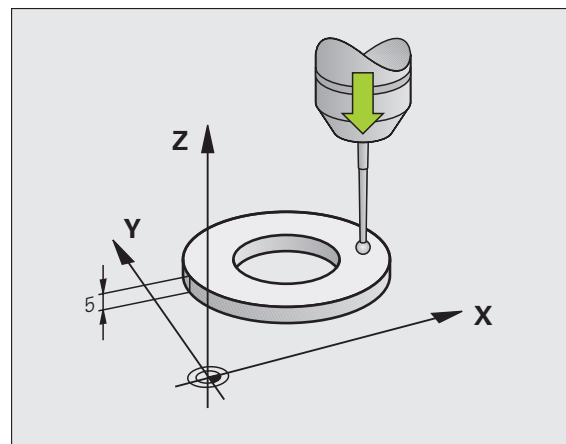
Calibrating the effective radius and compensating center misalignment

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 calibration routine varies depending on the setting of Machine Parameter 6165 (spindle orientation active/inactive), (see "Orient the infrared touch probe to the programmed probe direction: MP6165" on page 23). If the function for orienting the infrared touch probe to the programmed probe direction is active, the calibration cycle is executed after you have pressed NC Start once. If the function is not active, you can decide whether you want to compensate the center misalignment by calibrating the effective radius.

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 6160.

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.
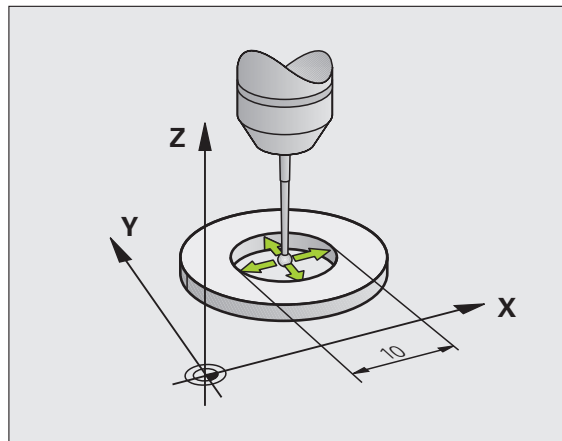
Displaying calibration values

The TNC stores the effective length and radius, as well as the center misalignment, for use when the touch probe is needed again. You can display the values on the screen with the soft keys CAL. L and CAL. R.

☞ If you want to use several touch probes or calibration data blocks: See "Managing more than one block of calibrating data," page 34.

Managing more than one block of calibrating data

If you use several touch probes or measuring contacts arranged in a cross shape on your machine, you must also use several blocks of calibration data.

To be able to use more than one block of calibration data, you must set Machine Parameter 7411=1. To find the calibration data, proceed in the same way as is done with one single touch probe. When exiting the Calibration menu, press the ENT key to confirm the entry of the calibration data in the tool table and for the TNC to save the calibration data in the tool table. The line of the tool table, to which the TNC saves the data, is determined by the active tool number.

☞ 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.

```
Manual operation                        Programming
                                        and editing




Radius ring gauge =              0
Effective probe radius =        +5
Styl.tip center offset    X=+0
Styl.tip center offset    Y=+0


                    0% S-IST
                    0% S[Nm]  LIMIT 1  13:05
X    +179.522  Y    +164.718  Z    +152.834
+a    +0.000 +A     +0.000 +B      +0.000
+C    +0.000
                              S1   0.000
ACTL.     :15      T 5       Z S 2500   F 0       M 5 / 9

  X+      X-      Y+      Y-                PRINT   END
```
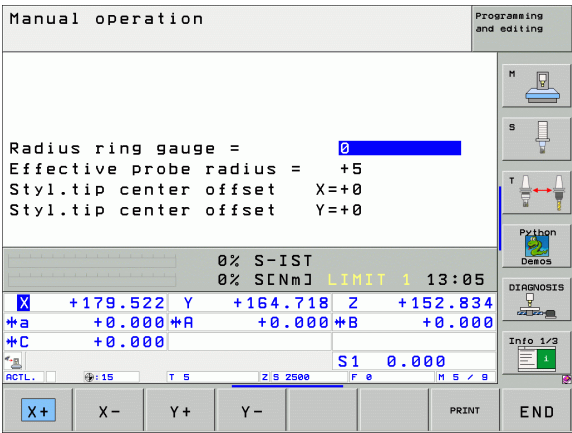
2.3 Compensating Workpiece Misalignment

Introduction

The TNC electronically 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.

☞ 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.

You can also use a basic rotation in conjunction with the PLANE function. In this case, first activate the basic rotation and then the PLANE function.

If you change the basic rotation, the TNC asks you if you also want to save the changed basic rotation in the active line of the Preset table when you exit the menu. In this case, confirm with the ENT key.
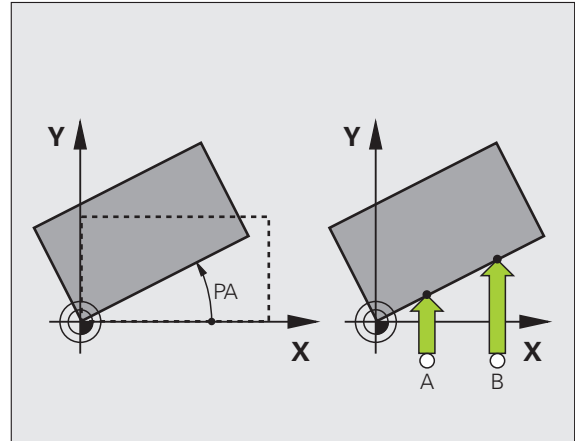
If your machine has been prepared for it, the TNC can also conduct a real, three-dimensional set-up compensation. If necessary, contact your machine tool builder.

Measuring the basic rotation

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 =**

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 a rotation angle of zero and confirm with the ENT key.

▶ Terminate the probe function by pressing the END key.
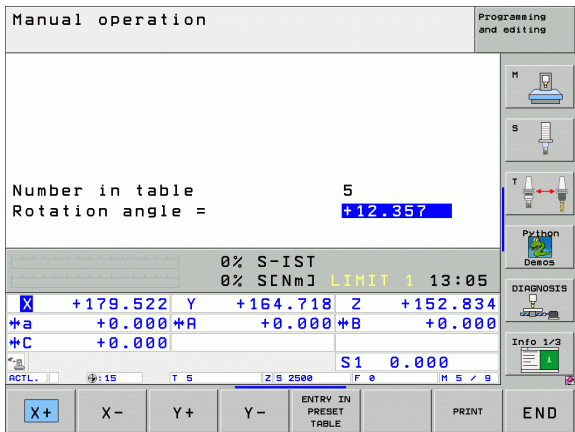
2.4 Setting the Datum with a 3-D Touch Probe

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
- Setting a center line as the datum with PROBING

☞ Note that during an active datum shift the TNC always bases the probed value on the active preset (or on the reference point most recently set in the Manual operating mode), although the datum shift is included in the position display.

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 30, or see "Writing the measured values from touch probe cycles in the preset table," page 31).

▶ To terminate the probe function, press the END key.
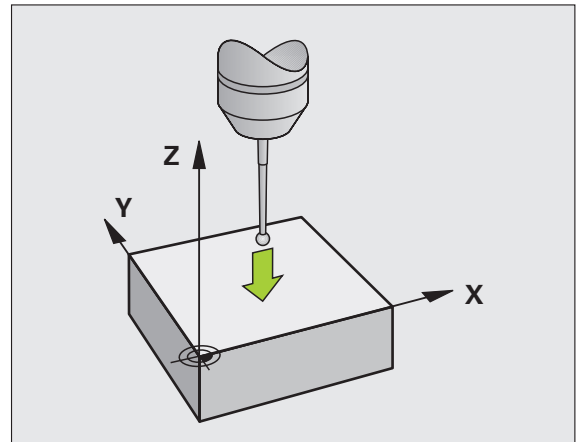
Corner as datum—using points that were already probed for a basic rotation

▶ Select the probe function by pressing the PROBING P soft key.

▶ **Touch points of basic rotation ?**: Press ENT to transfer the touch point coordinates.

▶ Position the touch probe at a starting position near the first touch point of the side that was not probed for basic rotation.

▶ 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 30, or see "Writing the measured values from touch probe cycles in the preset table," page 31).

▶ To terminate the probe function, press the END key.
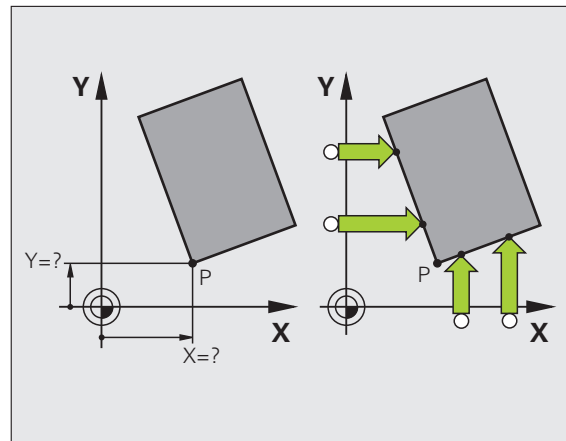
Corner as datum—without using points that were already probed for a basic rotation

▶ Select the probe function by pressing the PROBING P soft key.

▶ **Touch points of basic rotation?**: Press NO ENT to ignore the previous touch points. (The dialog question only appears if a basic rotation was made previously.)

▶ Probe both workpiece sides twice.

▶ **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 30, or see "Writing the measured values from touch probe cycles in the preset table," page 31).

▶ 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.

    ▶ If you are probing to find the stylus center (only available on machines with spindle orientation, depending on MP6160), press the 180° soft key and probe another four points on the inside of the circle.

    ▶ If you are not probing to find the stylus center, press the END key.

    ▶ **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 30, or see "Writing the measured values from touch probe cycles in the preset table," page 31).

    ▶ To terminate the probe function, press the END key.
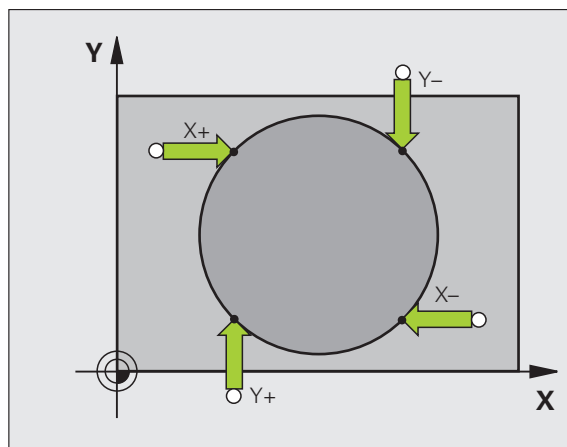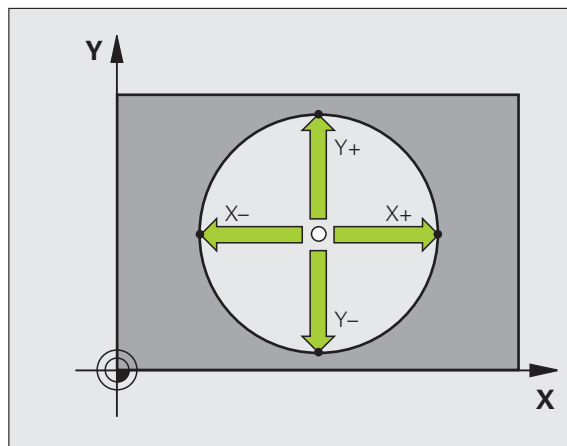
Outside circle

▶ 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 30, or see "Writing the measured values from touch probe cycles in the preset table," page 31).
▶ 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.

Center line as datum

▶ Select the probe function by pressing the PROBING soft key.

▶ Position the ball tip at a starting position near the first touch point.

▶ Select the probing direction 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.

▶ **Datum:** Enter the coordinate of the datum in the menu window, confirm with the SET DATUM soft key, or write the value to a table (see "Writing the measured values from touch probe cycles in datum tables," page 30, or see "Writing the measured values from touch probe cycles in the preset table," page 31).

▶ To terminate the probe function, press the END key.
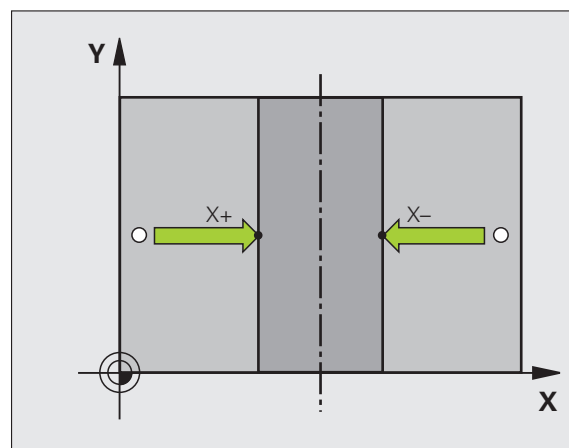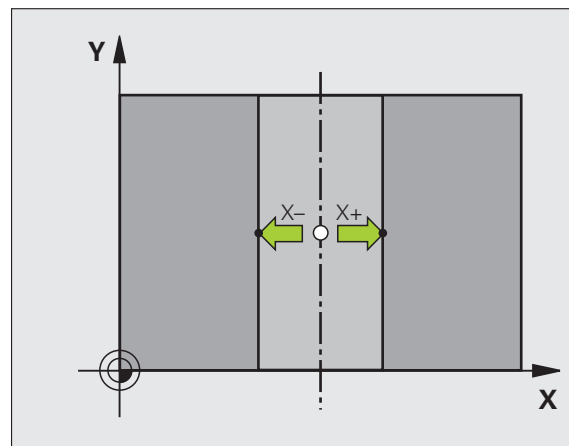
Setting datum points using holes/cylindrical studs

A second soft-key row provides soft keys for using holes or cylindrical studs to set a reference point.

Define whether a hole or stud is to be probed

The default setting is for probing holes.

▶ Select the probe function by pressing the TOUCH PROBE soft key, shift the soft-key row.

▶ Select the probe function: For example, press the PROBING ROT soft key.

▶ Circular studs are to be probed. Define by soft key.

▶ Holes are to be probed. Define by soft key.

Probing holes

Pre-position the touch probe approximately in the center of the hole. After you have pressed the external START key, the TNC automatically probes four points on the wall of the hole.

Move the touch probe to the next hole repeat the probing process. and have the TNC repeat the probing procedure until all the holes have been probed to set reference points.

Probing cylindrical studs

Position the ball tip at a starting position near the first touch point of the stud. Select the probing direction by soft key and press the machine START button to start probing. Perform the above procedure four times.

Overview

Cycle	Soft key
Basic rotation using 2 holes: The TNC measures the angle between the line connecting the centers of two holes and a nominal position (angle reference axis).	PROBING ROT
Datum using 4 holes: The TNC calculates the intersection of the line connecting the first two probed holes with the line connecting the last two probed holes. You need to probe diagonally opposite holes one after another (as shown on the soft key), as otherwise the datum calculated by the TNC will be incorrect.	PROBING P
Circle center using 3 holes: The TNC calculates a circle that intersects the centers of all three holes, and finds the center.	PROBING CC
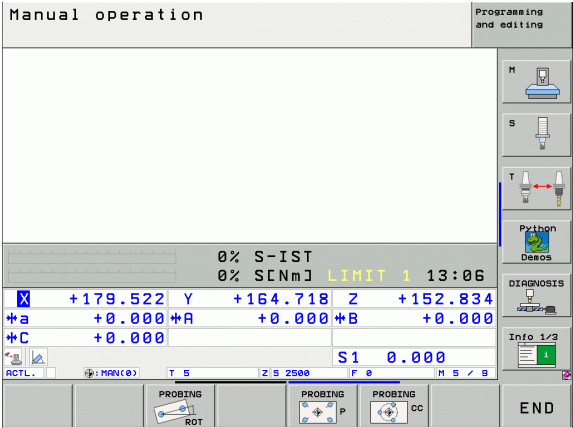
2.5 Measuring Workpieces with a 3-D Touch Probe

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 108). 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—without using points that were already probed for a basic rotation," page 38. 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°.
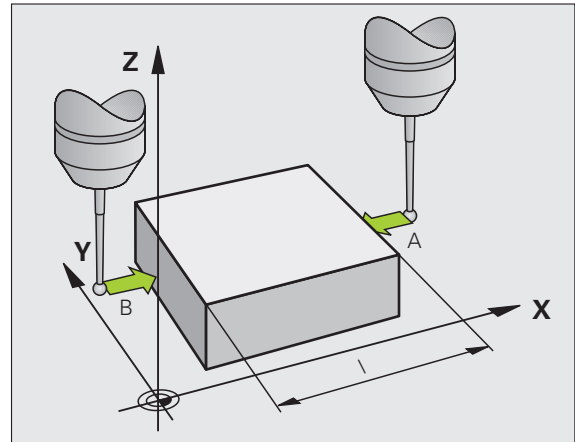
To find the angle between the angle reference axis and a side of the workpiece

ROTATION

▶ 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 35).

