HEIDENHAIN

Touch Probe Cycles
TNC 426
TNC 430

User's Manual

TNC Model, Software and Features

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

TNC model	NC software number
TNC 426, TNC 430	280 472-10
TNC 426, TNC 430	280 474-13
TNC 426, TNC 430	280 476-04

The suffixes E and F indicate the export versions of the TNC The export versions of the TNC have the following limitations:

■ Linear movement is possible in no more than 4 axes simultaneously.

The machine tool builder adapts the useable 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 your machine tool.

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

■ Digitizing option
■ 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 respective control. Please contact HEIDENHAIN if you need a copy of this User's Manual.

Location of use

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 features of the NC software 280 476-xx

■ Management of an arbitrary number of calibration data with the TS triggering touch probe (see "Managing more than one block of calibrating data (as of NC software 280 476-xx)" on page 15)

■ Cycles for automatic tool measurement with the TT 130 according to ISO (see "Overview" on page 112)

■ Cycle for measuring the thermal behavior of a machine (see "MEASURE AXIS SHIFT (touch probe cycle 440, ISO: G440, available as of NC software 280 476-xx)" on page 106)

Changed features of the NC software 280 476-xx

■ All cycles for the automatic datum setting can now also be run during an active basic rotation (see "Characteristics common to all touch probe cycles for datum setting" on page 43).

■ Cycle 431 find the angular values needed for tilting the working plane with a spatial angle (see "MEASURE PLANE (touch probe cycle 431, ISO: G431)" on page 97).

Contents

3 Touch Probe Cycles for Automatic Workpiece Inspection 27

x

1

Introduction

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).

If you are working alternately with a triggering and a measuring touch probe, be sure that

- You have selected the correct touch probe in MP 6200.
- The measuring and triggering touch probes are never connected to the control at the same time.

The TNC cannot detect which probe is actually in the spindle.

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 in rapid traverse.

If the stylus is not deflected within a distance defined in MP 6130), the TNC displays an error message.
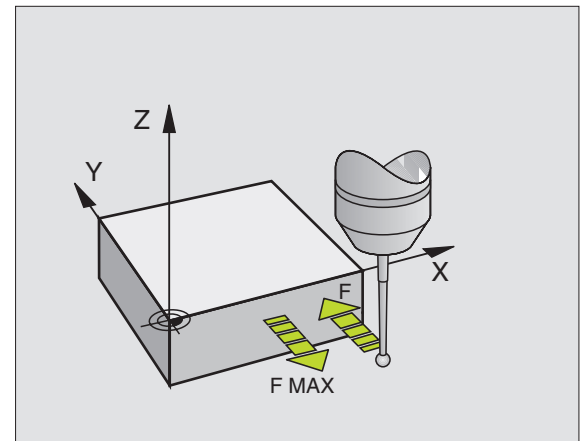
Touch Probe Cycles in the Manual and Electronic Handwheel Modes

In the Manual and Electronic Handwheel operating modes, the TNC provides touch probe cycles that allow you to:

- Calibrate the touch probe
- Compensate workpiece misalignment
- Datum setting

Touch probe cycles for automatic operation

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

- Calibrating the touch probe (Chapter 3)
- Compensating workpiece misalignment (Chapter 3)
- Setting datums (Chapter 3)
- Automatic workpiece inspection (Chapter 3)
- Automatic workpiece measurement (Chapter 4)
- Digitizing with triggering or measuring touch probe (Option, Chapter 5)

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 the same function that the TNC requires in several cycles always have the same number: For example, Q260 is always assigned the clearance height, Q261 the measuring height, etc.

To simplify programming, the TNC shows an illustration during cycle definition. In the illustration, the parameter that needs to be entered is highlighted (see figure at right).
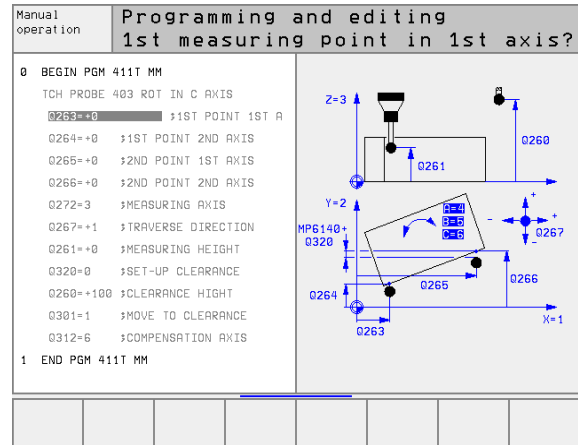
☞ To improve clarity, the help illustrations sometimes omit certain parameters.

Defining the touch probe cycle in the Programming and Editing operation mode

▶ 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 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
Cycles for automatic measurement and compensation of workpiece misalignment	
Cycles for automatic datum setting	
Cycles for automatic workpiece inspection	
Automatic calibration cycle	SPECIAL CYCLES
Cycles for digitizing with measuring touch probe (option, not available for ISO)	TM
Cycles for digitizing with measuring touch probe (option, not available for ISO)	TS
Cycles for automatic tool measurement (enabled by the machine tool builder, not ISO)	

Example: NC blocks

5 TCH PROBE 410 DATUM INSIDE RECTAN.	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q323=60	;1ST SIDE LENGTH
Q323=60	;1ST SIDE LENGTH
Q324=20	;2ND SIDE LENGTH
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;TRAVERSE TO CLEAR HEIGHT
Q305=10	;NO. IN TABLE
Q331=+0	;DATUM
Q332=+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 (as of 280 476-10)

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.

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.
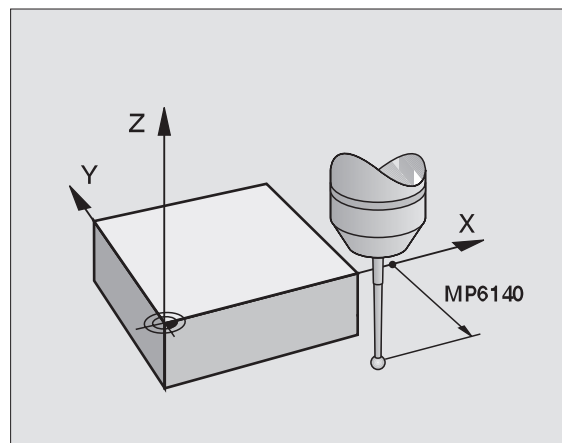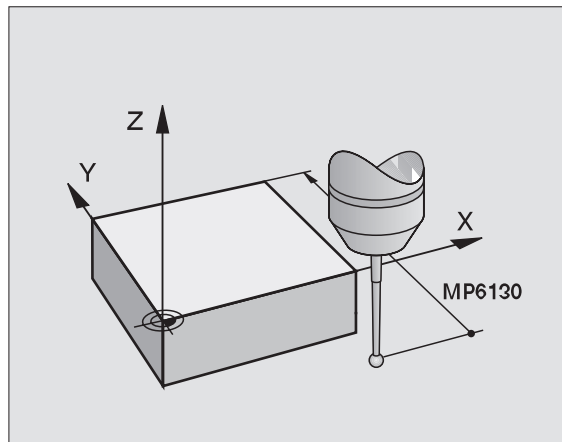
Touch trigger probe, probing feed rate: MP6120

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

Touch trigger probe, rapid traverse for pre-positioning: MP6150

In MP6150 you define the feed rate at which the TNC pre-positions the touch probe, or positions it between measuring points.

Measuring touch probe, probing feed rate: MP6360

In MP6360 you define the feed rate at which the TNC is to probe the workpiece.

Measuring touch probe, rapid traverse for pre-positioning: MP6361

In MP6361 you define the feed rate at which the TNC pre-positions the touch probe, or positions it between measuring points.
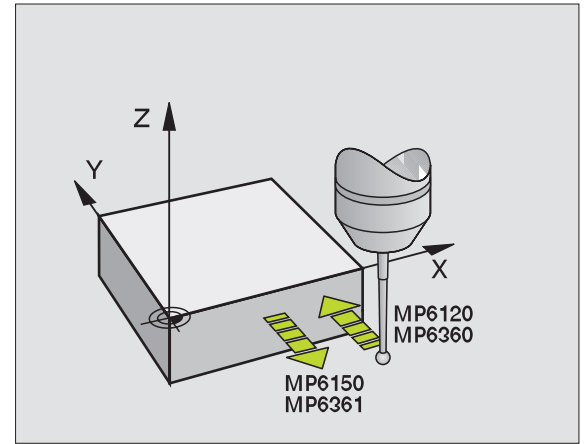
Running touch probe cycles

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 respective control, "General User Parameters").

NC software 280.476-xx

You can also run the touch probe cycles 410 to 418 during an active basic rotation. Make sure, however, that the basic rotation angle does not change when you use cycle 7 "zero shift from datum table" 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
Calibrate the effective length	CAL L
Calibrate the effective radius	CAL R
Measure a basic rotation using a line	PROBING ROT
Datum setting in any axis	PROBING POS
Set the datum at a corner	PROBING P
Set the datum at a circle center	PROBING CC
Measure a basic rotation using two holes/cylindrical studs	PROBING ROT
Set the datum using four holes/cylindrical studs	PROBING P
Set the circle center using three holes/cylindrical studs	PROBING CC

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 at right.