▶ 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.
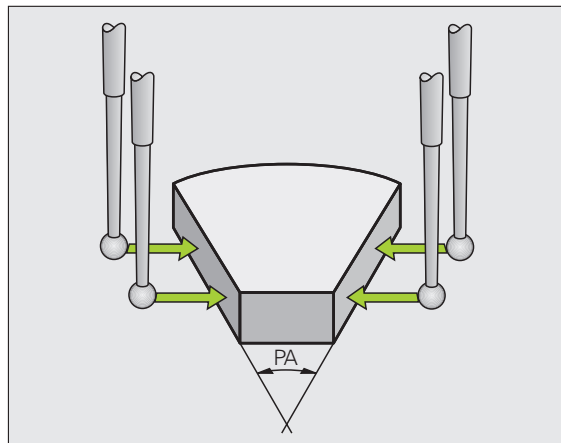
To measure the angle between two workpiece sides:

▶ 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 value that appears under Rotation angle.

▶ Make a basic rotation with the side of the workpiece (see "Compensating Workpiece Misalignment" on page 35).

▶ 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.
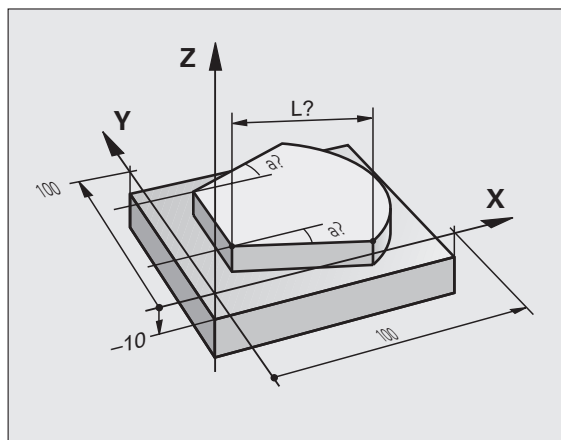
2.6 Using the Touch Probe Functions with Mechanical Probes or Dial Gauges

Introduction

If you do not have an electronic 3-D touch probe on your machine, you can also use all the previously described manual touch probe functions (exception: calibration function) with mechanical probes or by simply touching the workpiece with the tool.

In place of the electronic signal generated automatically by a 3-D touch probe during probing, you can manually initiate the trigger signal for capturing the **probing position** by pressing a key. Proceed as follows:

▶ Select any touch probe function by soft key.

▶ Move the mechanical probe to the first position to be captured by the TNC.

▶ Confirm the position: Press the actual-position-capture key for the TNC to save the current position.

▶ Move the mechanical probe to the next position to be captured by the TNC.

▶ Confirm the position: Press the actual-position-capture key for the TNC to save the current position.

▶ If required, move to additional positions and capture as described previously.

▶ **Datum:** In the menu window, enter the coordinates of the new datum, 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 30, or see "Writing the measured values from touch probe cycles in the preset table," page 31).

▶ To terminate the probe function, press the END key.

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 50
401 ROT OF 2 HOLES Automatic measurement using two holes. Compensation via basic rotation.		Page 52
402 ROT OF 2 STUDS Automatic measurement using two studs. Compensation via basic rotation.		Page 55
403 ROT IN ROTARY AXIS Automatic measurement using two points. Compensation by turning the table.		Page 58
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 62
404 SET BASIC ROTATION Setting any basic rotation.		Page 61

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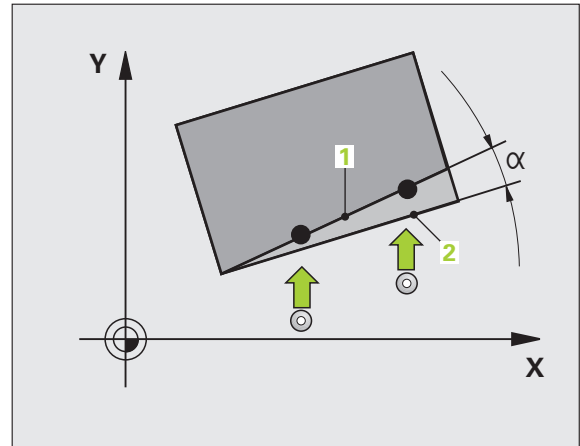
BASIC ROTATION (touch probe cycle 400, DIN/ISO: G400)

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 35).

1 The TNC positions the touch probe to the starting points at rapid traverse (value from MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 26) 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 (MP6120 or MP6360).

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.

3 Touch Probe Cycles for Automatic Workpiece Inspection

▶ **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 MP6140.

▶ **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 35). 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 26), the TNC positions the touch probe at rapid traverse (value from MP6150 or MP6361) 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

3 Touch Probe Cycles for Automatic Workpiece Inspection

![401 icon]

▶ **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 set 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 (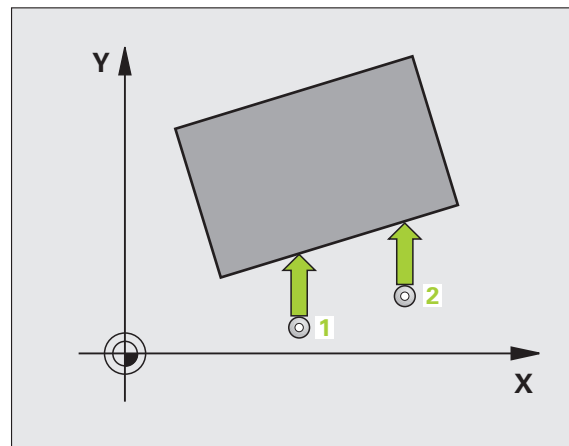See also \gCompensating Workpiece Misalignment" on page 35). As an alternative, you can also compensate the determined misalignment by rotating the rotary table.

1 Following the positioning logic (see "Running touch probe cycles" on page 26), the TNC positions the touch probe in rapid traverse (value from MP6150 or MP6361) to the starting point for probing the first stud **1**.

2 Then the probe moves to the entered `measuring height 1` and probes four points to find the center of the first stud. The touch probe moves on a circular arc between the touch points, each of which is offset by 90°.

3 The touch probe returns to the clearance height and then 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 MP6140.

▶ **Clearance height** Q260 (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.

3 Touch Probe Cycles for Automatic Workpiece Inspection

▶ **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 set 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 MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 26) 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 (MP6120 or MP6360).

3 Then the touch probe moves to the next starting position **2** and probes the second position.
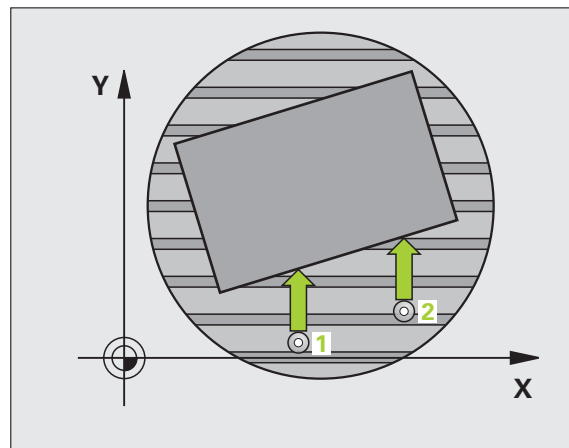
4 The TNC returns the touch probe to the clearance height and moves the rotary axis, which was defined in the cycle, by the measured value. Optionally you can have the display set to 0 after alignment.

☞ **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 MP6140.

▶ **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
```

3.1 Measuring Workpiece Misalignment

HEIDENHAIN iTNC 530

61

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 to the starting points at rapid traverse (value from MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 26) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120 or MP6360). 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.

![405 icon]

▶ **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 MP6140.

▶ **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.
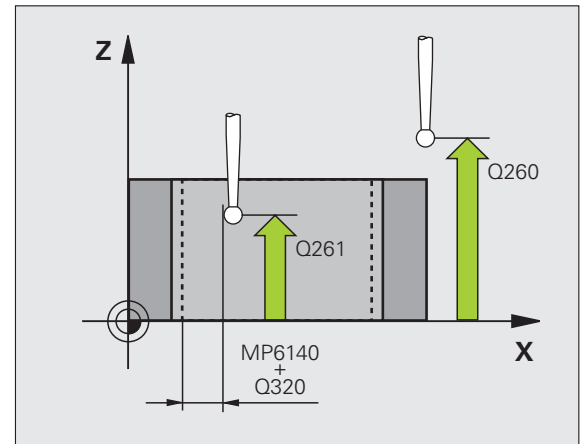
Example: NC blocks

5 TCH PROBE 405 ROT IN C AXIS	
Q321=+50	;CENTER IN 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
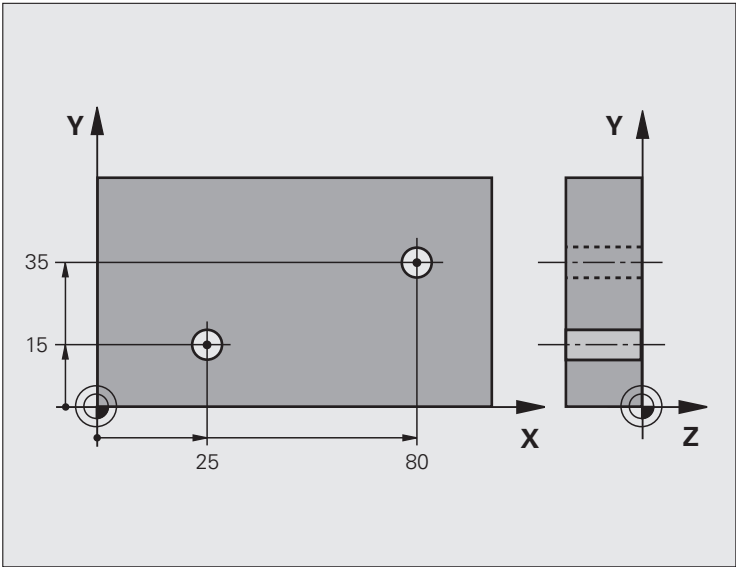
0 BEGIN PGM CYC401 MM	
1 TOOL CALL 0 Z	
2 TCH PROBE 401 ROT 2 HOLES	
Q268=+25 ;1ST CENTER IN 1ST AXIS	Center of the 1st hole: X coordinate
Q269=+15 ;1ST CENTER IN 2ND AXIS	Center of the 1st hole: Y coordinate
Q270=+80 ;2ND CENTER IN 1ST AXIS	Center of the 2nd hole: X coordinate
Q271=+35 ;2ND CENTER IN 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	408	Page 70
409 RIDGE CENTER REF PT. Measuring the outside width of a ridge, and defining the ridge center as datum	409	Page 73
410 DATUM INSIDE RECTAN. Measuring the inside length and width of a rectangle, and defining the center as datum	410	Page 76
411 DATUM OUTSIDE RECTAN. Measuring the outside length and width of a rectangle, and defining the center as datum	411	Page 79
412 DATUM INSIDE CIRCLE Measuring any four points on the inside of a circle, and defining the center as datum	412	Page 82
413 DATUM OUTSIDE CIRCLE Measuring any four points on the outside of a circle, and defining the center as datum	413	Page 85
414 DATUM OUTSIDE CORNER Measuring two lines from the outside of the angle, and defining the intersection as datum	414	Page 88
415 DATUM INSIDE CORNER Measuring two lines from within the angle, and defining the intersection as datum	415	Page 91
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	416	Page 94

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	417	Page 97
418 DATUM FROM 4 HOLES (2nd soft-key level) Measuring 4 holes crosswise and defining the intersection of the lines between them as datum	418	Page 99
419 DATUM IN ONE AXIS (2nd soft-key level) Measuring any position in any axis and defining it as datum	419	Page 102

3.2 Presetting automatically

Characteristics common to all touch probe cycles for datum setting

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

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 or W	X and Y
Y or V	Z and X
X or U	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.

SLOT CENTER REFERENCE POINT (touch probe cycle 408, DIN/ISO: G408, FCL 3 function)

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 to the starting points at rapid traverse (value from MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 26) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120 or MP6360).

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 69) 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.
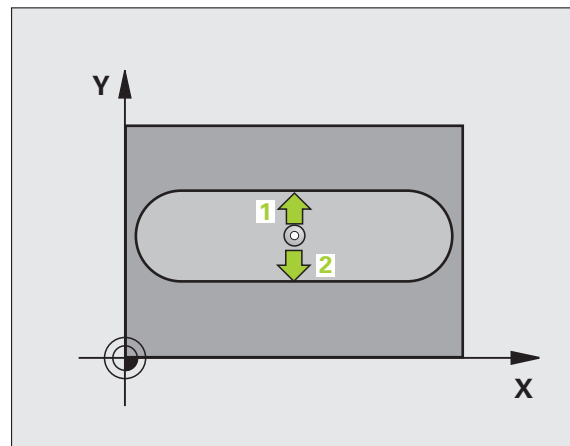
▶ **Center in 1st axis** Q321 (absolute value): Center of the slot in the reference axis of the working plane.

▶ **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 MP6140.

▶ **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. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, 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=+1       ;DATUM
```

SLOT CENTER REFERENCE POINT (touch probe cycle 409, DIN/ISO: G409, FCL 3 function)

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 to the starting points at rapid traverse (value from MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 26) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120 or MP6360).

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 69) 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.

3.2 Presetting automatically
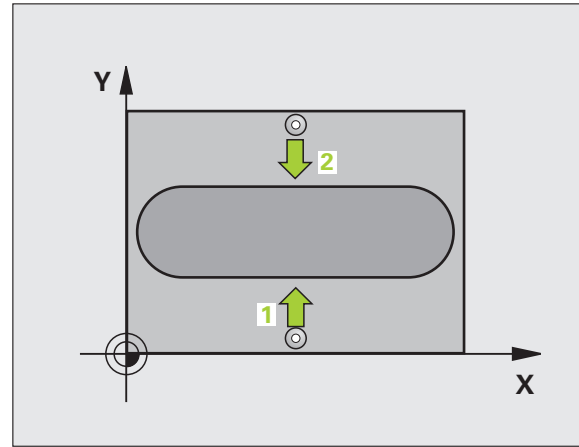
▶ **Center in 1st axis** Q321 (absolute): Center of the ridge in the reference axis of the working plane.

▶ **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 MP6140.

▶ **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. 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. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the reference point 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=+1	;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 to the starting points at rapid traverse (value from MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 26) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120 or MP6360).

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 69)

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.
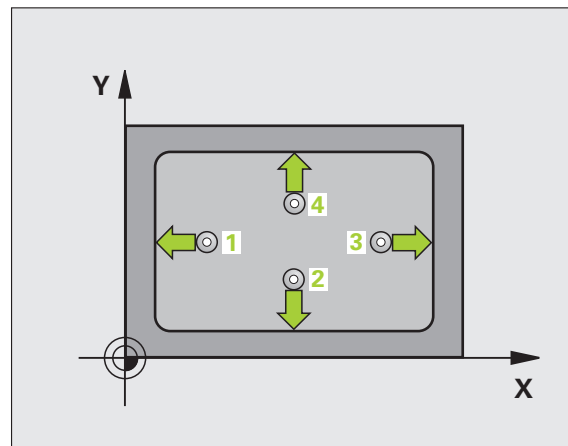
▶ **Center in 1st axis** Q321 (absolute value): Center of the pocket in the reference axis of the working plane.

▶ **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 MP6140.