PROBING ROT

▶ 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 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 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 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.

```
Manual        Programming and editing
operation

File: %TCHPRNT.A          Line: 0     Column: 1   INSERT
@ALIBRATE TM:
-------------
08-26-1999, 17:15:02
TCH PROBE AXIS     : Z
PROBE TIP RADIUS 1 : 2.000 MM
PROBE TIP RADIUS 2 : 2.000 MM
RING GAUGE DIAMETER : 50.000 MM
COMPENSATION FACTOR : X = 1.0000
                    : Y = 1.0000
                    : Z = 1.0000
SPRING FORCE RATIO  : FX/FZ = 1.0000
                    : FY/FZ = 1.0000
[END]
```

| INSERT OVERWRITE | MOVE WORD >> | MOVE WORD << | PAGE ⇑ | PAGE ⇓ | BEGIN ⇑ | END ⇓ | FIND |
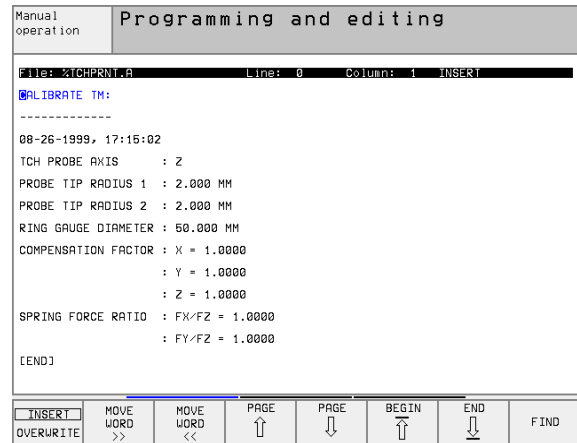
Writing the measured values from probe cycles in datum tables

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

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

▶ Select any probe function.

▶ Enter the desired coordinates of the datum in the appropriate input fields (depends on the touch probe cycle being run).

▶ Enter the datum number in the **datum number =** input box.

▶ Enter the name of the datum table (complete path) in the datum table input box.

▶ Press the soft key ENTER IN DATUM TABLE. The TNC displays whether the data are to be transferred to the indicated datum table as actual values or reference values.

If, in addition to the desired coordinate of the datum, you wish to enter an incremental distance in the table, switch the soft key DISTANCE to ON. The TNC then displays an additional input box for each axis, in which you can enter the desired distance. The TNC then writes the sum of the desired datum and its assigned distance into the table.

👆 If immediately after probing you have used the probing menu to reset the datum, do not write the probe values to a datum table. The probe values saved by the TNC are always based on the datum that was active at the time of probing. Writing the probe values to a datum table would result in incorrect entries.

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

▶ 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 fields.

▶ 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.
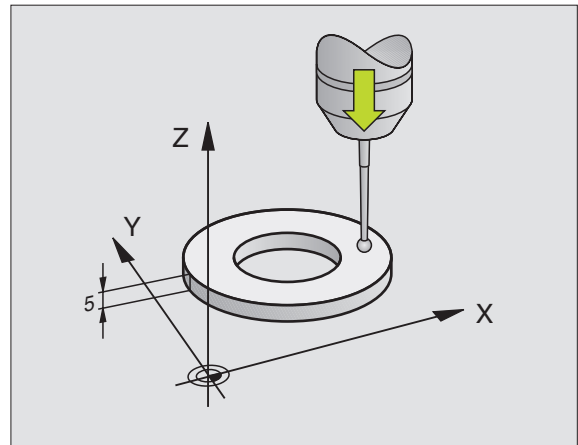
Calibrating the effective radius and compensating center misalignment

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.

For this operation the TNC rotates the 3-D touch probe by 180°. The rotation is initiated by a miscellaneous function that is set by the machine tool builder in machine parameter 6160.

The center misalignment is measured after the effective ball tip radius is calibrated.

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

CAL R
⊚
▶ 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 ENDE 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.

180°
▶ If you want to determine the ball-tip center misalignment, press the180° 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.
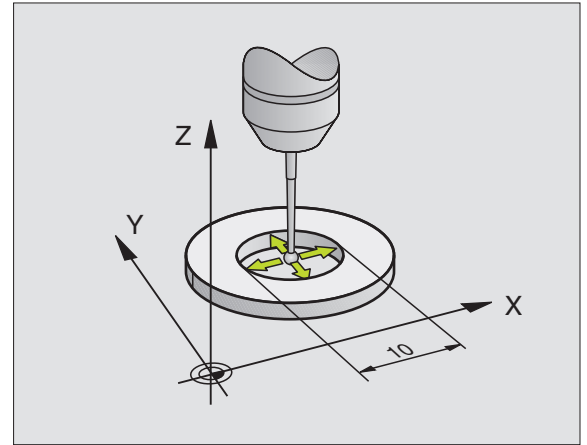
Displaying calibration values

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.

Storing calibration values in the TOOL.T tool table

☞ This function is only available if bit 0 in machine parameter 7411 = 1 is set (activate touch probe data with **TOOL CALL**), and tool table TOOL.T is active (machine parameter 7260 not equal to 0).

If you conduct measurements during program run, the compensation data for the touch probe can be activated from the tool table via a **TOOL CALL.** To store the calibration data in the TOOL.T tool table, enter the tool number in the calibration menu (confirm with ENT) and then press the ENTER R IN TOOL TABLE or the ENTER L IN TOOL TABLE soft key.

Managing more than one block of calibrating data (as of NC software 280 476-xx)

To be able to use more than one block of calibration data, you must set bit one in machine parameter 7411. The calibration data (length, radius, center misalignment, and spindle angle) are then always saved by the TNC in the tool table TOOL.T under a tool number that can be selected in the calibration menu (see also User's Manual, section 5.2, "Tool Data").

☞ If you use this function, you must first activate the corresponding tool number with a tool call before executing a touch probe cycle, regardless of whether you wish to run the touch probe cycle in automatic mode or manual mode.

You can view and edit the calibration data in the calibration menu, but you must make sure to write the changes back into the tool table by pressing the ENTER R IN TOOL TABLE or ENTER L IN TOOL TABLE soft key. The TNC does not write the calibration values into the table automatically!

```
Manual operation                          Programming
                                          and editing




Radius ring gauge =              15.001
Effective probe radius =         13.3136
Styl.tip center offset    X=+0
Styl.tip center offset    Y=+0
Tool number =                    0

                              0%  S-IST  17:11
                              3%  S-MOM  LIMIT 1
 X      -46.252  Y     -45.224  Z       -24.447
+B       -0.477 +c     +8.439
                                   S    257.123
ACTL.           T 5    Z S 1300    F 0        M 5/9

 X+    X-    Y+    Y-         ENTER R   PRINT   END
                             IN TOOL
                              TABLE
```

2.3 Calibrating a Measuring Touch Probe

Introduction

☞ If the TNC displays the error message "Stylus already in contact," select the 3-D calibration menu and press the RESET 3D soft key.

The measuring touch probe must be calibrated whenever the machine parameters for 3-D touch probes are changed.

The effective length is calibrated in the same way as with triggering touch probes. You must also enter tool radius R2 (corner radius).

With MP6321 you can define whether the TNC should probe to find the stylus center.

The 3-D calibration cycle for measuring touch probes enables you to measure a standard ring gauge fully automatically. (The standard ring gauge is available from HEIDENHAIN). Fix the standard ring gauge to the machine table with fixing clamps.

From the data measured during calibration, the TNC calculates the spring rate of the touch probe, the stylus deflection and the stylus center misalignment. At the end of the calibration cycle, the TNC automatically stores these values in the input menu.

Course of actions

▶ In the Manual Operation mode, position the touch probe to a position approximately in the center of the standard ring gauge and set it to 180°.

CAL 3D
⊚
▶ To select the 3-D calibration cycle, press the 3D CAL soft key

▶ Enter the values for stylus radius 1 and stylus radius 2. Enter the same value for stylus radius 1 and 2 if you are using a stylus with ball tip. Enter different values for stylus radius 1 and 2 if you are using a stylus with a corner radius.

▶ Diameter ring gauge: The diameter is engraved on the standard ring gauge.

▶ To start the calibration cycle, press the machine START button: The touch probe measures the standard ring gauge in a programmed sequence of steps.

▶ Rotate the touch probe to 0° as soon as the TNC asks you to.

▶ To start the calibration cycle once again to determine center misalignment, press the machine START button. The touch probe again measures the standard ring gauge in a programmed sequence of steps.

Displaying calibration values

The compensation factors and force ratios are stored in the TNC for later use whenever the measuring touch probe is needed.

You can display the stored values on the screen by pressing the 3D CAL soft key.

Storing calibration values in the TOOL.T tool table

> ☞ This function is only available if machine parameter 7411 = 1 is set (activate touch probe data with **TOOL CALL**), and tool table TOOL.T is active (machine parameter 7260 not equal to 0).

If you conduct measurements during program run, the compensation data for the touch probe can be activated from the tool table via a **TOOL CALL.** To store the calibration data in the TOOL.T tool table, enter the tool number in the calibration menu (confirm with ENT) and then press the ENTER R IN TOOL TABLE soft key.

The TNC stores the stylus radius 1 in the R column, and the stylus radius 2 in the R2 column.

2.4 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.
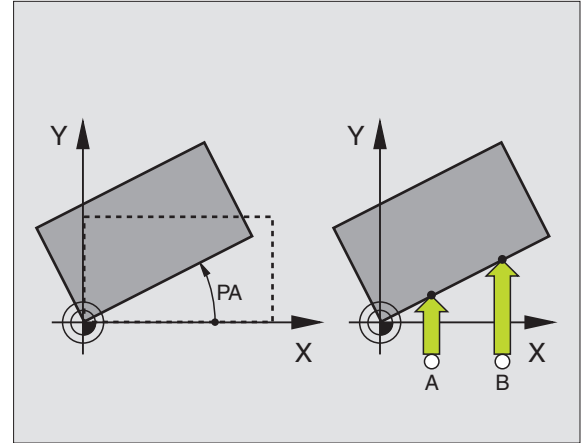
Measuring the basic rotation

PROBING
ROT

▶ Select the probing 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 saves the basic rotation in non-volatile memory. The basic rotation is effective for all subsequent program runs and graphic simulation.

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.

Cancel a basic rotation

▶ Select the probing function by pressing the PROBING ROT soft key.
▶ Enter a rotation angle of zero and confirm with the ENT key.
▶ To terminate the probe function, press the END key.

```
Manual operation                    Programming
                                    and editing




Rotation angle =                    +12.357

                              0% S-IST 7:39
                              1% S-MOM LIMIT 1
+X      +6.278 +Y      +0.809 +Z       -95.962
+B      -2.887 +C     +357.479
                              S     0.034
ACTL.        T 0     Z S 150     F 0        M 5/9

 X+    X-    Y+    Y-              PRINT   END
```

2.5 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

To set the datum in any axis (see figure at right)

PROBING POS

▶ Select the probing 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 ENT.
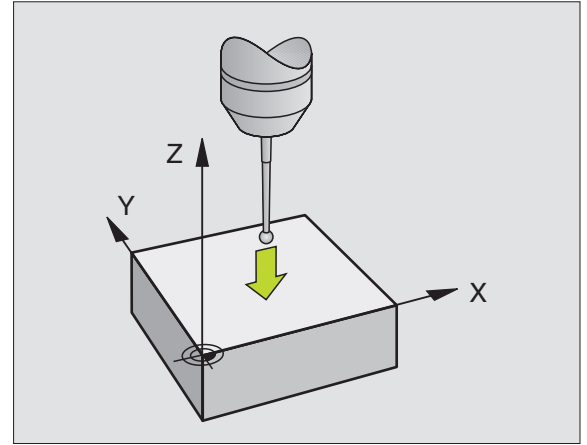
Corner as datum—using points that were already probed for a basic rotation (see figure at right)

PROBING
P

▶ To select the probe function, press ANTASTEN P.

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

▶ 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 with a 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 datum coordinates into the menu window, and confirm your entry with the ENT key.

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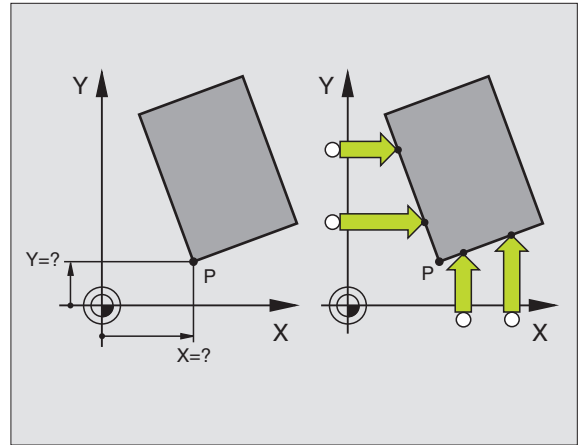
Corner as datum—without using points that were already probed for a basic rotation

▶ To select the probe function, press PROBING P.

▶ 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.

▶ Enter the coordinates of the datum and confirm your entry with ENT.

▶ 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.

▶ To select the probe function, press ANTASTEN CC.

▶ 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: Enter both circle center coordinates into the menu window, and confirm your entry with ENT.

▶ 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 with a 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.

▶ Enter the coordinates of the datum and confirm your entry with ENT.

After the probing procedure is completed, the TNC displays the coordinates of the circle center and the circle radius PR.
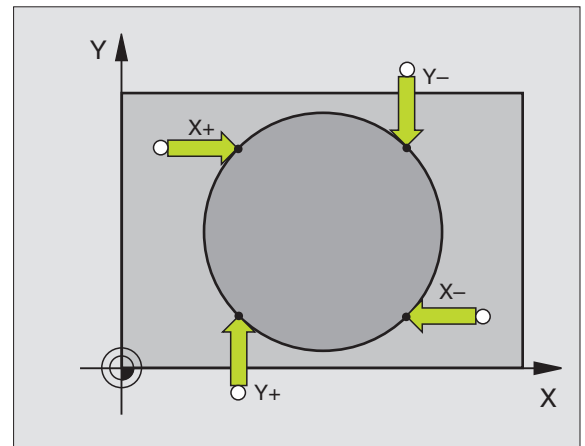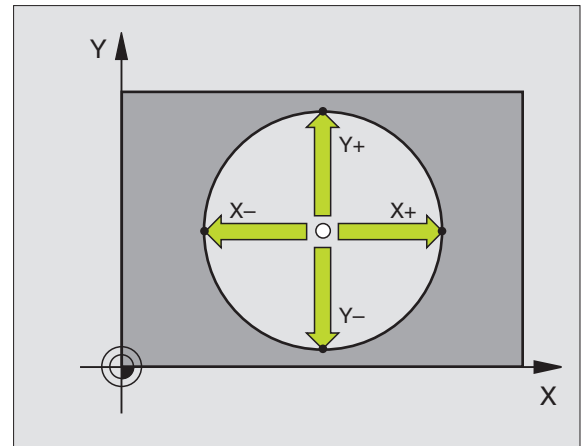
Setting datum points using holes/cylindrical studs

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

Define whether a hole or stud is to be probed

TOUCH
PROBE

▶ Select the probing functions with the TOUCH PROBE, shift the soft-key row.

PROBING
ROT

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

▶ Select holes or cylindrical studs: the selected element appears in a box

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 and have the TNC repeat the probing procedure until all the holes have been probed to set datums.

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

```
Manual operation                    Programming
                                    and editing




                              0% S-IST  6:50
                              1% S-MOM  LIMIT 1
+X      +6.277 +Y      +0.809 +Z     -95.962
+B      -2.887 +C    +357.479
                                S    0.034
ACTL.        T 0     Z S 150        F 0        M 5/9
        PROBING      PROBING PROBING           END
          ROT           P      CC
```
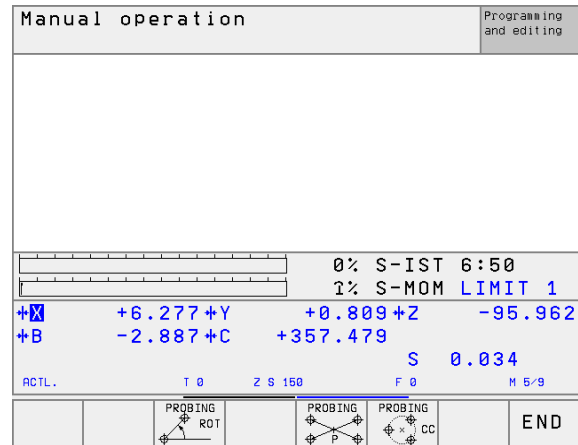
2.6 Measuring Workpieces with a 3-D Touch Probe

Introduction

You can also use the touch probe in the Manual and Electronic Handwheel operating modes to make simple measurements on the workpiece. 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 probing 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 datum.

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 21. The TNC displays the coordinates of the probed corner as datum.

Measuring workpiece dimensions

PROBING
POS

▶ Select the probing 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 with a 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 probing 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 probing function by pressing the PROBING POS soft key.

▶ Probe the first touch point again.

▶ Set the datum to the value that you wrote down previously.

▶ To terminate the dialog, press the END key.

Measuring angles

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

■ the angle between the angle reference axis and a workpiece side, or

■ the angle between two sides.

The measured angle is displayed as a value of maximum 90°.
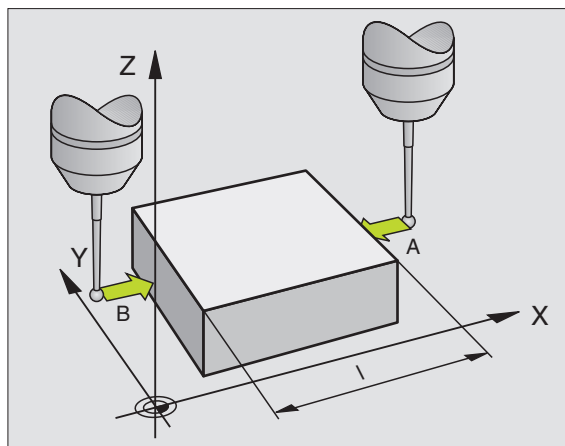
To find the angle between the angle reference axis and a side of the workpiece

▶ Select the probing 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 18).

▶ 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 wrote down previously.
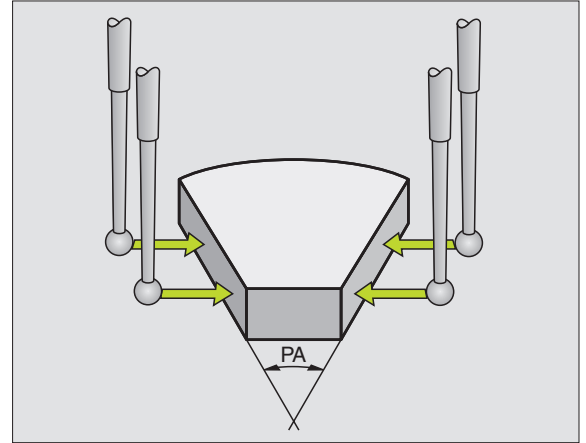
To measure the angle between two workpiece sides:

▶ Select the probing 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 18).

▶ Probe the second side as for a basic rotation, but do not set the rotation angle to zero!

▶ Press the PROBING ROT soft key to display the angle PA between the two sides as the rotation angle.

▶ Cancel the basic rotation, or restore the previous basic rotation by setting the rotation angle to the value that you wrote down previously.