▶ **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 69).
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. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the reference point 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=+1	;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 to the starting points at rapid traverse (value from MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 26) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120 or MP6360).

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 69)

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.
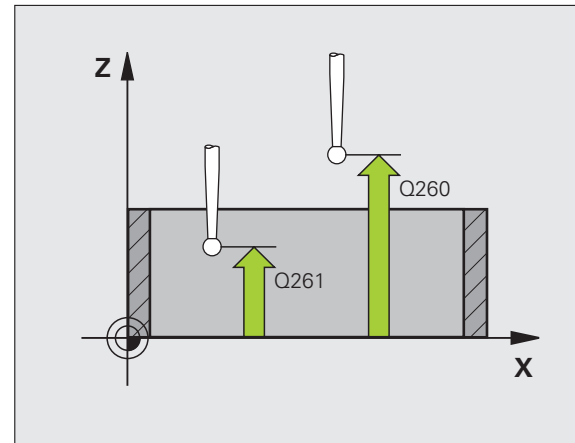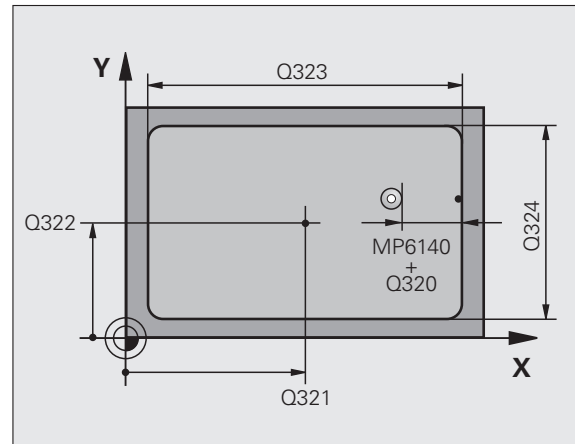
Before a cycle definition you must have programmed a tool call to define the touch probe axis.
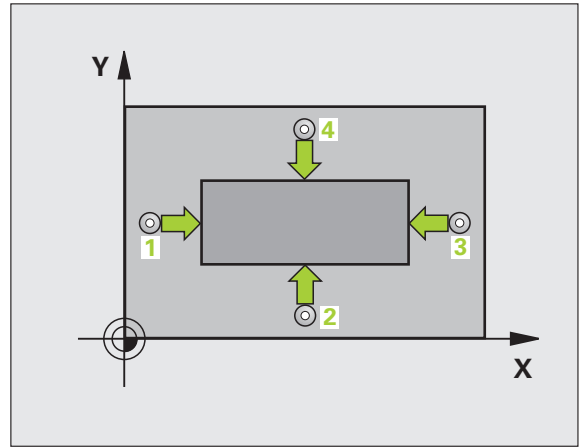
▶ **Center in 1st axis** Q321 (absolute value): Center of the stud in the reference axis of the working plane.

▶ **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 MP6140.

▶ **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

3 Touch Probe Cycles for Automatic Workpiece Inspection

▶ **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 69).
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. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the reference point 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=+1	;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 to the starting points at rapid traverse (value from MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 26) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120 or MP6360). 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 69) 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 (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 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 MP6140.

▶ **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 or 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 on the pocket center.

▶ **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 69).
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. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the reference point 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
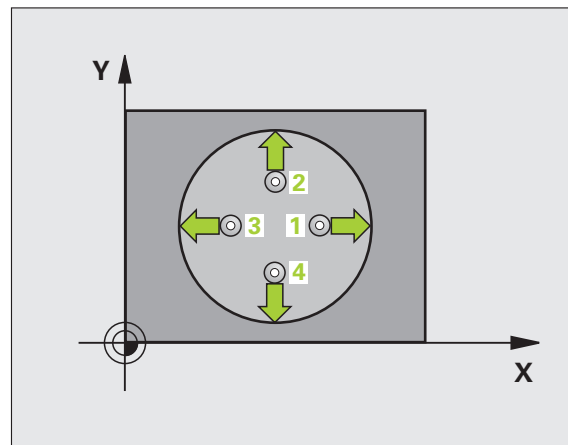
▶ **No. of measuring points (4/3)** Q423: Specify whether the TNC should measure the hole with 4 or 3 probing points:
4: Use 4 measuring points (standard setting)
3: Use 3 measuring points (standard setting)

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=+1	;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 to the starting points at rapid traverse (value from MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 26) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120 or MP6360). 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 69) 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 **high** estimate for the nominal diameter of the pocket (or hole).
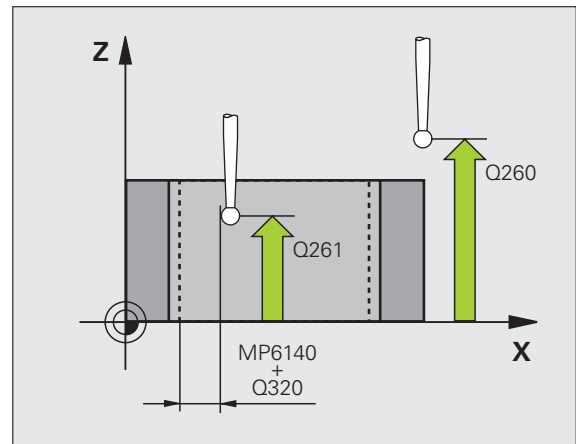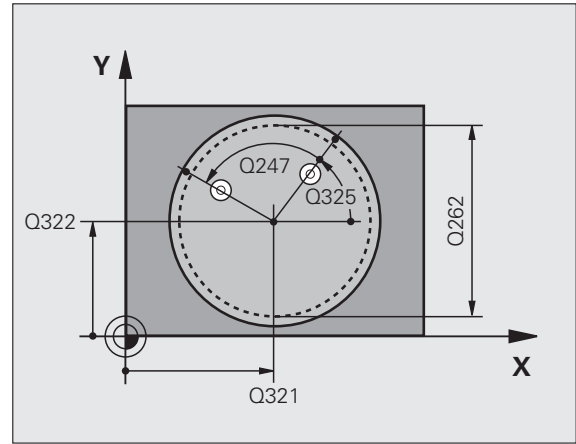
Before a cycle definition you must have programmed a tool call to define the touch probe axis.

▶ **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 (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 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 MP6140.

▶ **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 69).
  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. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the reference point 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)
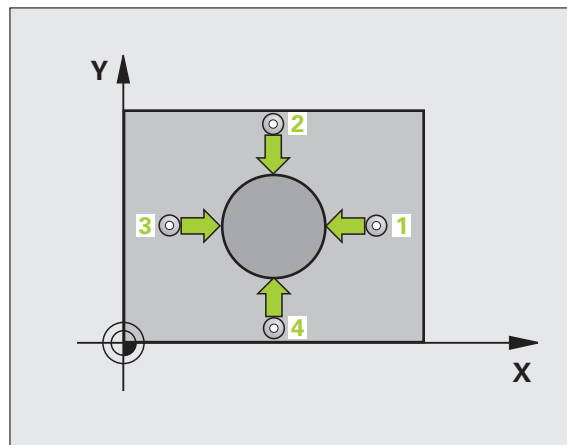 **3:** Use 3 measuring points (standard setting)

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=+1	;DATUM
Q423=4	;NO. OF MEAS. POINTS

3.2 Presetting automatically

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 26), the TNC positions the touch probe at rapid traverse (value from MP6150 or MP6361) 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 probes the first touch point at the probing feed rate (MP6120 or MP6360). 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 69) 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.
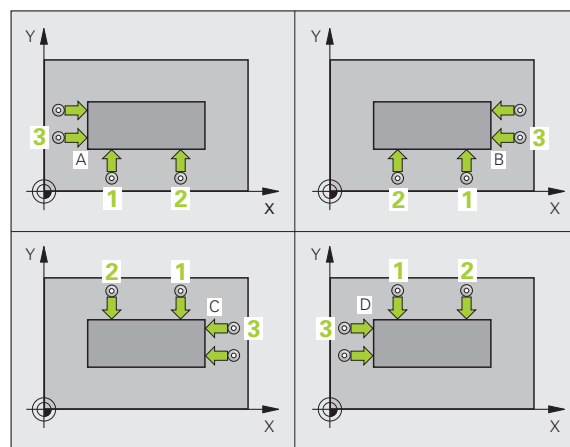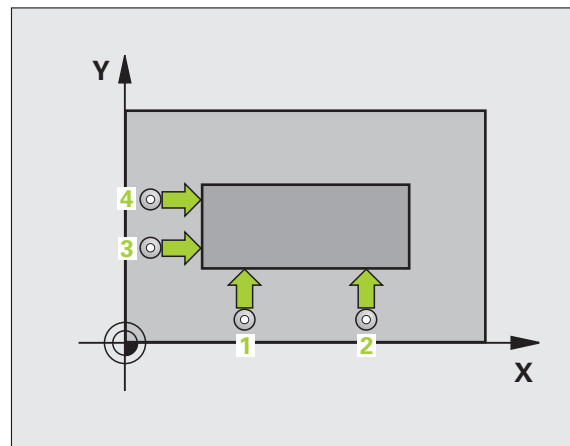
Corner	X coordinate	Y coordinate
A	Point **1** greater than point **3**	Point **1** less than point **3**
B	Point **1** less than point **3**	Point **1** less than point **3**
C	Point **1** less than point **3**	Point **1** greater than point **3**
D	Point **1** greater than point **3**	Point **1** greater than point **3**

▶ **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 MP6140.

▶ **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 69).
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. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the reference point 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=+1	;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 26), the TNC positions the touch probe at rapid traverse (value from MP6150 or MP6361) 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 probes the first touch point at the probing feed rate (MP6120 or MP6360). 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 69) 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.

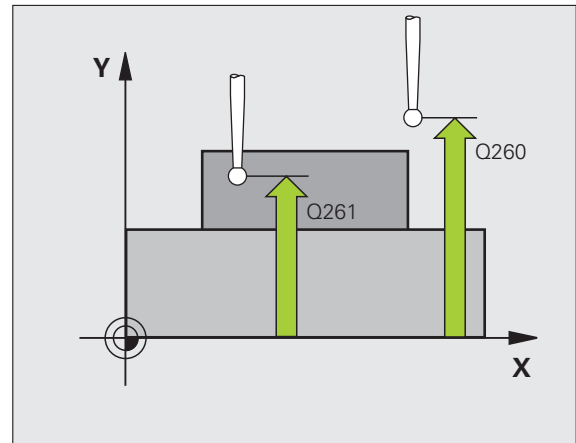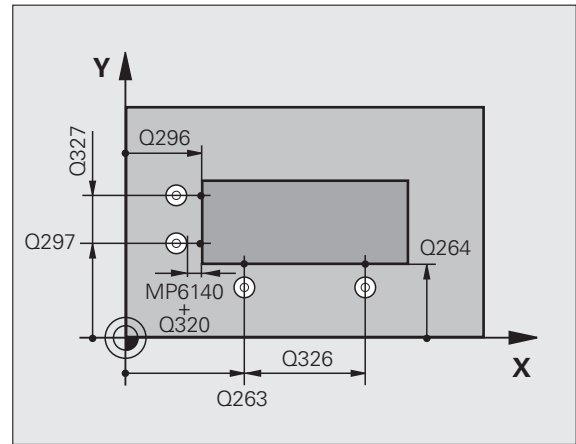| Parameter number	Meaning
Q151	Actual value of corner in reference axis
Q152	Actual value of corner in minor axis

☞   **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 MP6140.

▶ **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

3 Touch Probe Cycles for Automatic Workpiece Inspection

▶ **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 69).
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. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the reference point 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=+1	;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 Following the positioning logic (see "Running touch probe cycles" on page 26), the TNC positions the touch probe at rapid traverse (value from MP6150 or MP6361) 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 69) 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.

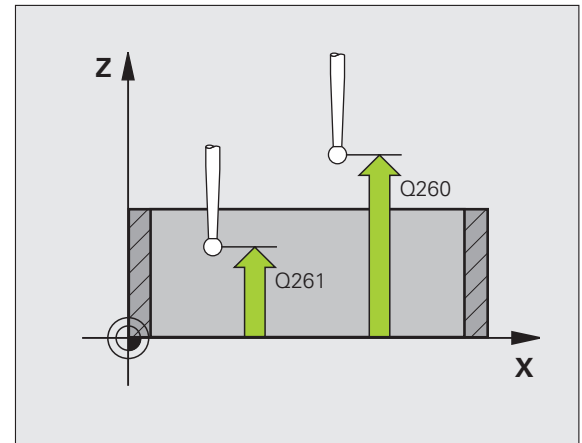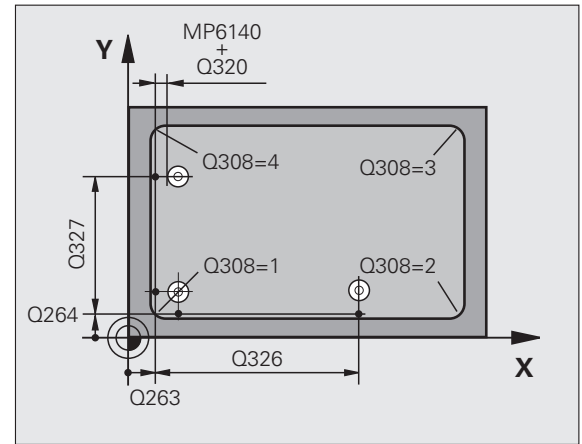| Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of bolt hole circle diameter

☞ **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 69).
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. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the reference point 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=+1	;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 MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 26) 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 69) and saves the actual values in the Q parameters listed below

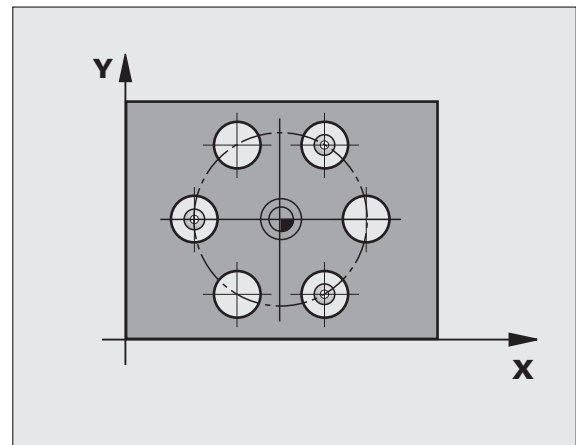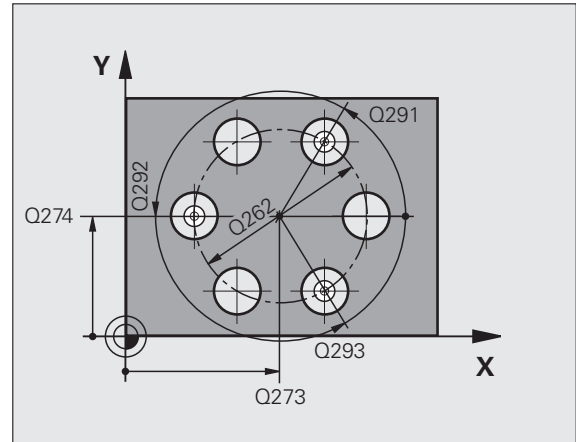| Parameter number	Meaning
Q160	Actual value of measured point

☞ **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 MP6140.

▶ **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 69).
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 Following the positioning logic (see "Running touch probe cycles" on page 26), the TNC positions the touch probe at rapid traverse (value from MP6150 or MP6361) 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 69). 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.
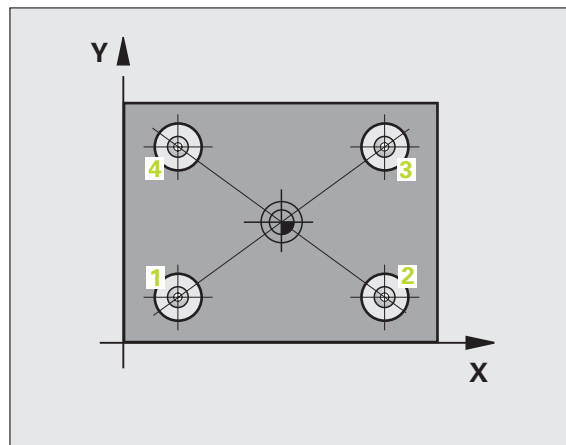
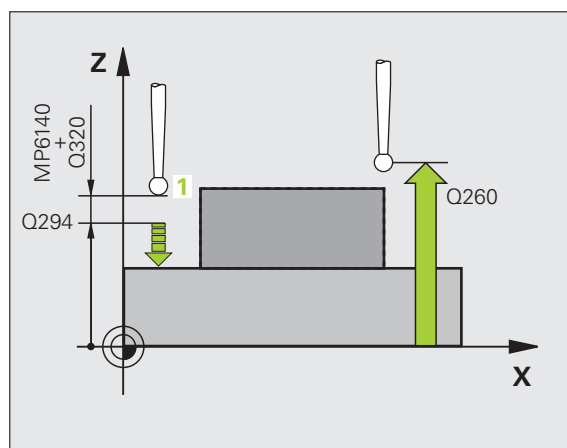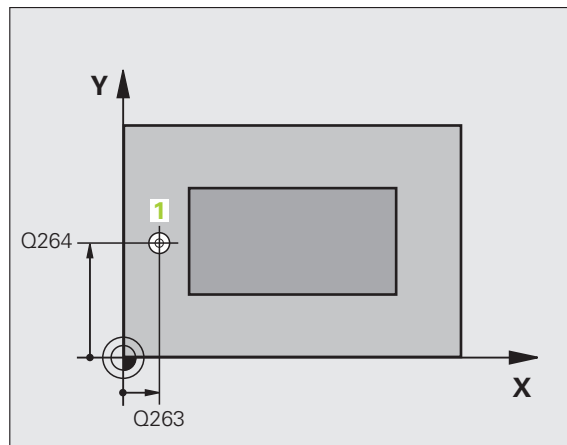| Parameter number	Meaning
Q151	Actual value of intersection point in reference axis
Q152	Actual value of intersection point in minor axis

☞ **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.

3 Touch Probe Cycles for Automatic Workpiece Inspection
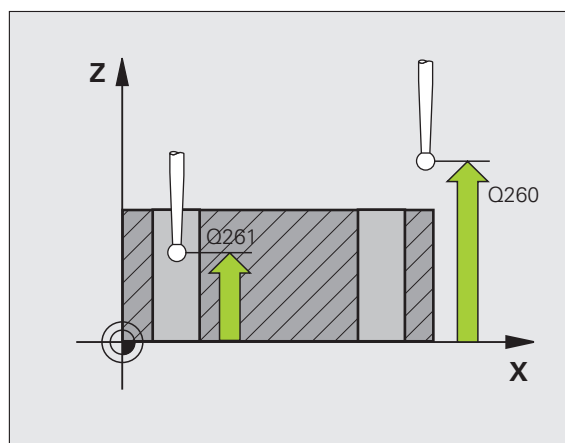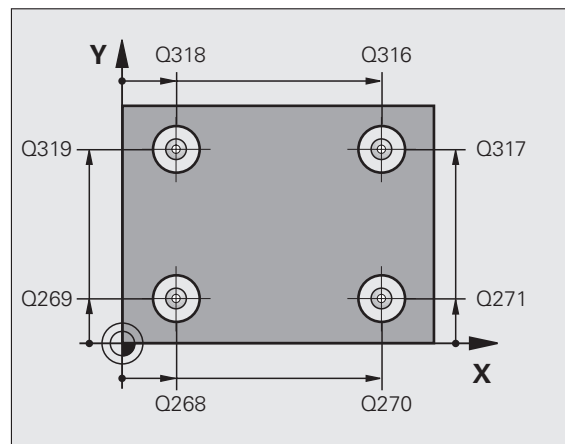
▶ **Datum number in table** Q305: Enter the number in the datum or preset table in which the TNC is to save the coordinates of the line intersection. If you enter Q305=0, the TNC automatically sets the display so that the new datum is at the intersection of the connecting lines.

▶ **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 69).
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. 3rd axis** Q384 (absolute): Coordinate of the probe point in the touch probe axis, at which point the reference point 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 MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 26) 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 69)

☞ **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 MP6140.

▶ **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 69
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

3.2 Presetting automatically

Example: Datum setting in center of a circular segment and on top surface of workpiece
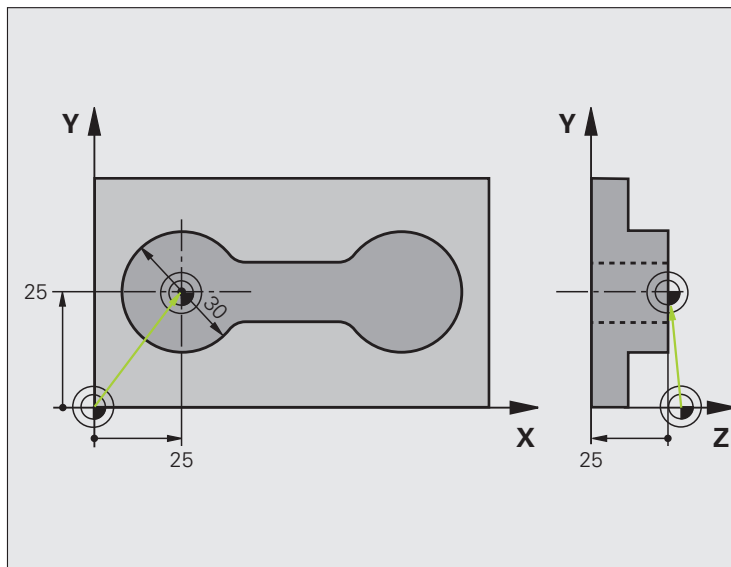
0 BEGIN PGM CYC413 MM	
1 TOOL CALL 0 Z	Call tool 0 to define the touch probe axis

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 MP6140
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 35K47		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.
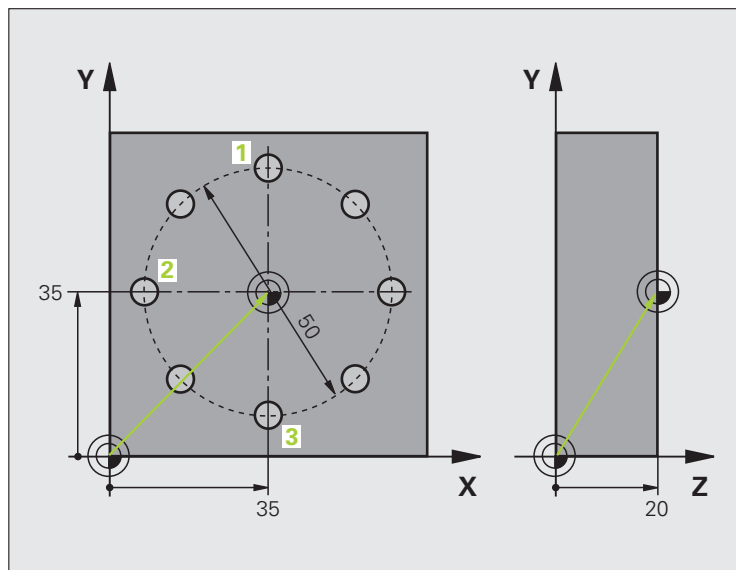
0 BEGIN PGM CYC416 MM	
1 TOOL CALL 0 Z	Call tool 0 to define the touch probe axis
2 TCH PROBE 417 DATUM IN TS AXIS	Cycle definition for datum setting in the touch probe axis
Q263=+7.5 ;1ST POINT 1ST AXIS	Touch point: X coordinate
Q264=+7.5 ;1ST POINT 2ND AXIS	Touch point: Y coordinate
Q294=+25 ;1ST POINT 3RD AXIS	Touch point: Z coordinate
Q320=0 ;SET-UP CLEARANCE	Safety clearance in addition to MP6140
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 35KLZ		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 114
1 POLAR DATUM PLANE Measuring a point in a probing direction		Page 115
420 MEASURE ANGLE Measuring an angle in the working plane		Page 116
421 MEASURE HOLE Measuring the position and diameter of a hole		Page 118
422 MEAS. CIRCLE OUTSIDE Measuring the position and diameter of a circular stud		Page 121
423 MEAS. RECTAN. INSIDE Measuring the position, length and width of a rectangular pocket		Page 124
424 MEAS. RECTAN. OUTSIDE Measuring the position, length and width of a rectangular stud		Page 127
425 MEASURE INSIDE WIDTH (2nd soft-key level) Measuring slot width		Page 130
426 MEASURE RIDGE WIDTH (2nd soft-key level) Measuring the width of a ridge		Page 132
427 MEASURE COORDINATE (2nd soft-key level) Measuring any coordinate in a selectable axis		Page 134
430 MEAS. BOLT HOLE CIRC. (2nd soft-key level) Measuring position and diameter of a bolt hole circle		Page 137
431 MEASURE PLANE (2nd soft-key level) Measuring the A and B axis angles of a plane		Page 140

3 Touch Probe Cycles for Automatic Workpiece Inspection

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 as a file, the TNC, by default, saves the measuring log as an ASCII file in the directory from which you run the measuring program. As an alternative, you can also send the measuring log directly to a printer or transfer it to a PC via the data interface. To do this, set the print function (in the interface configuration menu) to RS232:\ (see also the User's Manual under "MOD Functions, Setting Up the Data Interface").

☞ All measured values listed in the log file are referenced to the datum that is active during the respective cycle you are running. In addition, the coordinate system may have been rotated in the plane or the plane may have been tilted by using 3D-ROT. In this case, the TNC converts the measuring results to the respective active coordinate system.

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.
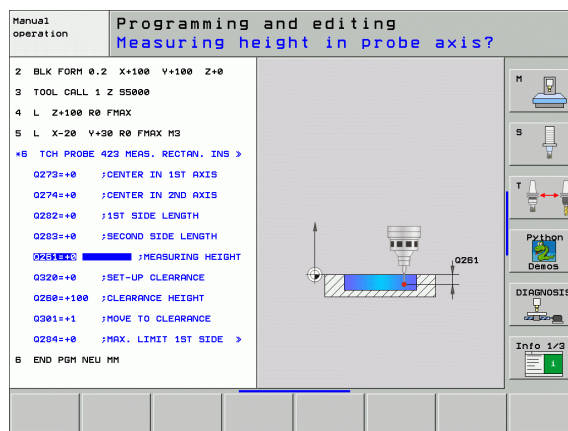
Tool monitoring

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 (enter a tool name or **Q330** unequal to 0. Select the tool name input by soft key. Specifically to AWT Weber: The TNC no longer displays the right single quotation mark.

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 MP6150 or MP6361) to the starting position **1** programmed in the cycle.

2 Then the touch probe approaches the workpiece at the feed rate assigned in MP6120 or MP6360. 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.
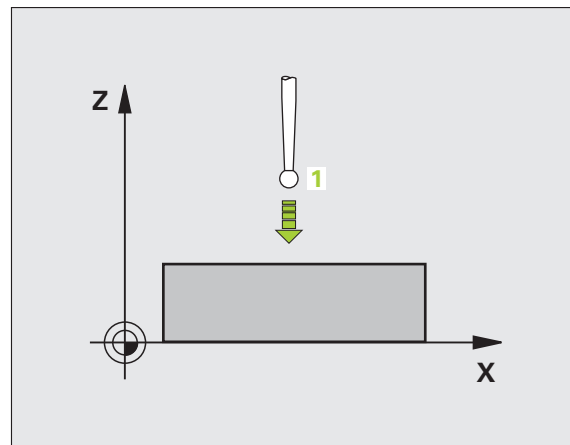
Example: NC blocks

```
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 MP6150 or MP6361) to the starting position **1** programmed in the cycle.

2 Then the touch probe approaches the workpiece at the feed rate assigned in MP6120 or MP6360. 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.

▶ **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.
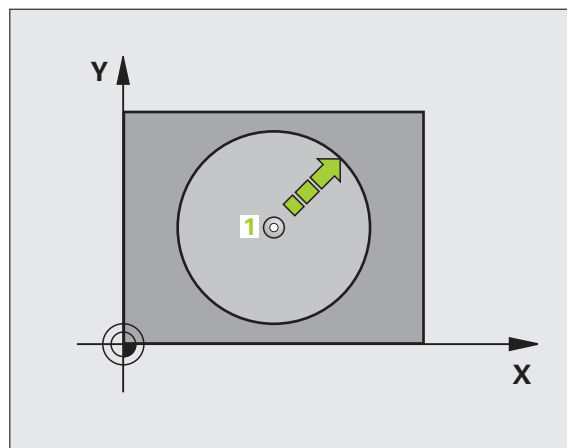
Example: NC blocks

```
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 MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 26) 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 (MP6120 or MP6360).

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 MP6140.

▸ **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: with the standard setting the TNC saves the **log file TCHPR420.TXT** in the directory in which your measuring program is also stored.
2: Interrupt the program run and display the measuring log on the screen. Resume program run with NC Start.
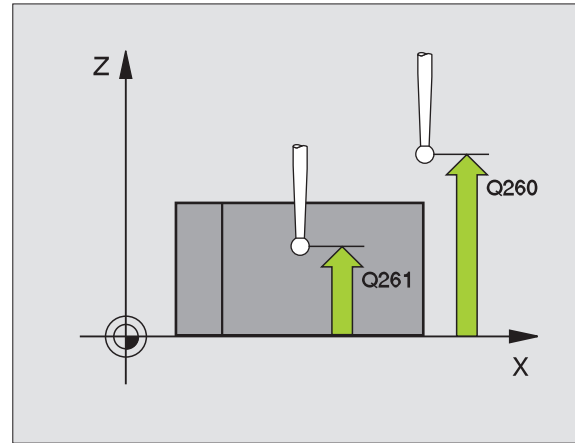
Example: NC blocks

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 to the starting points at rapid traverse (value from MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 26) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120 or MP6360). 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

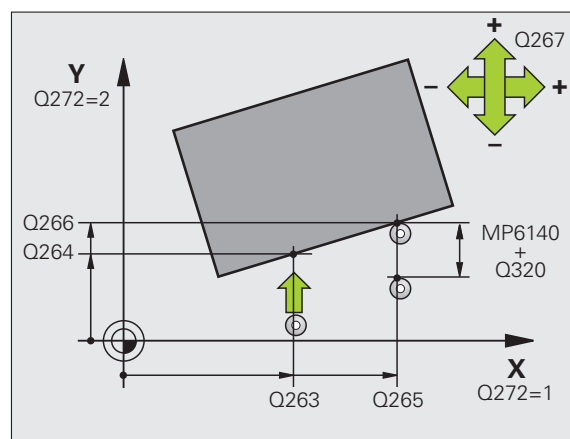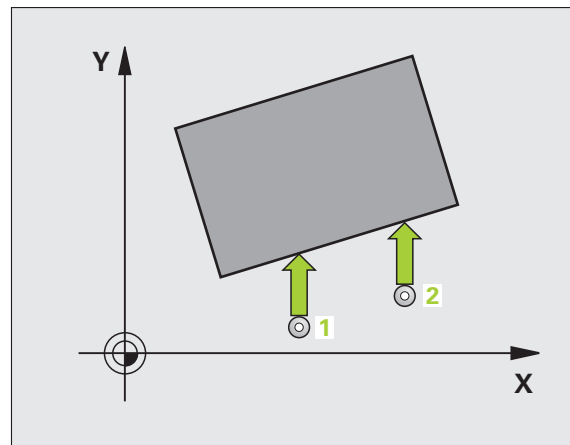☞ **Before programming, note the following**

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

3 Touch Probe Cycles for Automatic Workpiece Inspection

▶ **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 MP6140.