3
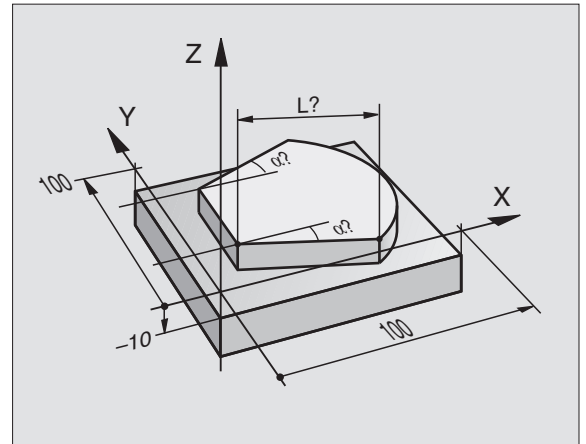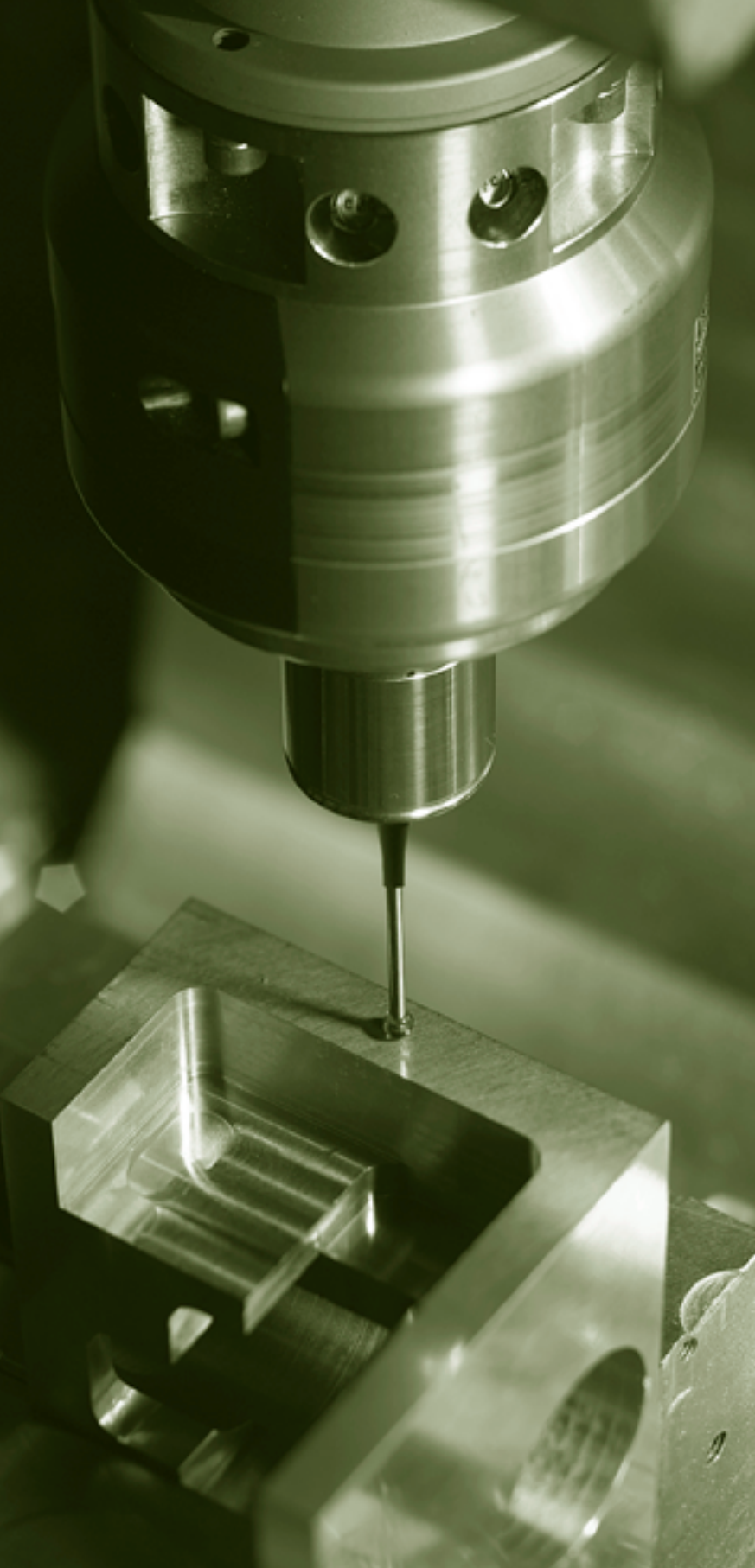
Touch Probe Cycles for
Automatic Workpiece
Inspection

3.1 Measuring Workpiece Misalignment

Overview

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

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

Characteristics common to all touch probe cycles for measuring workpiece misalignment

For the 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 *a* (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**.
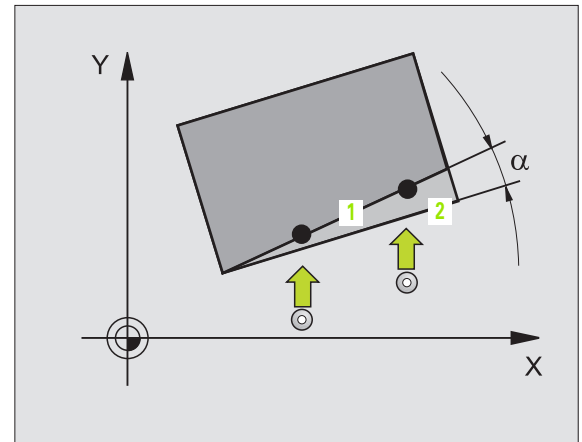
BASIC ROTATION (touch probe cycle 400, ISO: G400)

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 "Compensating Workpiece Misalignment" on page 18).

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 7) 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.

▶ **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

▶ **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 measured value and the angle of the reference line for the basic rotation.

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	;TRAVERSE TO CLEAR HEIGHT
	Q307=+0	;PRESET BASIC ROTATION

BASIC ROTATION from two holes (touch probe cycle 401, 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 "Compensating Workpiece Misalignment" on page 18).

1 Following the positioning logic (see "Running touch probe cycles" on page 7), 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.
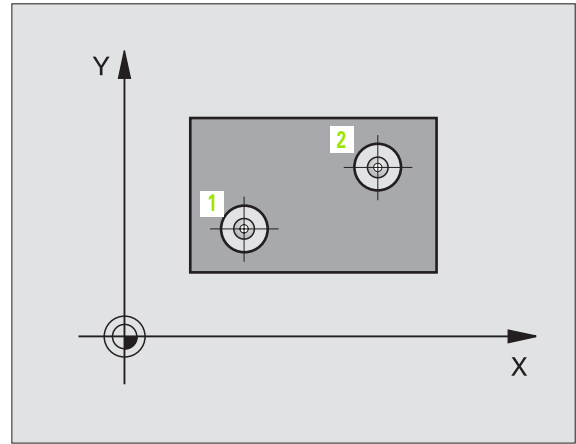
▶ **First hole: Center in 1st axis** Q268 (absolute): center of the first hole in the reference axis of the working plane.

▶ **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 measured value and the angle of the reference line for the basic rotation.
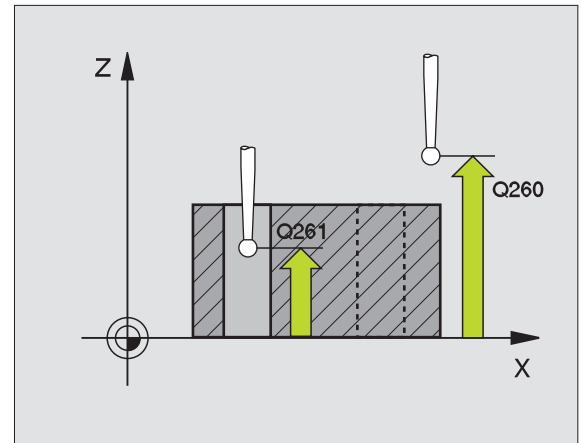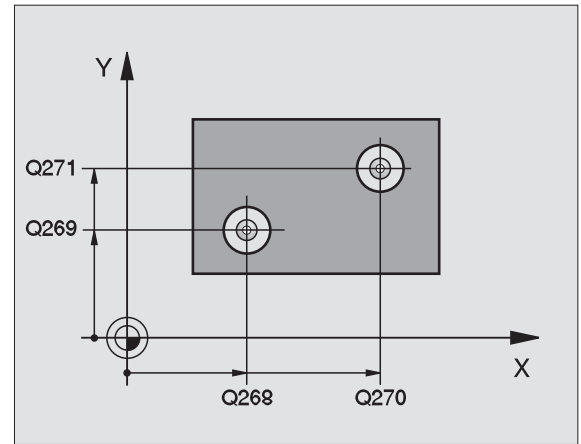
Example: NC blocks

5 TCH PROBE 401 ROT OF 2 HOLES	
Q268=-37	;1ST CENTER 1ST AXIS
Q269=+12	;1ST CENTER 2ND AXIS
Q270=+75	;2ND CENTER 1ST AXIS
Q271=+20	;2ND CENTER 2ND AXIS
Q261=-5	;MEASURING HEIGHT
Q260=+20	;CLEARANCE HEIGHT
Q307=+0	;PRESET BASIC ROT.

BASIC ROTATION over two studs (touch probe cycle 402, 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 "Compensating Workpiece Misalignment" on page 18).
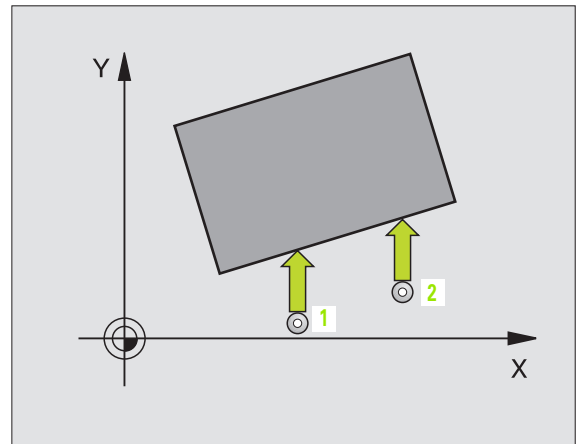
1 Following the positioning logic (see "Running touch probe cycles" on page 7), the TNC positions the touch probe in rapid traverse (value from MP6150 or MP6361) to the starting point for probing the first stud **1**.