▶ **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.
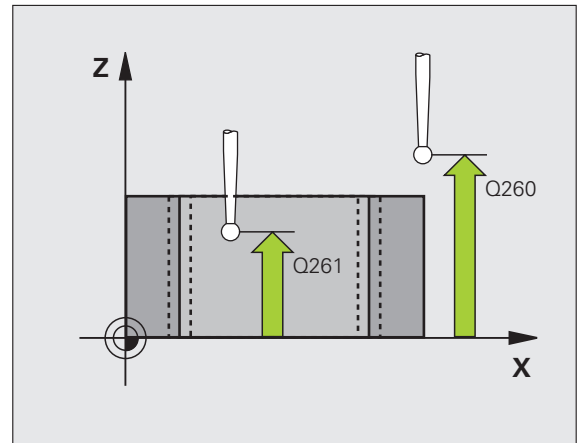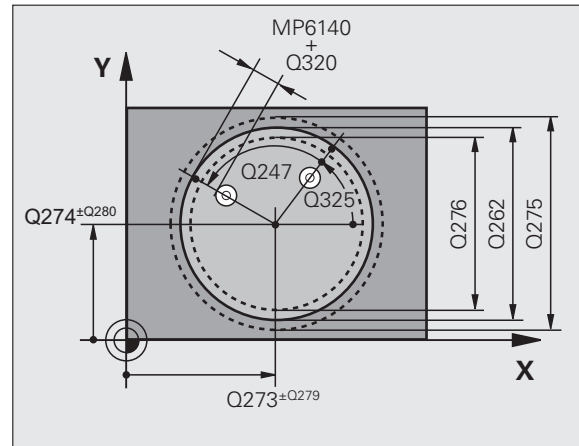
▶ **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
0: No measuring log
1: Generate measuring log: with the standard setting the TNC saves the **log file TCHPR421.TXT** in the directory in which your measuring program is also stored.
2: Interrupt the program run and 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 112)
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

MEAS. CIRCLE OUTSIDE (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.
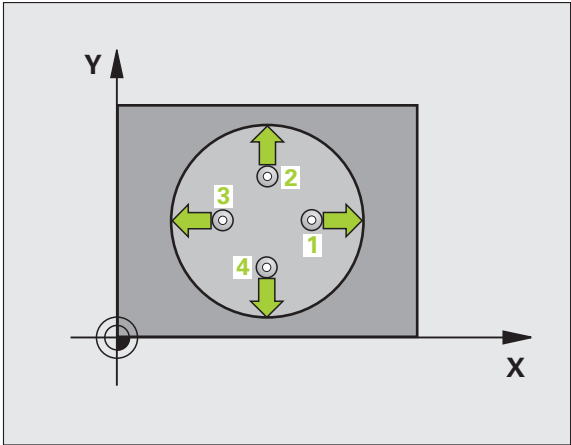
1   The TNC positions the touch probe to the starting points at rapid traverse (value from MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 26) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2   Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120 or MP6360). 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 MP6140.

▶ **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.

3 Touch Probe Cycles for Automatic Workpiece Inspection
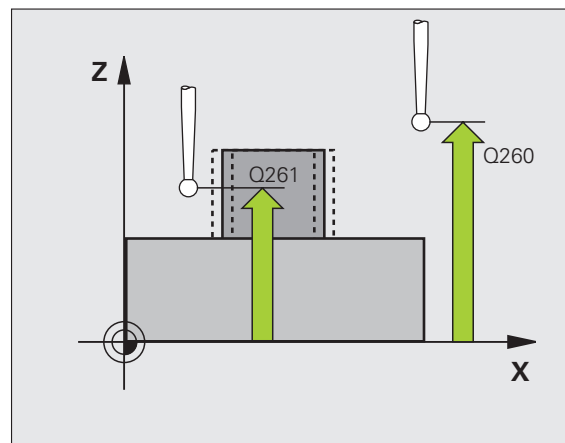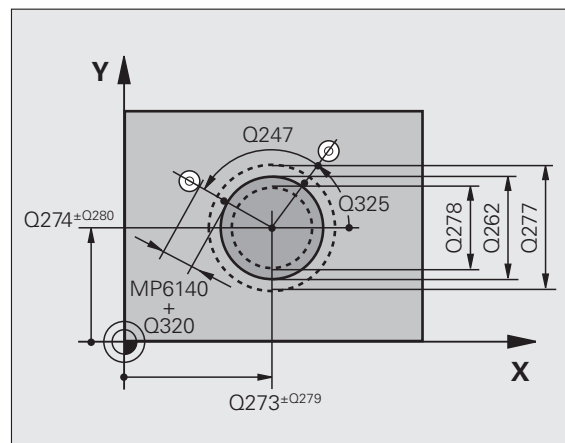
▶ **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
0: No measuring log
1: Generate measuring log: with the standard setting the TNC saves the **log file TCHPR422.TXT** in the directory in which your measuring program is also stored.
2: Interrupt the program run and 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 112):
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

3.3 Automatic Workpiece Measurement

MEAS. RECTAN. INSIDE (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 to the starting points at rapid traverse (value from MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 26) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120 or MP6360).

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.
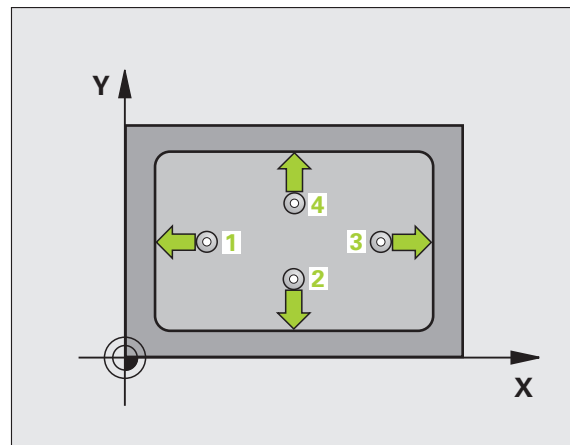
▶ **Center in 1st axis** Q273 (absolute value): Center of the pocket in the reference axis of the working plane.

▶ **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 MP6140.

▶ **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.
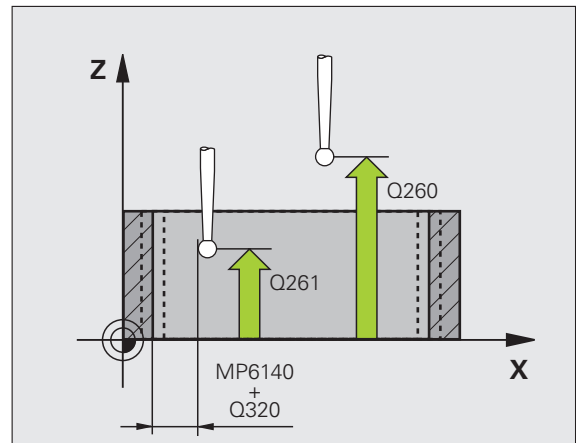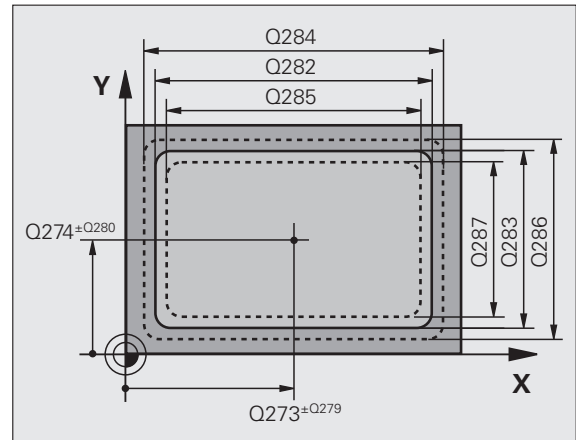
▶ `Measuring log` Q281: Definition of whether the TNC is
to create a measuring log:
0: No measuring log
1: Generate measuring log: with the standard setting
the TNC saves the **log file TCHPR423.TXT** in the
directory in which your measuring program is also
stored.
2: Interrupt the program run and display 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 112)
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 to the starting points at rapid traverse (value from MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 26) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120 or MP6360).

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.

3.3 Automatic Workpiece Measurement

424

▶ **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 MP6140.

▶ **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.

Figure labels: Q284, Q282, Q285, Q274±Q280, Q287, Q283, Q286, Q273±Q279

Figure labels: Z, Q260, Q261, X, MP6140 + Q320
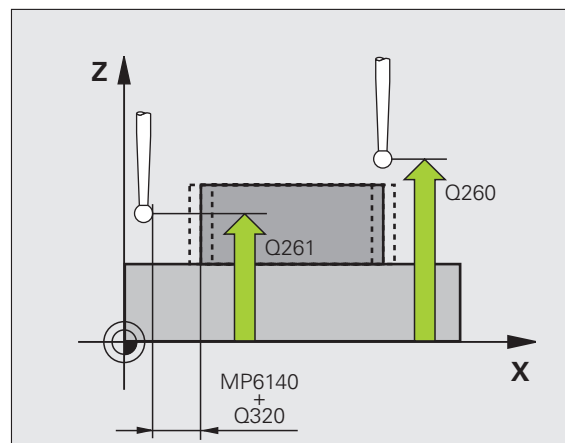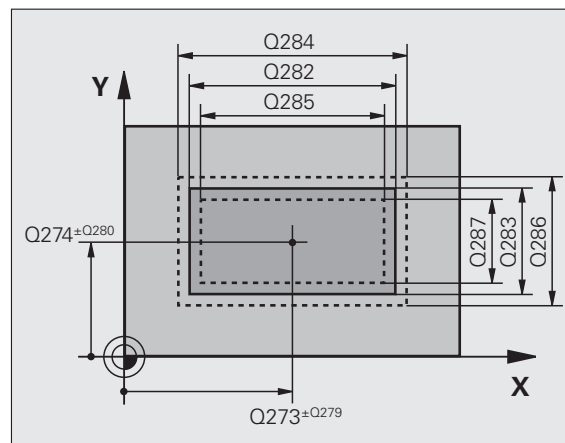
▶ **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
0: No measuring log
1: Generate measuring log: with the standard setting the TNC saves the **log file TCHPR424.TXT** in the directory in which your measuring program is also stored.
2: Interrupt the program run and 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 112):
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 to the starting points at rapid traverse (value from MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 26) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120 or MP6360). 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.
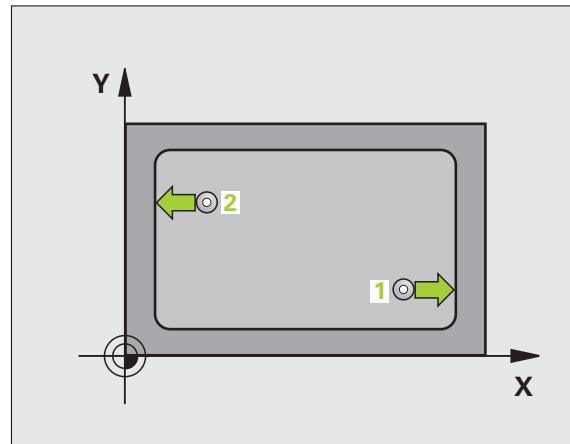
▶ **Starting point in 1st axis** Q328 (absolute): Starting point for probing in the reference axis of the working plane.

▶ **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: with the standard setting the TNC saves the **log file TCHPR425.TXT** in the directory in which your measuring program is also stored.
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 112):
0: Monitoring not active
>0: Tool number in the tool table TOOL.T
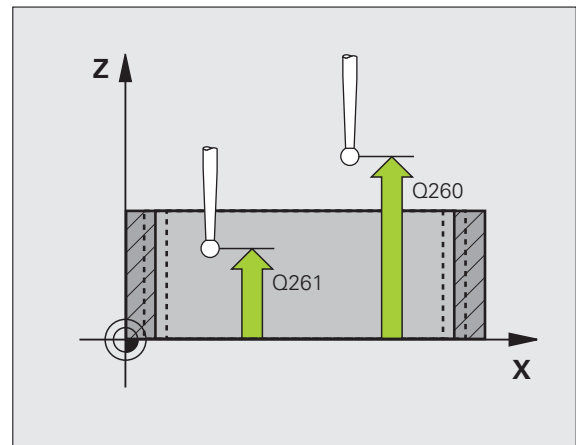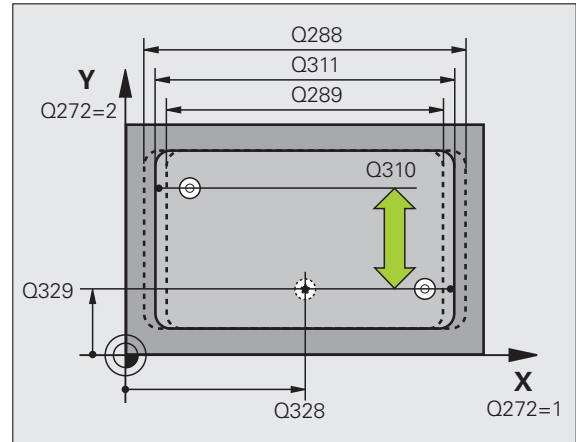
Example: NC blocks

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 RIDGE WIDTH (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 to the starting points at rapid traverse (value from MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 26) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance from MP6140.

2 Then the touch probe moves to the entered measuring height and probes the first touch point at the probing feed rate (MP6120 or MP6360). 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.

▶ **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 MP6140.

- ▶ **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: with the standard setting the TNC saves the **log file TCHPR426.TXT** in the directory in which your measuring program is also stored.
  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 112)
  0: Monitoring not active
  >0: Tool number in the tool table TOOL.T
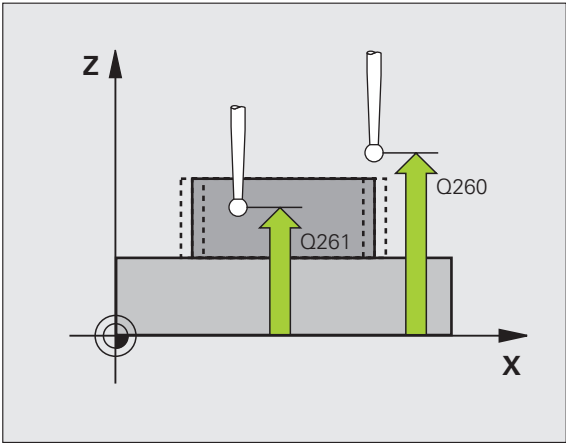
Example: NC blocks

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 to the starting points at rapid traverse (value from MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 26) 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 MP6140.

▶ **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.
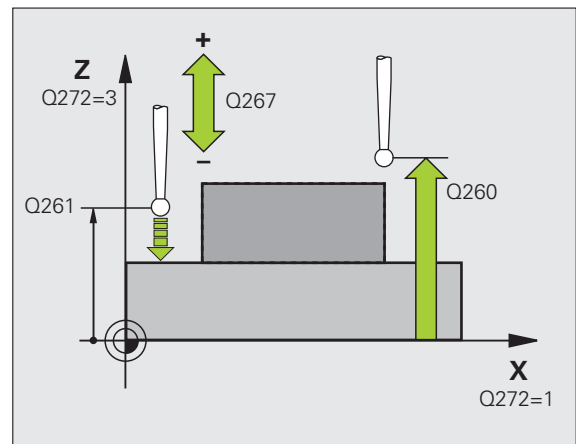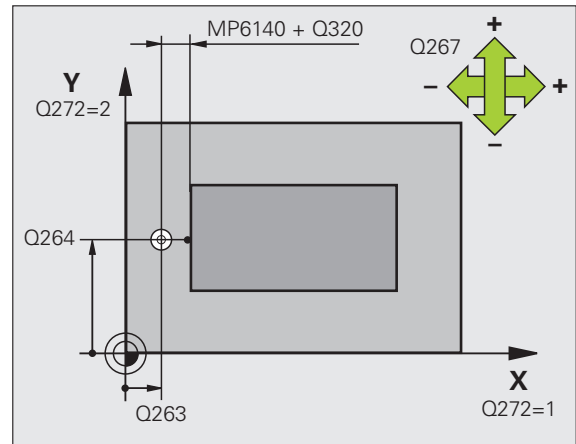
▶ **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
0: No measuring log
1: Generate measuring log: with the standard setting the TNC saves the **log file TCHPR427.TXT** in the directory in which your measuring program is also stored.
2: Interrupt the program run and 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 112):
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.
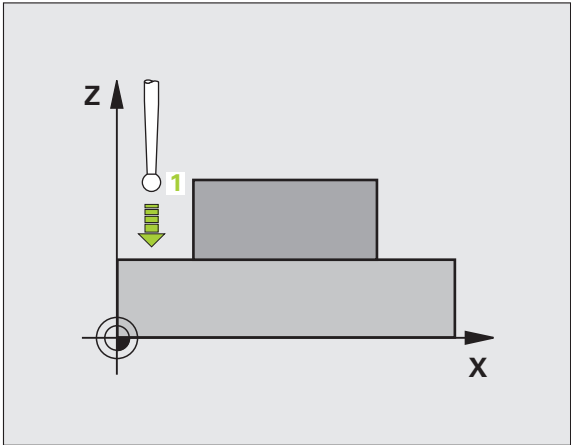
1 Following the positioning logic (see "Running touch probe cycles" on page 26), the TNC positions the touch probe at rapid traverse (value from MP6150 or MP6361) to the point entered as center of the first hole **1**.

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.
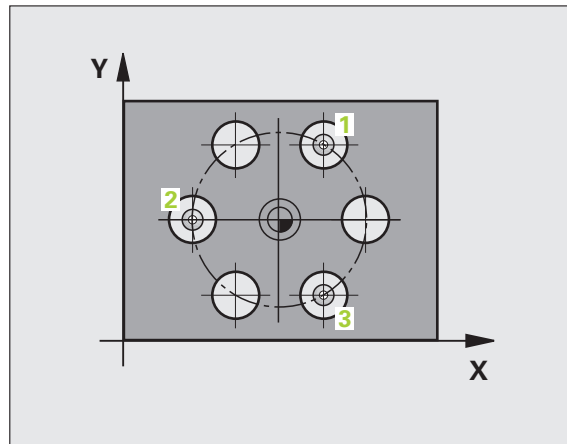
▶ **Center of 1st axis** Q273 (absolute): Bolt hole circle center (nominal value) in the reference axis of the working plane.

▶ **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.
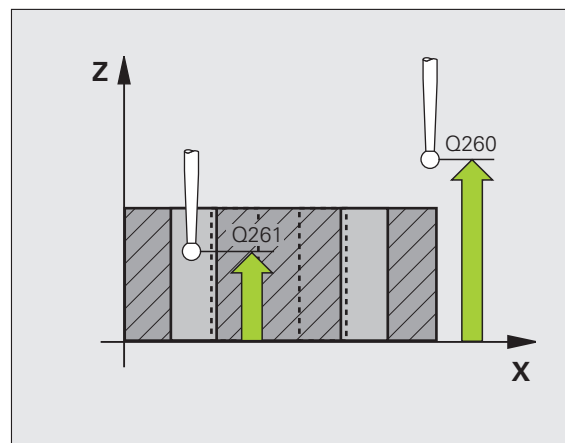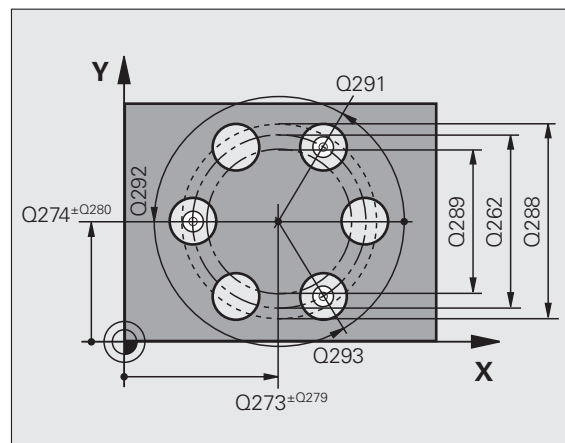
▶ **Measuring log** Q281: Definition of whether the TNC is to create a measuring log:
0: No measuring log
1: Generate measuring log: with the standard setting the TNC saves the **log file TCHPR430.TXT** in the directory in which your measuring program is also stored.
2: Interrupt the program run and 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 112):
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 26), the TNC positions the touch probe at rapid traverse (value from MP6150 or MP6361) 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
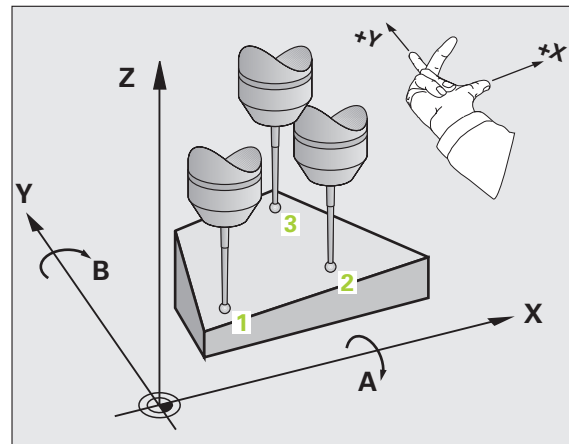
☞ **Before programming, note the following**

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 this case, use the measured spatial angle with **PLANE RELATIV.**

▶ **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 MP6140.

▶ **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: with the standard setting the TNC saves the **log file TCHPR431.TXT** in the directory in which your measuring program is also stored.
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
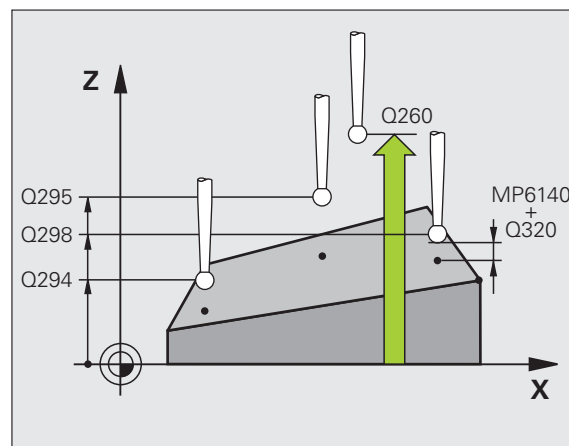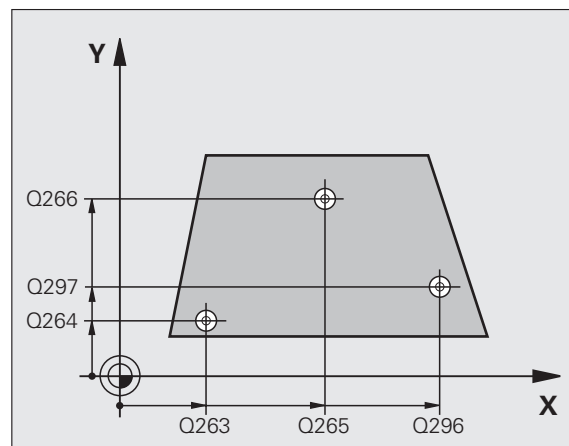
Example: Measuring and reworking a rectangular stud

Program sequence:

- Roughing with 0.5 mm finishing allowance

- Measuring

- Rectangular stud finishing in accordance with the measured values
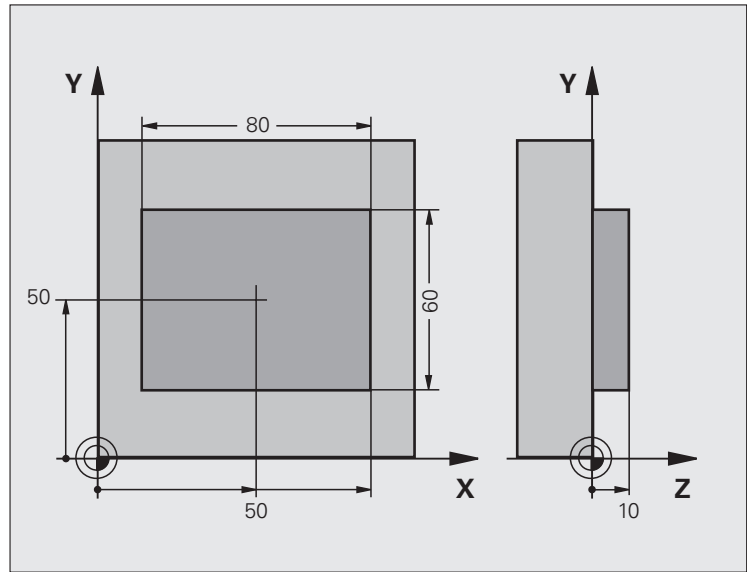
0 BEGIN PGM BEAMS MM	
1 TOOL CALL 0 Z	Prepare tool call
2 L Z+100 R0 FMAX	Retract the tool
3 FN 0: Q1 = +81	Pocket length in X (roughing dimension)
4 FN 0: Q2 = +61	Pocket length in Y (roughing dimension)
5 CALL LBL 1	Call subprogram for machining
6 L Z+100 R0 FMAX	Retract the tool, change the tool
7 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 FMA		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
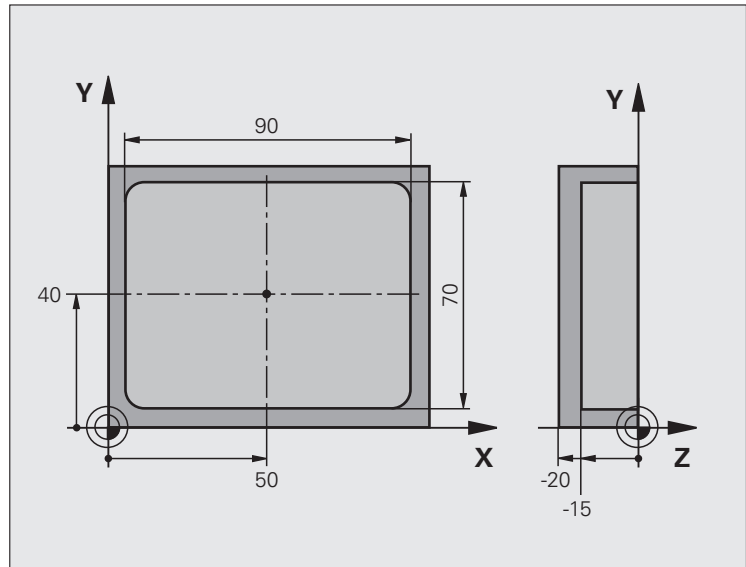
0 BEGIN PGM BSMESS MM		
1 TOOL CALL 1 Z		Tool call for touch probe
2 L Z+100 R0 FMA		Retract the touch probe
3 TCH PROBE 423 MEAS. RECTAN. INSIDE		
Q273=+50	;CENTER 1ST AXIS	
Q274=+40	;CENTER 2ND AXIS	
Q282=90	;FIRST SIDE LENGTH	Nominal length in X
Q283=70	;SECOND SIDE LENGTH	Nominal length in Y
Q261=-5	;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 Touch Probe Cycles for Automatic Workpiece Inspection

3.4 Special Cycles

Overview

The TNC provides four cycles for the following special purposes:

Cycle	Soft key	Page
2 CALIBRATE TS Radius calibration of the touch trigger probe		Page 148
9 CALIBRATE TS LENGTH Length calibration of the touch trigger probe		Page 149
3 MEASURING Cycle for defining OEM cycles		Page 150
4 MEASURING IN 3-D Measuring cycle for 3-D probing for defining OEM cycles		Page 152
440 MEASURE AXIS SHIFT		Page 154
441 FAST PROBING		Page 156

CALIBRATE TS (touch probe cycle 2)

Touch probe cycle 2 automatically calibrates a touch trigger probe using a ring gauge or a precision stud as calibration standard.

☞ Before you begin calibrating, you must define in Machine Parameters 6180.0 to 6180.2 the center of the calibrating workpiece in the working space of the machine (REF coordinates).

If you are working with several traverse ranges you can save a separate set of coordinates for the center of each calibrating workpiece (MP6181.1 to 6181.2 and MP6182.1 to 6182.2).

1 The touch probe moves at rapid traverse (value from MP6150) to the clearance height (but only if the current position is below the clearance height).

2 Then the TNC positions the touch probe in the working plane to the center of the ring gauge (calibration from inside) or in its proximity (calibration from outside).

3 The touch probe then moves to the measuring depth (result of machine parameters 618x.2 and 6185.x) and probes the ring gauge successively in X+, Y+, X- and Y-.

4 Finally, the TNC moves the touch probe to the clearance height and writes the effective radius of the ball tip to the calibration data.

▶ **Clearance height** (absolute): Coordinate in the touch probe axis at which the touch probe cannot collide with the calibration workpiece or any fixtures.

▶ **Radius of ring gauge:** Radius of the calibrating workpiece.

▶ **Inside calib. =0/outs. calib.=1:** Definition of whether the TNC is to calibrate from inside or outside:
0: Calibrate from inside
1: Calibrate from outside

Example: NC blocks

```
5 TCH PROBE 2.0 CALIBRATE TS
```

```
6 TCH PROBE
2.1 HEIGHT: +50 R +25.003 DIRECTION: 0
```

CALIBRATE TS LENGTH (touch probe cycle 9)

Touch probe cycle 9 automatically calibrates the length of a touch trigger probe at a point that you determine.

1 Pre-position the touch probe so that the coordinate defined in the cycle can be accessed without collision.

2 The TNC moves the touch probe in the direction of the negative tool axis until a trigger signal is released.

3 Finally, the TNC moves the touch probe back to the starting point of the probing process and writes the effective touch probe length into the calibration data.

▶ **Coordinate of datum** (absolute): Exact coordinate of the point that is to be probed.

▶ **Reference system? (0=ACT/1=REF):** Specify the coordinate system on which the entered datum is to be based:
0: Entered datum is based on the active workpiece coordinate system (ACT system)
1: Entered datum is based on the active machine coordinate system (REF system)

Example: NC blocks

```
5 L X-235 Y+356 R0 FMAX
```

```
6 TCH PROBE 9.0 CALIBRATE TS LENGTH
```

```
7 TCH PROBE 9.1 DATUM +50 REFERENCE
SYSTEM 0
```

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 6130 (maximum traverse to touch point) and 6120 (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
```

MEASURING IN 3-D (touch probe cycle 4, FCL 3 function)

Touch probe cycle 4 measures any position on the workpiece in the probing direction defined by a vector. Unlike other measuring cycles, Cycle 4 enables you to enter the measuring path and feed rate directly. Also, the touch probe retracts by a definable value after determining the measured value.

1 The touch probe moves from the current position at the entered feed rate in the defined probing direction. Define the probing direction in the cycle by using a vector (delta values in X, Y and Z).

2 After the 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. You define the number of the first 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 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.

Remember that the TNC always writes to 4 successive Q parameters. If the TNC could not determine a valid touch point, the fourth result parameter will have the value –1.

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 coordinate (X).

▶ **Relative measuring path in X:** X component of the direction vector defining the direction in which the touch probe is to move.

▶ **Relative measuring path in Y:** Y component of the direction vector defining the direction in which the touch probe is to move.

▶ **Relative measuring path in Z:** Z component of the direction vector defining the direction in which the touch probe is to move.

▶ **Maximum measuring path:** Enter the maximum distance from the starting point by which the touch probe may move along the direction vector.

▶ **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.

▶ **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).

Example: NC blocks