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.
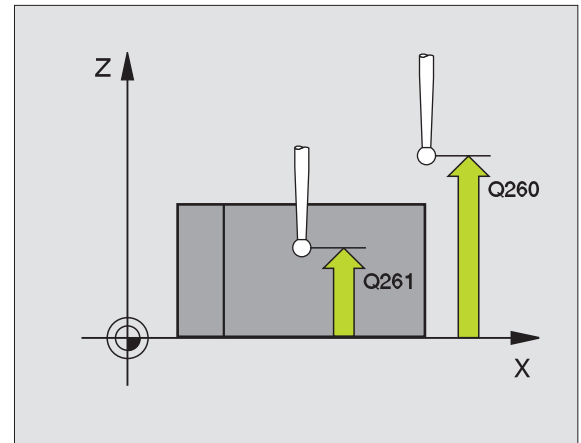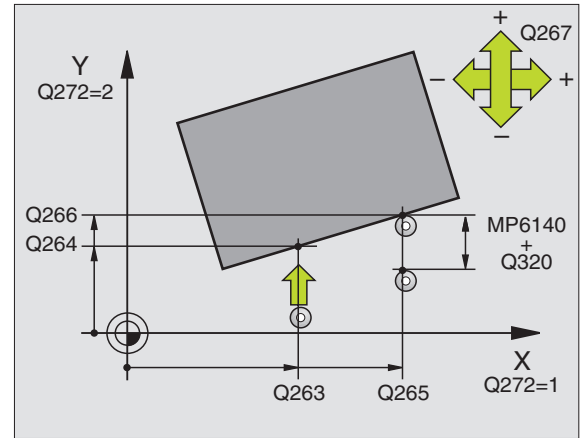
> ☞ **Before programming, note the following:**
>
> Before a cycle definition you must have programmed a tool call to define the touch probe axis.
>
> The TNC will reset an active basic rotation at the beginning of the cycle.
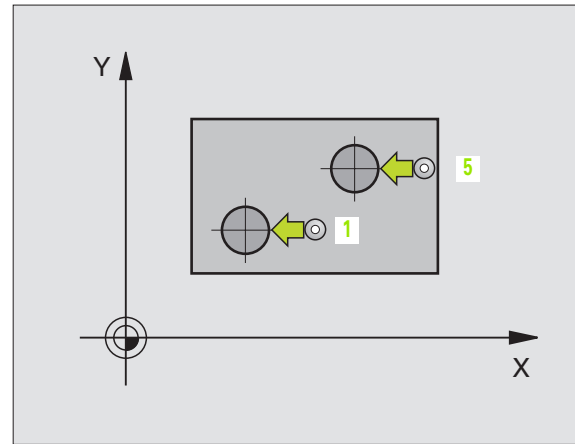
▶ **First stud: Center in 1st axis** Q268 (absolute): center of the first stud in the reference axis of the working plane.

▶ **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.

▶ **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 measured value and the angle of the reference line for the basic rotation.

Example: NC blocks

5 TCH PROBE 402 ROT OF 2 STUDS	
Q268=-37	;1ST CENTER 1ST AXIS
Q269=+12	;1ST CENTER 2ND AXIS
Q313=60	;DIAMETER OF STUD 1
Q261=-5	;MEASURING HEIGHT 1
Q270=+75	;2ND CENTER 1ST AXIS
Q271=+20	;2ND CENTER 2ND AXIS
Q314=60	;DIAMETER STUD 2
Q315=-5	;MEASURING HEIGHT 2
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;TRAVERSE TO CLEAR HEIGHT
Q307=+0	;PRESET BASIC ROT.

BASIC ROTATION compensation via rotary axis (touch probe cycle 403, 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 misalignment by rotating the A, B or C axis. The workpiece can be clamped in any position on the rotary 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 7) 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.

☞ **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 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
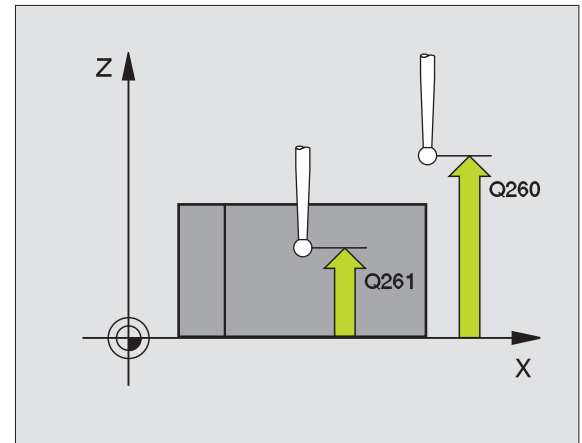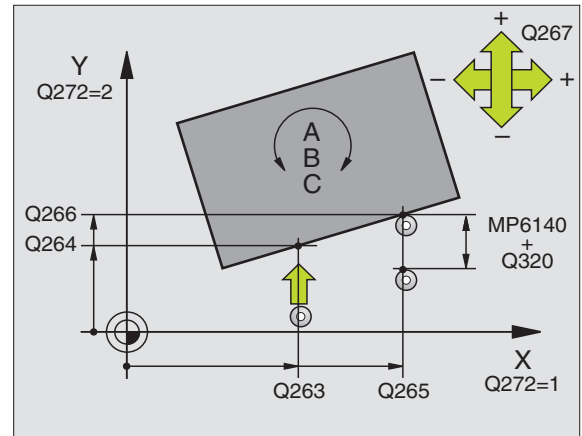
Example: NC blocks

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	;TRAVERSE TO CLEAR HEIGHT
Q312=6	;COMPENSATION AXIS

SET BASIC ROTATION (touch probe cycle 404, ISO: G404, available as of NC software 280 474-xx)