```
5 TCH PROBE 4.0 MEASURING IN 3-D
```

```
6 TCH PROBE 4.1 Q1
```

```
7 TCH PROBE 4.2 IX-0.5 IY-1 IZ-1
```

```
8 TCH PROBE
4.3 DIST +45 F100 MB50 REFERENCE SYSTEM:0
```

MEASURE AXIS SHIFT (touch probe cycle 440, DIN/ISO: G440)

Touch probe cycle 440 measures the axis shifts of the machine. Make sure that the cylindrical calibrating tool used in connection with the TT 130 has the correct dimensions.

☞ **Prerequisites:**

Before running cycle 440 for the first time, you must have calibrated the tool touch probe with tool-touch-probe cycle 30.

Ensure that the tool data of the calibrating tool has been entered in the tool table TOOL.T.

Before running the cycle, you must activate the calibrating tool with TOOL CALL.

Ensure that the TT tool touch probe is connected to input X13 of the logic unit and is ready to function (Machine Parameter 65xx).

1 The TNC positions the calibrating tool at rapid traverse (value from MP6150 or MP6361) and following the positioning logic (refer to chapter 1.2) in the vicinity of the TT.

2 At first the TNC makes a measurement in the touch probe axis. The calibrating tool is offset by the value you have defined in the tool table TOOL.T under TT: R-OFFS (standard = tool radius). The TNC always performs the measurement in the touch probe axis.

3 Then the TNC makes the measurement in the working plane. You define via parameter Q364 in which axis and in which direction of the working plane the measurement is to be made.

4 If you make a calibration, the TNC saves the calibration data. Whenever you make a measurement, the TNC compares the measured values to the calibration data and writes the deviations to the following Q parameters:

Parameter number	Meaning
Q185	Deviation from calibration value in X
Q186	Deviation from calibration value in Y
Q187	Deviation from calibration value in Z

You can use this value for compensating the deviation through an incremental datum shift (Cycle 7).

5 Finally, the calibrating tool returns to the clearance height.

Before programming, note the following

Before you perform a measurement, you must have made at least one calibration, otherwise the TNC will output an error message. If you are working with several traverse ranges, you have to make a calibration for each of them.

Each time you run cycle 440, the TNC resets the result parameters Q185 to Q187.

If you want to set a limit for the axis shift in the machine axes, enter the desired limits in the tool table TOOL.T under LTOL for the spindle axis and under RTOL for the working plane. If the limits are exceeded, the TNC outputs a corresponding error message after a control measurement.

After the cycle is completed, the TNC restores the spindle settings that were active before the cycle (M3/M4).

▶ **Type of measurement: 0=calibr., 1=measure?**: Specify whether you want to calibrate or make a verification measurement:
0: Calibrate
1: Measure

▶ **Probing directions:** Definition of probing direction(s) in the working plane:
0: Measuring only in the positive direction of the reference axis
1: Measuring only in the positive direction of the minor axis
2: Measuring only in the negative direction of the reference axis
3: Measuring only in the negative direction of the minor axis
4: Measuring in the positive directions of the reference axis and the minor axis
5: Measuring in the positive direction of the reference axis and in the negative direction of the minor axis
6: Measuring in the negative direction of the reference axis and in the positive direction of the minor axis
7: Measuring in the negative directions of the reference axis and the minor axis

The TNC calculates incorrect values if the probing direction(s) for calibrating and measuring do not correspond.

▶ **Setup clearance** (incremental): Additional distance between measuring point and probe contact. Q320 is added to MP6540.

▶ **Clearance height** (absolute): Coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur (referenced to the active datum).

Example: NC blocks

5 TCH PROBE 440 MEASURE AXIS SHIFT	
Q363=1	;DIRECTION
Q364=0	;PROBING DIRECTIONS
Q320=2	;SET-UP CLEARANCE
Q260=+50	;CLEARANCE HEIGHT

FAST PROBING (touch probe cycle 441, DIN/ISO: G441, FCL 2 function)

Touch probe cycle 441 allows the global setting of different touch probe parameters (e.g. positioning feed rate) for all subsequently used touch probe cycles. This makes it easy to optimize the programs so that reductions in total machining time are achieved.

☞ **Before programming, note the following**

There are no machine movements contained in Cycle 441. It only sets different probing parameters.

END PGM, M02, M30 resets the global settings of Cycle 441.

You can activate automatic angle tracking (Cycle Parameter Q399) only if Machine Parameter 6165=1. If you change Machine Parameter 6165, you must recalibrate the touch probe.

▶ **Positioning feed rate Q396:** Define the feed rate at which the touch probe is moved to the specified positions.

▶ **Positioning feed rate=FMAX (0/1) Q397:** Define whether the touch probe is to move at **FMAX** (rapid traverse) to the specified positions.
0: Move at feed rate from **Q396**
1: Move at **FMAX**

▶ **Angle tracking Q399:** Define whether the TNC is to orient the touch probe before each probing process.
0: Do not orient
1: Orient the spindle before each probing process to increase the accuracy

▶ **Automatic interruption Q400:** Define whether the TNC is to interrupt program run and display the measurement results on the screen after a measuring cycle for automatic workpiece measurement:
0: Never interrupt the program run, not even if the output of the measurement results on the screen is selected in the respective probing cycle.
1: Always interrupt program run and display the measurement results on the screen. To continue the program run, press the NC Start button

Example: NC blocks

5 TCH PROBE 441 FAST PROBING	
Q396=3000	;POSITIONING FEED RATE
Q397=0	;SELECT FEED RATE
Q399=1	;ANGLE TRACKING
Q400=1	;INTERRUPTION

4
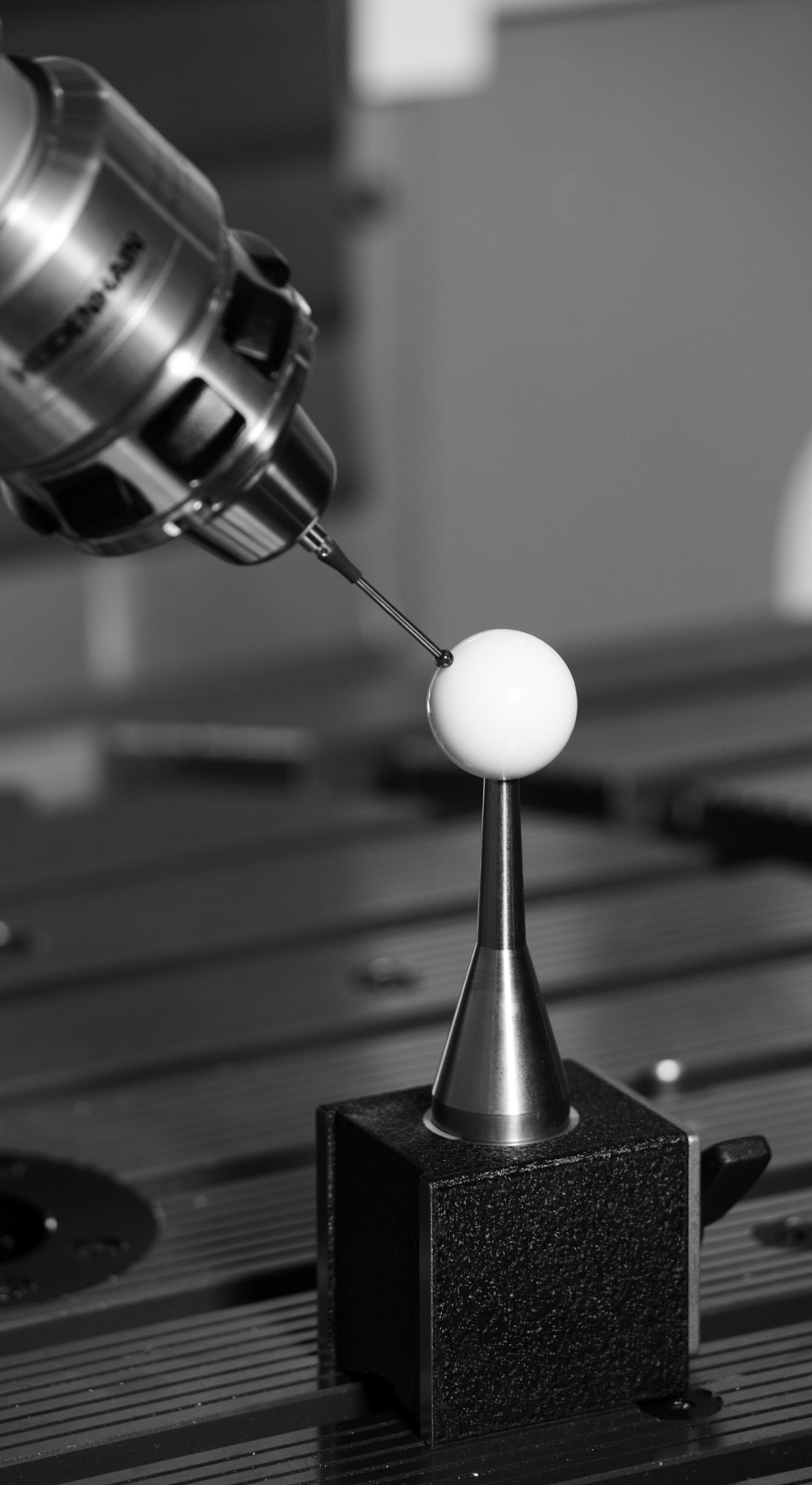
Touch Probe Cycles for Automatic Kinematic Measurement

4.1 Kinematic Measurement with TS Touch Probes (Option KinematicsOpt)

Fundamentals

Accuracy requirements are becoming increasingly stringent, particularly in the area of 5-axis machining. Complex parts are required to be manufactured with precision and reproducible accuracy even over long periods.

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

The new TNC function KinematicsOpt is an important component that helps you to really fulfill these complex requirements: A 3-D touch probe cycle measures the rotary axes on your machine fully automatically, regardless of whether they are in the form of tables or spindle heads. A calibration ball is fixed at any position on the machine table, and measured with a resolution that you define. You simply define for each rotary axis the area that you want to measure.

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

Overview

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

Cycle	Soft key	Page
450 SAVE KINEMATICS: Automatically saving and restoring kinematic configurations		Page 160
451 MEASURE KINEMATICS: Automatically checking or optimizing the machine kinematics		Page 162
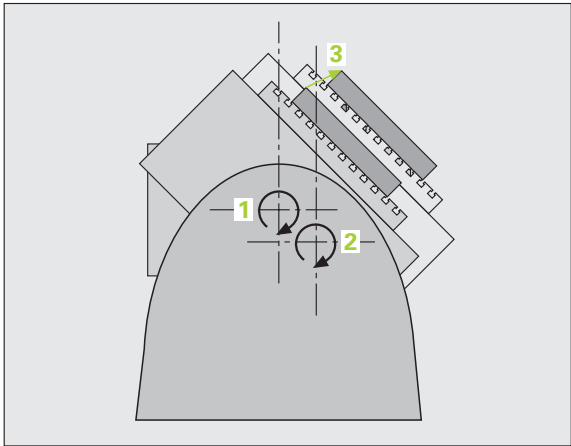
Prerequisites

The following are prerequisites for using the KinematicsOpt option:

- The software options 48 (KinematicsOpt) and 8 (software option 1) and FCL3 must be enabled.
- The 3-D touch probe used for the measurement must be calibrated.
- A calibration ball with an exactly known radius and sufficient rigidity must be attached to some position on the machine table. Calibrations balls can be purchase from various manufacturers of measuring equipment
- The kinematics description of the machine must be complete and correct. The transformation values must be entered with an accuracy of approx. 1 mm.
- All rotary axes must be NC axes. KinematicsOpt does not support measurement of manual axes
- The complete machine geometry must be measured (by the machine tool builder during commissioning)
- Machine parameter **MP6600** must define the tolerance limit starting from which the TNC displays a note in the Optimizing mode when the measured kinematic data is greater than this limit value (see "KinematicsOpt: Tolerance limit in Optimization mode: MP6600" on page 25).
- Machine parameter **MP6601** must define the maximum permissible deviation from the entered cycle parameter by the calibration ball radius measured in the cycles.(see "KinematicsOpt, permissible deviation of the calibration ball radius: MP6601" on page 25)

SAVE KINEMATICS (touch probe cycle 450, DIN/ISO: G450, option)

With the touch probe cycle 450 you can save the active machine kinematic configuration or restore a previously saved one. The are 10 memory spaces available (numbers 0 to 9).

☞ **Before programming, note the following**

Always save the active kinematic configuration before running a kinematic optimization to ensure the following benefit:

You can restore the old data if you are not satisfied with the results or if errors occur during optimization (e.g. power failure)

Save mode: In addition to the kinematic configuration, the TNC always saves the code number (freely definable) last entered under MOD. Then you cannot overwrite this memory space unless you enter this code number. If you have saved a kinematic configuration without a code number, the TNC automatically overwrites this memory space during the next saving process!

Restore mode: The TNC must restore saved data only to a matching kinematic configuration.

Restore mode: Note that a change in the kinematics always changes the preset as well. Set the preset again if necessary.

▶ **Mode (0=save/1=restore)** Q410: Specify whether to save or restore a kinematic configuration:
0: Save active kinematics
1: Restore previously saved kinematic configuration

▶ **Memory (0…9)** Q409: Number of the memory space to which you want to save the entire kinematic configuration, or the number of the memory space from which you want to restore it.
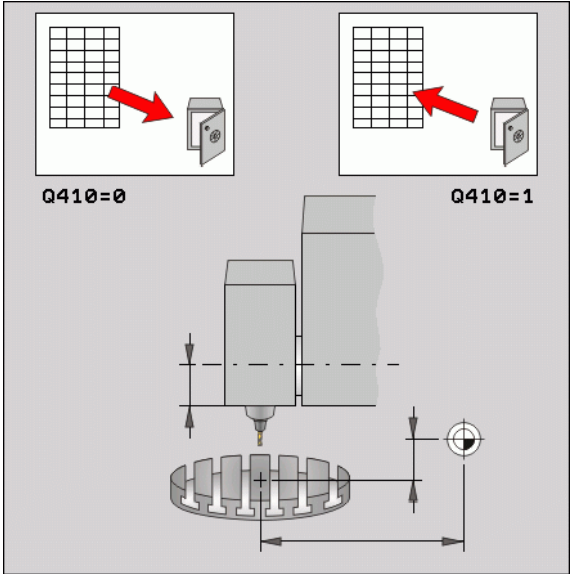
Example: NC blocks

5 TCH PROBE 450 SAVE KINEMATICS	
Q410=0	;MODE
Q409=1	;MEMORY

Log function

After running Cycle 450, the TNC makes a measuring log containing the following information:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Mode used (0=Save/1=Restore)?
- Number of the memory space (0 to 9)
- Line number of the kinematic configuration in the kinematic table
- Code number, if you entered one immediately before running Cycle 450.

MEASURE KINEMATICS (touch probe cycle 451, DIN/ISO: G451, option)

The touch probe cycle 451 enables you to check and, if required, optimize the kinematics of your machine. Use the 3-D TS touch probe to measure any calibration ball that you have attached to the machine table.

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

1 Clamp the calibration ball and check for potential collisions
2 In the Manual operating mode, set the reference point in the ball center
3 In the touch probe axis, manually position the touch probe over the calibration ball, and in the working plane, over the ball center
4 Select the Program Run mode and start the calibration program
5 The TNC automatically measures all three axes successively in the resolution you defined
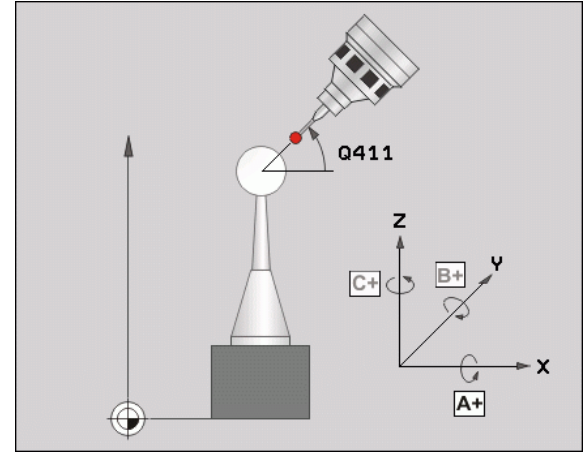
Positioning direction

The positioning direction of the rotary axis to be measured is determined from the start angle and the end angle that you define in the cycle. Specify the start and end angles to ensure that the same position is not measured twice. For example, a start angle of 0° and end angle of 360° result in an error message.

As mentioned above, a duplicated point measurement (e.g. measuring positions +90° und -270°) is not advisable because it can result in different measuring positions, however it does not cause an error message.

- Example: start angle = -270°, end angle = +90°
  The angular position would be identical but results in different measured positions:
  - Start angle = +90°
  - End angle = -270°

  - No. of measuring points = 4
  - Angle step resulting from the calculation = (-270 - +90) / (4-1) = -120°
  - Measuring point 1= +90°
  - Measuring point 2= -30°
  - Measuring point 3= -150°
  - Measuring point 4= -270°

Machines with Hirth-coupled axes

⚠️ In order to be positioned, the axis must move out of the Hirth coupling. So remember to leave a large enough safety clearance to prevent any risk of collision between the touch probe and calibration ball. Also ensure that there is enough space to reach the safety clearance (software limit switch).

Define a retraction height **Q408** greater than 0 if software option 9 (**M128**, **FUNCTION TCPM**) is not available.

When selecting the start and end angle, make sure that each angular step fits in the Hirth grid. With Hirth axes, the TNC checks at the beginning of the cycle whether the calculated angle step matches the Hirth grid. If it doesn't, the TNC shows an error message and terminates the cycle.

The positions are calculated from the start angle, end angle and number of measurements for the respective axis.

Example calculation of measuring positions for an A axis:

Start angle **Q411** = -30

End angle **Q412** = +90

Number of measuring points **Q414** = 4

Calculated angular step = (Q412 - Q411) / (Q414 -1)

Calculated angular step = = (90 - -30) / (4 - 1) = 120 / 3 = 40

Measuring position 1 = Q411 + 0 * angular step = -30°

Measuring position 2 = Q411 + 1 * angular step = +10°

Measuring position 3 = Q411 + 1 * angular step = +50°

Measuring position 4 = Q411 + 1 * angular step = +90°

Choice of number of measuring points

To save time you can make a rough optimization with a small number of measuring points (1-2).

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

This is why you should measure an axis with a tilting range of 0° – 360° at 3 measuring points at 90°, 180° and 270°.

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

☞ You must not measure a rotary axis at 0° or 360°. These positions do not provide any metrologically relevant data.

Choice of the calibrating ball position on the machine table

In principle, you can fix the calibrating ball to any position to the machine table. If it is possible, you can also fasten the calibration ball on fixtures or workpieces (e.g. using a magnetic clamp). The following factors can influence the result of measurement:

■ On machine with rotary tables/tilting tables,
Clamp the calibrating ball as far as possible away from the center of rotation.

■ On machines with very large traverse,
clamp the calibrating ball as closely as possible to the position intended for subsequent machining.

Notes on the accuracy

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

If there were no geometrical and positioning error, any values measured by the cycle at any point on the machine at a certain time would be exactly reproducible. The greater the geometrical and positioning error, the greater is the dispersion of measured results when you fix the calibrating ball to different positions in the machine coordinate system.

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

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

> ☞ If your machine is equipped with a controlled spindle, you should activate the angle tracking using machine parameter **MP6165.** This generally increases the accuracy of measurements with a 3-D touch probe.
>
> If required, deactivate the lock on the rotary axes for the duration of the calibration. Otherwise it may falsify the results of measurement. The machine tool manual provides further information.

In the optimization mode the TNC outputs an evaluation in the measuring log. The valuation number is a measure of the influence of the compensated translations on the measurement result. The higher the valuation number, the greater the benefit from optimization by the TNC.

The valuation of any rotary axis should not fall below a value of **2.** Values greater or equal to **4** are desirable.

> ☞ If the valuation numbers are too small, increase the measurement range of the rotary axis, or also the number of measuring points. If these measures do not improve the valuation number it may be because of an incorrect kinematic description. If necessary, inform your service agency.

Notes on various calibration methods

- **Rough optimization during commissioning after entering approximate dimensions.**
  - Number of measuring points between 1 and 2
  - Angular step of the rotary axes: Approx. 90°

- **Fine optimization over the entire range of traverse**
  - Number of measuring points between 3 and 6
  - The start and end angles should cover the largest possible traverse range of the rotary axes
  - Position the calibrating ball on the machine table so that on rotary table axes there is a large measuring circle, or so that on swivel head axes the measurement can be made at a representative position (e.g. in the center of the traverse range).

- **Optimization of a specific rotary axis position**
  - Number of measuring points between 2 and 3
  - The measurements are made near the rotary axis angle at which the workpiece is to be machined
  - Position the calibration ball on the machine table for calibration at the position subsequently intended for machining

- **Inspecting the machine accuracy**
  - Number of measuring points between 4 and 8
  - The start and end angles should cover the largest possible traverse range of the rotary axes

- **Determining the rotary axis backlash during inspection**
  - Number of measuring points between 8 and 12
  - The start and end angles should cover the largest possible traverse range of the rotary axes

Backlash

Backlash is a small amount of play between the rotary or angle encoder and the table that occurs when the traverse direction is reversed. If the rotary axes have backlash outside of the control loop it can result in significant error during tilting. The cycle automatically activated the internal backlash compensation in digital rotary axes without separate position measurement input.

In the Check mode, the TNC runs the measurement series for each axis to be able to reach the measuring positions from both directions. The TNC documents the arithmetic mean of absolute values of the measured rotary axis backlash.

☞ For reasons of accuracy, if the measuring circle radius is < 100 mm, the TNC does not calculate the backlash. The larger the measuring circle radius, the more accurately the TNC can determine the backlash.

Define the cycle

☞ **Before programming, note the following**

Note that all functions for tilting in the working plane are reset. **M128** or **TCPM FUNCTION** must not be active.

Position of the calibrating ball on the machine table so that there can be no collisions during the measuring process.

Before defining the cycle you must set the reference point in the center of the calibrating ball and activate it.

For the positioning feed rate when moving to the probing height in the touch probe axis, the TNC uses the value from cycle parameter **Q253** or machine parameter MP6150, whichever is smaller. The TNC always moves the rotary axes at positioning feed rate **Q253,** while the probe monitoring is inactive.

If the kinematic data attained in the optimize mode are greater than the permissible limit (**MP6600**), the TNC shows a warning. Then you have to confirm acceptance of the attained value by pressing NC start.

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

In the first probing process the TNC first measures the radius of the calibrating ball. If the measured ball radius differs from the entered ball radius by more than you have defined in machine parameter **MP6601,** the TNC shows an error message and ends the measurement.

If you interrupt the cycle during the measurement, the kinematic data might no longer be in the original condition. Save the active kinematic configuration before an optimization with Cycle 450, so that in case of a failure the most recently active kinematic configuration can be restored.

Programming in inches: The TNC always records the log data and results of measurement in millimeters.

i

451

▶ **Mode (0=Check/1=Measure)** Q406: Specify whether the TNC should check or optimize the active kinematics:
0: Check the active machine kinematics. The TNC measures the kinematics in the axes you have defined, but it does not make any changes to it. The TNC displays the results of measurement in a measurement log
1: Optimize the active machine kinematics. The TNC measures and optimizes the kinematics in the axes you have defined.

▶ **Exact calibration sphere radius** Q407: Enter the exact radius of the calibrating ball used

▶ **Setup clearance** Q320 (incremental): Additional distance between measuring point and ball tip. Q320 is added to MP6140.

▶ **Retraction height** Q408 (absolute):

■ Input 0:
Do not move to any retraction height. The TNC moves to the next measuring position in the axis to be measured. Not allowed for Hirth axes! The TNC moves to the first measuring position in the series A, the B, the then C

■ Input >0:
Retraction height in the untilted workpiece coordinate system to which the TNC positions before a rotary axis positioning in the spindle axis. Also, the TNC moves the touch probe in the working plane to the datum. The probe monitoring is not active in this mode. Define the positioning velocity in parameter Q253.

▶ **Feed rate for pre-positioning** Q253: Traversing speed of the tool during positioning in mm/min.

▶ **Reference angle** Q380 (absolute): Reference angle (basic rotation) for measuring the measuring points in the active workpiece coordinate system. Defining a reference angle can considerably enlarge the measuring range of an axis

Example: Calibration program

4 TOOL CALL "TASTER" Z	
5 TCH PROBE 450 SAVE KINEMATICS	
Q410=0	;MODE
Q409=5	;MEMORY
6 TCH PROBE 451 MEASURE KINEMATICS	
Q406=1	;MODE
Q407=14.9996	;SPHERE RADIUS
Q320=0	;SET-UP CLEARANCE
Q408=0	;RETR. HEIGHT
Q253=750	;F PRE-POSITIONING
Q380=0	;REFERENCE ANGLE
Q411=-90	;START ANGLE A AXIS
Q412=+90	;END ANGLE A AXIS
Q413=0	;INCID. ANGLE A AXIS
Q414=2	;MEAS. POINTS A AXIS
Q415=-90	;START ANGLE B AXIS
Q416=+90	;END ANGLE B AXIS
Q417=0	;INCID. ANGLE B AXIS
Q418=2	;MEAS. POINTS B AXIS
Q419=-90	;START ANGLE C AXIS
Q420=+90	;END ANGLE C AXIS
Q421=0	;INCID. ANGLE C AXIS
Q422=2	;MEAS. POINTS C AXIS

▶ **Start angle A axis** Q411 (absolute): Starting angle in the A axis at which the first measurement is to be made

▶ **End angle A axis** Q412 (absolute): Ending angle in the A axis at which the last measurement is to be made

▶ **Angle of incid. A axis** Q413: Angle of incidence in the A axis at which the other rotary axes are to be measured

▶ **Number meas. points A axis** Q414: Number of probe measurements with which the TNC is to measure the A axis

▶ **Start angle B axis** Q415 (absolute): Starting angle in the B axis at which the first measurement is to be made

▶ **End angle B axis** Q416 (absolute): Ending angle in the B axis at which the last measurement is to be made

▶ **Angle of incid. in B axis** Q417: Angle of incidence in the B axis at which the other rotary axes are to be measured

▶ **Number meas. points B axis** Q418: Number of probe measurements with which the TNC is to measure the B axis

▶ **Starting angle in C axis** Q419 (absolute): Starting angle in the C axis at which the first measurement is to be made

▶ **End angle C axis** Q420 (absolute): Ending angle in the C axis at which the last measurement is to be made

▶ **Angle of incid. in C axis** Q421: Angle of incidence in the C axis at which the other rotary axes are to be measured

▶ **Number meas. points C axis** Q422: Number of probe measurements with which the TNC is to measure the C axis

Log function

After running Cycle 451, the TNC makes a measuring log containing the following information:

- Creation date and time of the log
- Path of the NC program from which the cycle was run
- Mode used (0=Check/1=Optimize)
- Active kinematic number
- Entered calibrating ball radius
- For each measured rotary axis:
  - Start angle
  - End angle
  - Number of measuring points
  - Angle of incidence
  - Measuring circle radius
  - Ascertained backlash
  - Measured dispersion
  - Optimized dispersion
  - Compensation values
  - Valuations

5

Touch Probe Cycles for Automatic Tool Measurement

5.1 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

> The TNC uses the feed rate for probing defined in MP6520 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:

$n = MP6570 / (r \cdot 0.0063)$ where

n	Spindle speed (rpm)
MP6570	Maximum permissible cutting speed in m/min
r	Active tool radius in mm

The feed rate for probing is calculated from:

$v = $ meas. tolerance $\cdot$ n where

v	Feed rate for probing in mm/min
Measuring tolerance	Measuring tolerance [mm], depending on MP6507
n	Speed in rpm

MP6507 determines the calculation of the probing feed rate:

MP6507=0:

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 (MP6570) and the permissible tolerance (MP6510), the sooner you will encounter this effect.

MP6507=1:

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	MP6510
30 to 60 mm	2 • MP6510
60 to 90 mm	3 • MP6510
90 to 120 mm	4 • MP6510

MP6507=2:

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

Measuring tolerance = (r • MP6510)/ 5 mm), where

r   Active tool radius in mm
MP6510   Maximum permissible error of measurement

Entries in the tool table TOOL.T

Abbr.	Input	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 = –)?`
TT:R-OFFS	For tool length measurement: Tool offset between stylus center and tool center. Preset value: Tool radius R (NO ENT means **R**).	`Tool offset: radius?`
TT:L-OFFS	Tool radius measurement: Tool offset in addition to MP6530 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	TT:R-OFFS	TT: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 MP6530 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 MP6530 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)

Display the results of measurement

You can display the results of tool measurement in the additional status display (in the machine operating modes). The TNC then shows the program blocks in the left and the measuring results in the right screen window. The measuring results that exceed the permissible wear tolerance are marked in the status display with an asterisk "*"; the results that exceed the permissible breakage tolerance are marked with the character B.
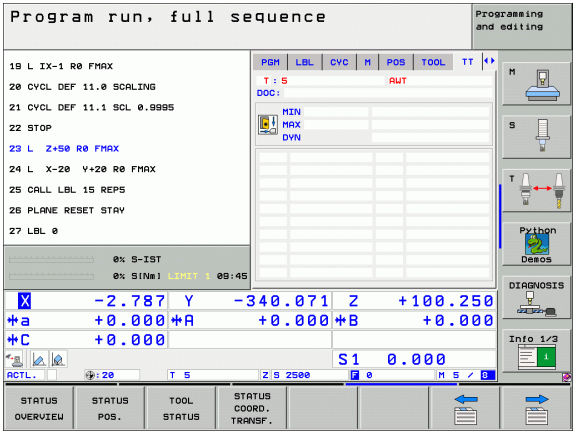
5.2 Available Cycles

Overview

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

Cycle	Old format	New format
Calibrating the TT	30 CAL.	480 CAL.
Measuring the tool length	31	481
Measuring the tool radius	32	482
Measuring tool length and radius	33	483

☞ 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.

You can also measure tools in a tilted working plane.

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 functioning of the calibration cycle is dependent on MP 6500. 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 6580.0 to 6580.2.

If you change the setting of any of the Machine Parameters 6580.0 to 6580.2, you must recalibrate the TT.

The TT 120 is calibrated automatically 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 178). 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.
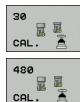
▶ **Clearance height:** Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from MP6540).

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 cycle TCH PROBE 31or TCH PROBE 480 (see also "Differences between Cycles 31 to 33 and Cycles 481 to 483" on page 178). Depending on the input parameters, you can measure the length of a tool by one of the following 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 (**TT: 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 (**TT: 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 MP6530. You can enter an additional offset with Tool offset: Length (**TT: 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. The TNC 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 MP6540).

▶ **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 178). Depending on the input parameters, you can measure the radius of a tool by:

■ 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 and adjust the machine parameter 6500. Refer to your machine manual.

Sequence of measurement

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 MP6530. 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 MP6540).

▶ `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 178). 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 and adjust the machine parameter 6500. Refer to your machine manual.

Sequence of measurement
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 MP6540).

▶ **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
```