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
   307=+0    ;PRESET BASIC ROTATION
```

Compensating workpiece misalignment by rotating the C axis (touch probe cycle 405, ISO: G405, available as of NC software 280 474-xx)

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 misalignment 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 conduct 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 7) 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 ROT

- ▶ **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 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 misalignment 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.
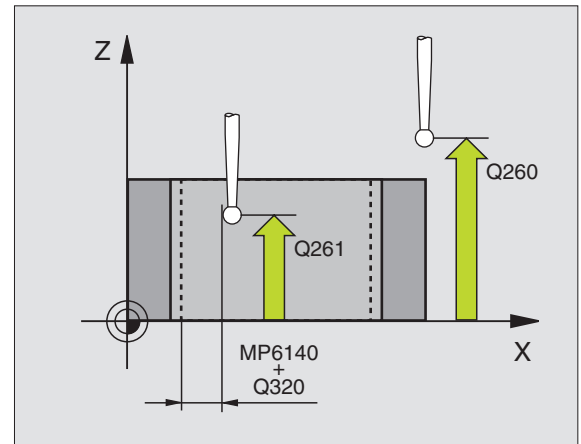
Example: NC blocks

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

Example: Determining a basic rotation from two holes
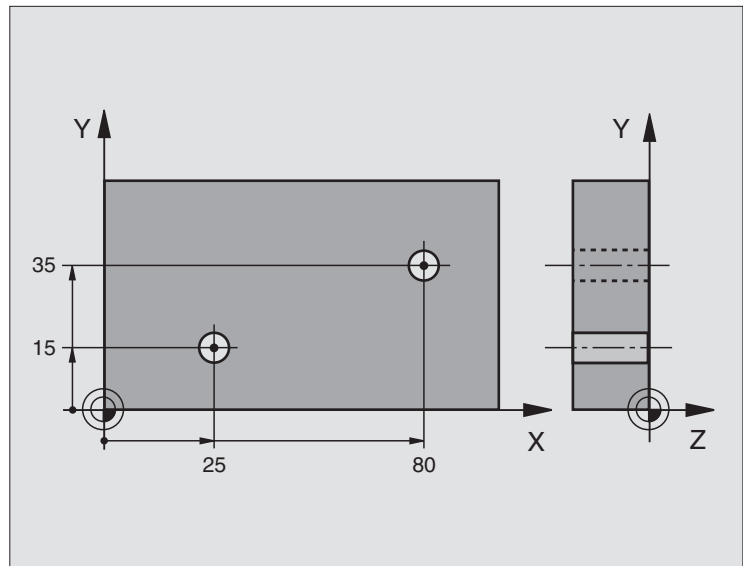
0 BEGIN PGM CYC401 MM	
1 TOOL CALL 0 Z	
2 TCH PROBE 401 ROT OF 2 HOLES	
Q268=+25 ;1ST CENTER 1ST AXIS	Center of the 1st hole: X coordinate
Q269=+15 ;1ST CENTER 2ND AXIS	Center of the 1st hole: Y coordinate
Q270=+80 ;2ND CENTER 1ST AXIS	Center of the 2nd hole: X coordinate
Q271=+35 ;2ND CENTER 2ND AXIS	Center of the 2nd hole: Y coordinate
Q261=-5 ;MEASURING HEIGHT	Coordinate in the touch probe axis in which the measurement is made
Q260=+20 ;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q307=+0 ;PRESET BASIC ROTATION	Angle of the reference line
3 CALL PGM 35K47	Part program call
4 END PGM CYC401 MM	

3.2 Automatic Datum Setting

Overview

The TNC provides nine cycles for automatic datum setting or automatic entry of the measured values into the active datum table:

Cycle	Soft key
410 DATUM INSIDE RECTAN. Measuring the inside length and width of a rectangle, and defining the center as datum	
411 DATUM OUTSIDE RECTAN. Measuring the outside length and width of a rectangle, and defining the center as datum	
412 DATUM INSIDE CIRCLE Measuring any four points on the inside of a circle, and defining the center as datum	
413 DATUM OUTSIDE CIRCLE Measuring any four points on the outside of a circle, and defining the center as datum	
414 DATUM OUTSIDE CORNER Measuring two lines from the outside of the angle, and defining the intersection as datum	
415 DATUM INSIDE CORNER Measuring two lines from within the angle, and defining the intersection as datum	
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	
417 DATUM IN TS AXIS (2nd soft-key level) Measuring any position in the touch probe axis and defining it as datum	
418 DATUM FROM 4 HOLES (2nd soft-key level) Measuring 4 holes crosswise and defining the intersection of the lines between them as datum	

Characteristics common to all touch probe cycles for datum setting

> In TNCs with NC software 280 476-xx you may also run the touch probe cycles 410 to 418 during an active basic rotation (basic rotation or Cycle 10). In previous versions the TNC displays an error message if the basic rotation is active.

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

Writing the calculated datum to a datum table

In all cycles for datum setting you can use the input parameter Q305 to define whether you wish the TNC to set the calculated datum shown in the display or enter it in a datum table.

> If you want the calculated datum to be entered in a datum table, you must first activate a datum table (status M) in a program run mode before starting the measuring program.
>
> When writing to a datum table, the TNC observes machine parameter 7475:
> **MP7475 = 0:** Values are referenced to workpiece datum,
> **MP7475 = 1:** Values are referenced to the machine datum.
>
> If you change the setting in MP7475, the TNC does not convert the values already saved in datum tables.

DATUM FROM INSIDE OF RECTANGLE
(touch probe cycle 410, 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.

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 7) 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 sets the datum in the pocket center, or enters the coordinates of the pocket center in the active datum table.

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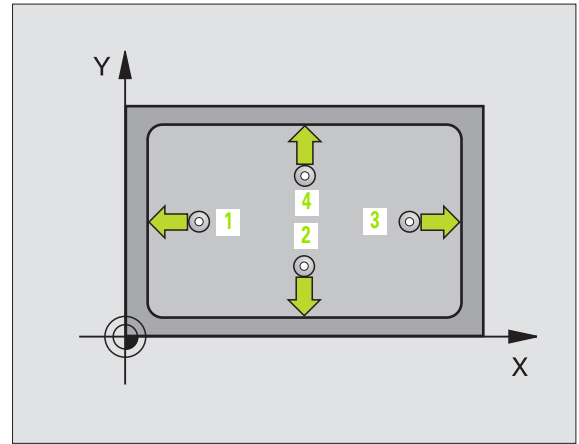
▶ **Center in 1st axis** Q321 (absolute value): Center of the pocket in the reference axis of the working plane.

▶ **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 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 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
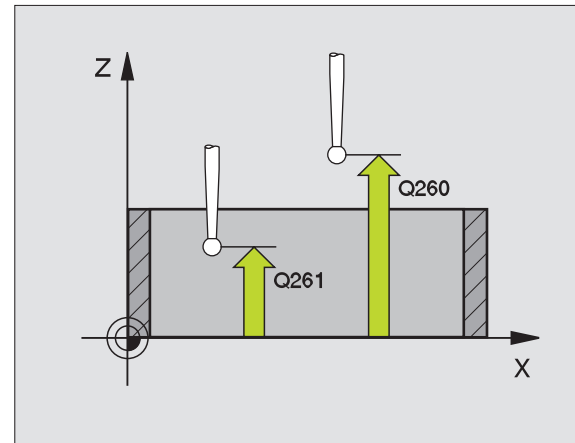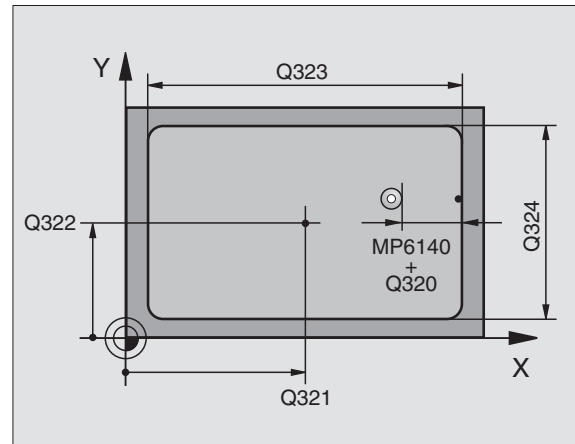
Example: NC blocks

5 TCH PROBE 410 DATUM INSIDE RECTAN.	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q323=60	;1ST SIDE LENGTH
Q324=20	;2ND SIDE-LENGTH
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;TRAVERSE TO CLEAR HEIGHT
Q305=10	;NO. IN TABLE
Q331=+0	;DATUM
Q332=+0	;DATUM

DATUM FROM OUTSIDE OF RECTANGLE
(touch probe cycle 411, 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.

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 7) 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 sets the datum at the measured center, or enters its coordinates in the active datum table.


☝ **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.

▶ **First side length** Q323 (incremental value): Stud length, parallel to the reference axis of the working plane

▶ **Second 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 stud 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

Example: NC blocks

5 TCH PROBE 411 DATUM OUTS. RECTAN.	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q323=60	;1ST SIDE LENGTH
Q324=20	;2ND SIDE-LENGTH