Overview

Touch probe cycles

Cycle number	Cycle designation	DEF-active	CALL-active	Page
0	Reference plane	■		Page 114
1	Polar datum	■		Page 115
2	Calibrate TS radius	■		Page 148
3	Measuring	■		Page 150
4	Measuring in 3-D	■		Page 152
9	Calibrate TS length	■		Page 149
30	Calibrating the TT	■		Page 179
31	Measure/Inspect the tool length	■		Page 180
32	Measure/Inspect the tool radius	■		Page 182
33	Measure/Inspect the tool length and the tool radius	■		Page 184
400	Basic rotation using two points	■		Page 50
401	Basic rotation from two holes	■		Page 52
402	Basic rotation from two studs	■		Page 55
403	Compensate misalignment with rotary axis	■		Page 58
404	Set basic rotation	■		Page 61
405	Compensate misalignment with the C axis	■		Page 62
408	Reference point at slot center (FCL 3 function)	■		Page 70
409	Reference point at ridge center (FCL 3 function)	■		Page 73
410	Datum from inside of rectangle	■		Page 76
411	Datum from outside of rectangle	■		Page 79
412	Datum from inside of circle (hole)	■		Page 82
413	Datum from outside of circle (stud)	■		Page 85
414	Datum from outside of corner	■		Page 88
415	Datum from inside of corner	■		Page 91
416	Datum from circle center	■		Page 94
417	Datum in touch probe axis	■		Page 97

Overview

Cycle number	Cycle designation	DEF-active	CALL-active	Page
418	Datum at center between four holes	■		Page 99
419	Datum in any one axis	■		Page 102
420	Workpiece—Measure angle	■		Page 116
421	Workpiece—Measure hole (center and diameter of hole)	■		Page 118
422	Workpiece—Measure circle from outside (diameter of circular stud)	■		Page 121
423	Workpiece—Measure rectangle from inside	■		Page 124
424	Workpiece—Measure rectangle from outside	■		Page 127
425	Workpiece—Measure inside width (slot)	■		Page 130
426	Workpiece—Measure outside width (ridge)	■		Page 132
427	Workpiece—Measure coordinate (in any one axis)	■		Page 134
430	Workpiece—Measure bolt hole circle	■		Page 137
431	Workpiece—Measure plane	■		Page 140
440	Measure axis shift	■		Page 154
441	Rapid probing: Set global touch probe parameters (FCL 2 function)	■		Page 156
450	Save kinematics (option)	■		Page 160
451	Measure kinematics (option)	■		Page 162
480	Calibrating the TT	■		Page 179
481	Measure/Inspect the tool length	■		Page 180
482	Measure/Inspect the tool radius	■		Page 182
483	Measure/Inspect the tool length and the tool radius	■		Page 184

HEIDENHAIN

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