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+20	;CLEARANCE HEIGHT
Q301=0	;TRAVERSE TO CLEAR HEIGHT
Q305=0	;NO. IN TABLE
Q331=+0	;DATUM
Q332=+0	;DATUM

DATUM FROM INSIDE OF CIRCLE (touch probe cycle 412, 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.

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 7) 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 sets the datum at the measured center, or enters its coordinates in the active datum table.

> 👆 **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 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 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
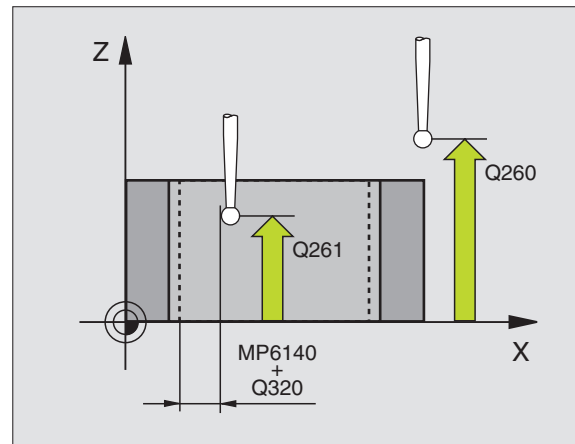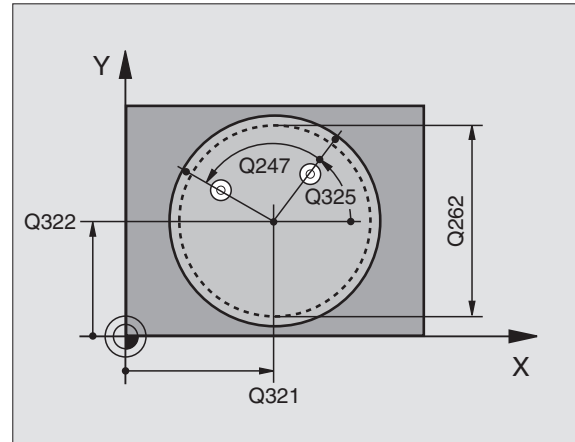
Example: NC blocks

5 TCH PROBE 412 DATUM INSIDE CIRCLE	
Q321=+50	;CENTER IN 1ST AXIS
Q322=+50	;CENTER IN 2ND AXIS
Q262=65	;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	;TRAVERSE TO CLEAR HEIGHT
Q305=12	;NO. IN TABLE
Q331=+0	;DATUM
Q332=+0	;DATUM

DATUM FROM OUTSIDE OF CIRCLE
(touch probe cycle 413, 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.
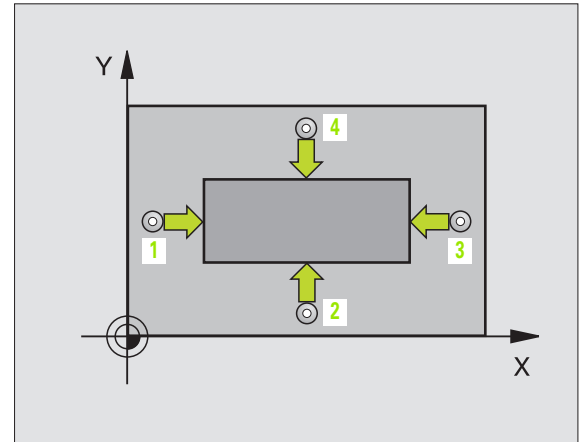
1   The TNC positions the touch probe to the starting points at rapid traverse (value from MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 7) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety 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 sets the datum at the measured center, or enters its coordinates in the active datum table.
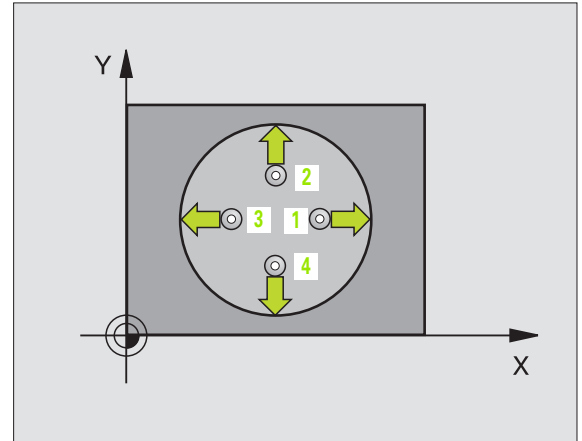

> ☞   **Before programming, note the following:**
>
> To prevent a collision between the touch probe and the workpiece, enter a **high** estimate for the nominal diameter of the pocket (or hole).
>
> Before a cycle definition you must have programmed a tool call to define the touch probe axis.

▸ **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 stud 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

Example: NC blocks

5	TCH PROBE 413 DATUM OUTSIDE CIRCLE	
	Q321=+50	;CENTER IN 1ST AXIS
	Q322=+50	;CENTER IN 2ND AXIS
	Q262=65	;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	;TRAVERSE TO CLEAR HEIGHT
	Q305=15	;NO. IN TABLE
	Q331=+0	;DATUM
	Q332=+0	;DATUM

DATUM FROM OUTSIDE OF CORNER
(touch probe cycle 414, 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.

1 Following the positioning logic (see "Running touch probe cycles" on page 7), 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 sets the datum at the intersection of the measured lines, or enters its coordinates in the active datum table.

☞ **Before programming, note the following:**

By defining the position 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.
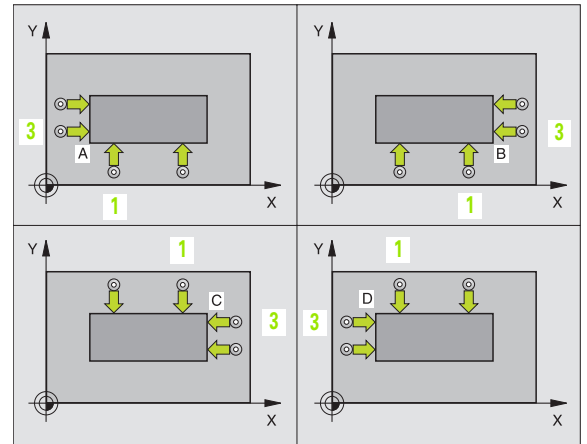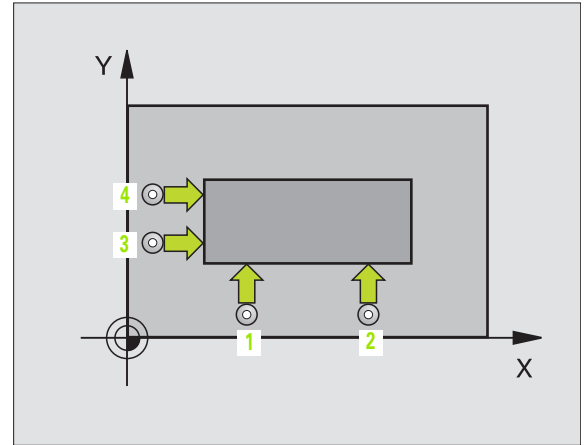
Corner	Condition X	Condition Y
A	X1 greater than X3	Y1 less than Y3
B	X1 less than X3	Y1 less than Y3
C	X1 less than X3	Y1 greater than Y3
D	X1 greater than X3	Y1 greater than Y3

▶ **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.

▶ **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 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 corner. Basic setting = 0

Example: NC blocks

5	TCH PROBE 414 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	;TRAVERSE TO CLEAR HEIGHT
	Q304=0	;BASIC ROTATION
	Q305=7	;NO. IN TABLE
	Q331=+0	;DATUM
	Q332=+0	;DATUM

DATUM FROM INSIDE OF CORNER (touch probe cycle 415, 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.

1 Following the positioning logic (see "Running touch probe cycles" on page 7), 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 sets the datum at the intersection of the measured lines, or enters its coordinates in the active datum table.

☞ **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.

▶ **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

▶ **Datum number in table** Q305: Enter the datum number in the 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 corner. 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
	Q327=45	;SPACING IN 2ND AXIS
	Q308=3	;CORNER
	Q261=-5	;MEASURING HEIGHT
	Q320=0	;SET-UP CLEARANCE
	Q260=+20	;CLEARANCE HEIGHT
	Q301=0	;TRAVERSE TO CLEAR HEIGHT
	Q304=0	;BASIC ROTATION
	Q305=8	;NO. IN TABLE
	Q331=+0	;DATUM
	Q332=+0	;DATUM

DATUM CIRCLE CENTER (touch probe cycle 416, 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.

1 Following the positioning logic (see "Running touch probe cycles" on page 7), 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 sets the datum at the measured center of the bolt hole circle, or enters its coordinates in the active datum table.

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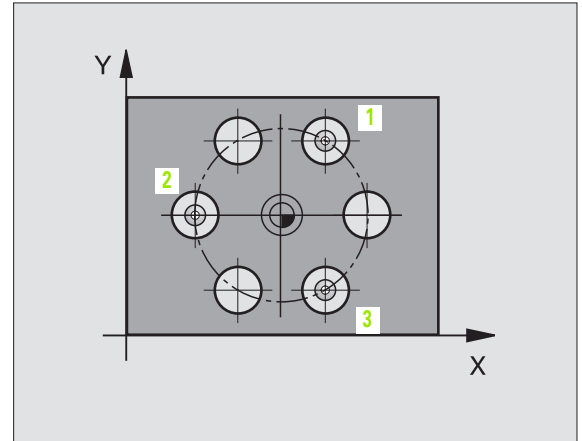
▶ **Center of 1st axis** Q273 (absolute): bolt hole circle center (nominal value) in the reference axis of the working plane.

▶ **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.

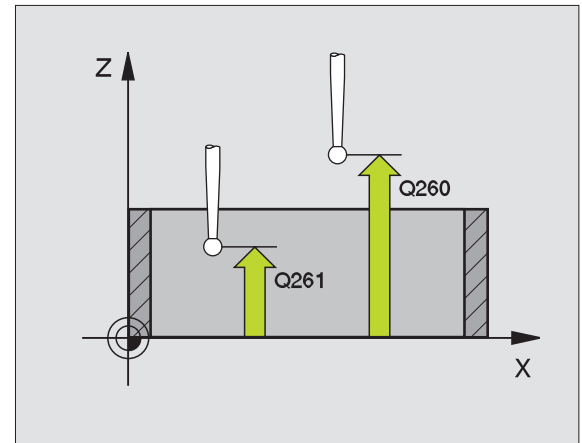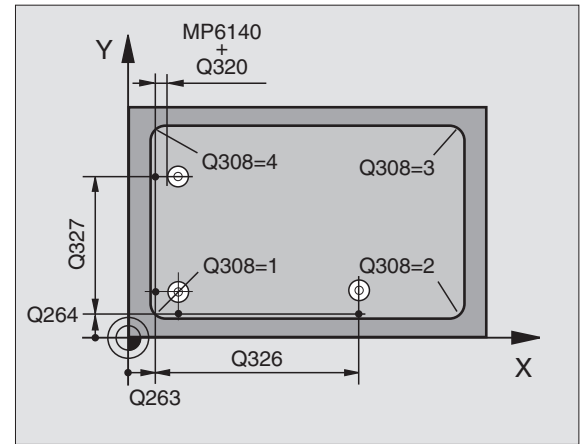▶ **Angle of 2nd hole** Q292 (absolute): polar coordinate angle of the second hole center in the working plane.

▶ **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 datum number in the 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
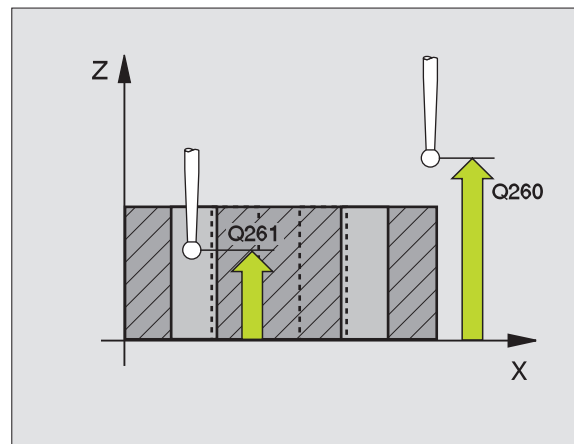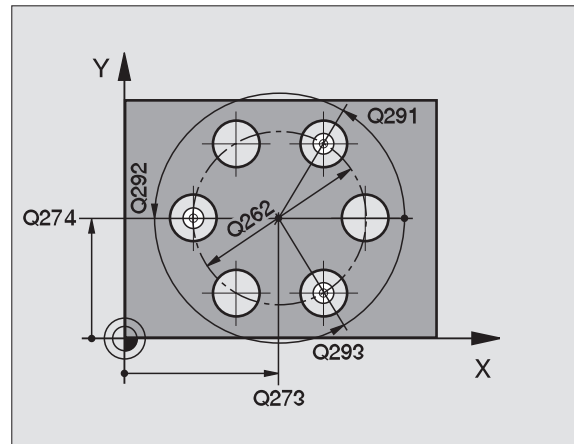
Example: NC blocks

5 TCH PROBE 416 DATUM CIRCLE CENTER	
Q273=+50	;CENTER IN 1ST AXIS
Q274=+50	;CENTER IN 2ND AXIS
Q262=90	;NOMINAL DIAMETER
Q291=+35	;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

DATUM IN TOUCH PROBE AXIS (touch probe cycle 417, 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 into a datum 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 7) 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 sets the datum in the touch probe axis, or enters its coordinates in the active datum table.

☞ **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.

▶ **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.

▶ **First measuring point in the 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 datum number in the 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
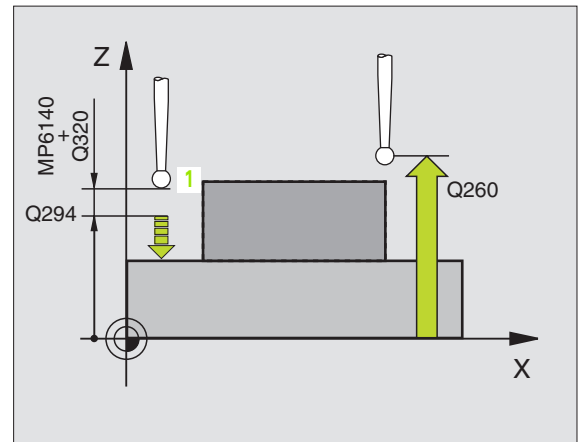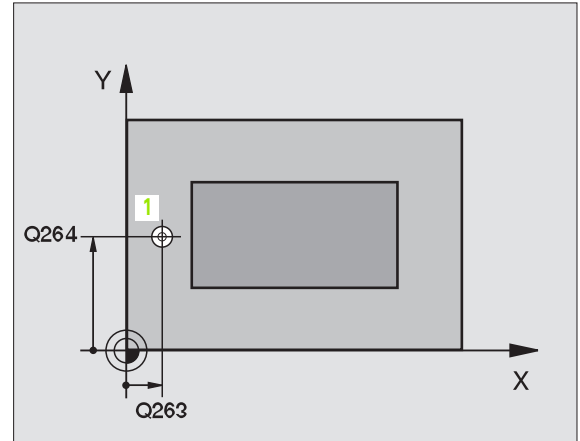
Example: NC blocks

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

DATUM AT CENTER BETWEEN 4 HOLES
(touch probe cycle 418, 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.

1 Following the positioning logic (see "Running touch probe cycles" on page 7), 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 sets the datum at the intersection of the diagonal lines connecting opposite holes **1**/**3** and **2**/**4** or writes the coordinates of the intersection to the active datum table.

👉 **Before programming, note the following:**

Before a cycle definition you must have programmed a tool call to define the touch probe axis.
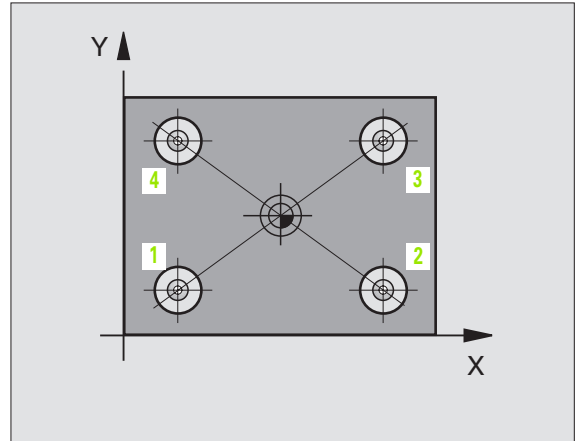
▶ **First center in 1st axis** Q268 (absolute): center of the 1st hole in the reference axis of the working plane.

▶ **First center in 2nd axis** Q269 (absolute): center of the 1st hole in the minor axis of the working plane.

▶ **Second center in 1st axis** Q270 (absolute): center of the 2nd hole in the reference axis of the working plane.

▶ **Second center in 2nd axis** Q271 (absolute): center of the 2nd hole in the minor axis of the working plane.

▶ **Third center in 1st axis** Q316 (absolute): center of the 3rd hole in the reference axis of the working plane.

▶ **Third center in 2nd axis** Q317 (absolute): center of the 3rd hole in the minor axis of the working plane.

▶ **Fourth center in 1st axis** Q318 (absolute): center of the 4th hole in the reference axis of the working plane.

▶ **Fourth center in 2nd axis** Q319 (absolute): center of the 4th hole in the minor axis of the working plane.

▶ **Measuring height in the touch probe axis** Q261 (absolute): coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.

▶ **Clearance height** Q260 (absolute): coordinate in the touch probe axis at which no collision between tool and workpiece (fixtures) can occur.
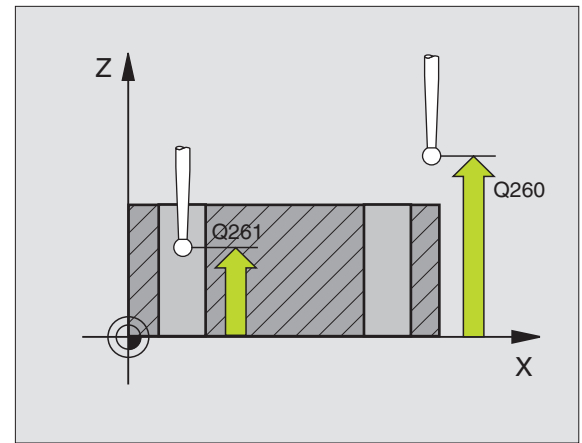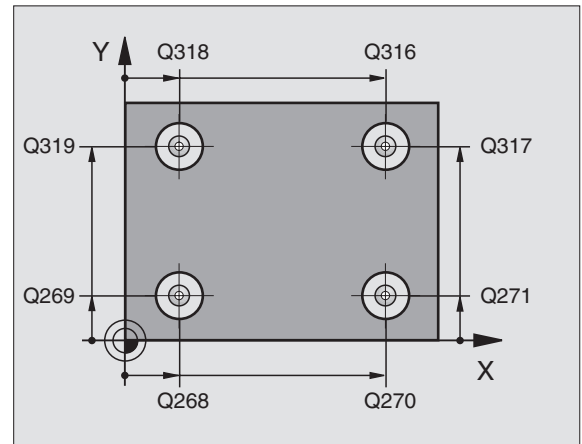
- ▶ **Datum number in table:** Enter the datum number in the table in which the TNC is to save the coordinates of the line intersection. If you enter Q305=0, the TNC automatically sets the display so that the new datum is at the intersection of the connecting lines.

- ▶ **New datum for reference axis** Q331 (absolute): coordinate in the reference axis at which the TNC should set the 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 intersection of the connecting lines. Basic setting = 0

Example: NC blocks

5	TCH PROBE 418 DATUM FROM 4 HOLES	
	Q268=+20	;1ST CENTER 1ST AXIS
	Q269=+25	;1ST CENTER 2ND AXIS
	Q270=+150	;2ND CENTER 1ST AXIS
	Q271=+25	;2ND CENTER 2ND AXIS
	Q316=+150	;3RD CENTER 1ST AXIS
	Q317=+85	;3RD CENTER 2ND AXIS
	Q318=+22	;4TH CENTER 1ST AXIS
	Q319=+80	;4TH CENTER 2ND AXIS
	Q261=-5	;MEASURING HEIGHT
	Q260=+10	;CLEARANCE HEIGHT
	Q305=12	;NO. IN TABLE
	Q331=+0	;DATUM
	Q332=+0	;DATUM

3.2 Automatic Datum Setting

Example: Datum setting on top surface of workpiece and in center of a circular segment
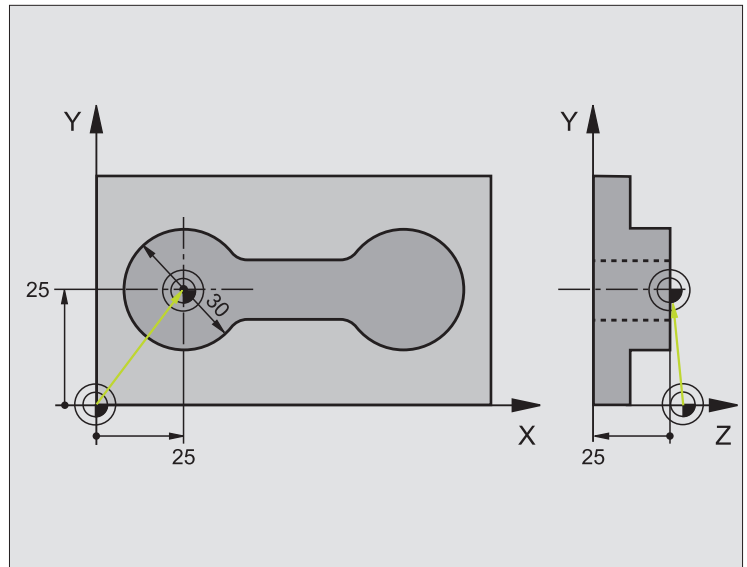
0 BEGIN PGM CYC413 MM	
1 TOOL CALL 0 Z	Call tool 0 to define the touch probe axis
2 TCH PROBE 417 DATUM IN TS AXIS	Cycle definition for datum setting in the touch probe axis
Q263=+25 ;1ST POINT 1ST AXIS	Touch point: X coordinate
Q264=+25 ;1ST POINT 2ND AXIS	Touch point: Y coordinate
Q294=+25 ;1ST POINT 3RD AXIS	Touch point: Z coordinate
Q320=2 ;SET-UP CLEARANCE	Safety clearance in addition to MP6140
Q260=+20 ;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q305=0 ;NO. IN TABLE	Set display
Q333=+0 ;DATUM	Set touch-probe axis to 0

3 TCH PROBE 413 DATUM OUTSIDE CIRCLE	
Q321=+25 ;CENTER IN 1ST AXIS	Center of circle: X coordinate
Q322=+25 ;CENTER IN 2ND AXIS	Center of circle: Y coordinate
Q262=30 ;NOMINAL DIAMETER	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 ;TRAVERSE TO CLEAR HEIGHT	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
Q332=+10 ;DATUM	Set the display in Y to 10
4 CALL PGM 35K47	Part program call
5 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 datum table so that it may be used at a later time.
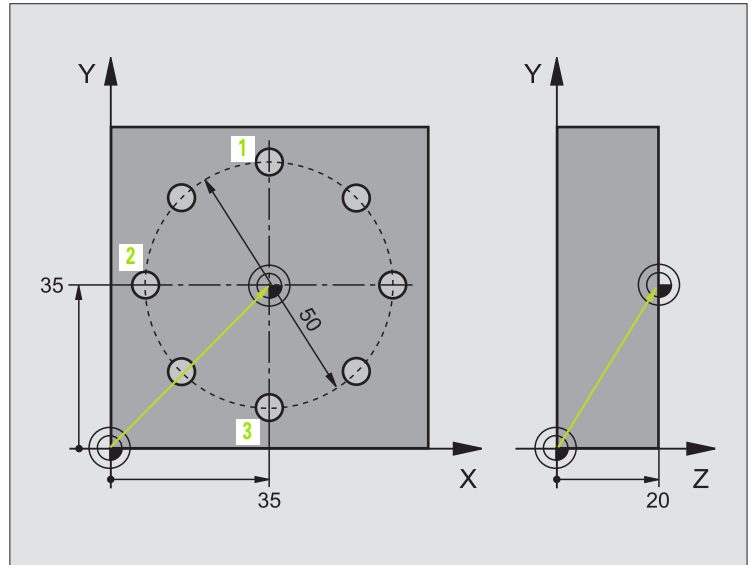
0 BEGIN PGM CYC416 MM	
1 TOOL CALL 0 Z	Call tool 0 to define the touch probe axis
2 TCH PROBE 417 DATUM IN TS AXIS	Cycle definition for datum setting in the touch probe axis
Q263=+7.5 ;1ST POINT 1ST AXIS	Touch point: X coordinate
Q264=+7.5 ;1ST POINT 2ND AXIS	Touch point: Y coordinate
Q294=+25 ;1ST POINT 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	Enter Z coordinate in the datum table
Q333=+0 ;DATUM	Set touch-probe axis to 0

3 TCH PROBE 416 DATUM CIRCLE CENTER	
Q273=+35 ;CENTER 1ST AXIS	Center of the bolt hole circle: X coordinate
Q274=+35 ;CENTER IN 2ND AXIS	Center of the bolt hole circle: Y coordinate
Q262=50 ;NOMINAL DIAMETER	Diameter of the bolt hole circle
Q291=+90 ;ANGLE OF 1ST HOLE	Polar coordinate angle for 1st hole center **1**
Q292=+180 ;ANGLE OF 2ND HOLE	Polar coordinate angle for 2nd hole center **2**
Q293=+270 ;ANGLE OF 3RD HOLE	Polar coordinate angle for 3rd hole center **3**
Q261=+15 ;MEASURING HEIGHT	Coordinate in the touch probe axis in which the measurement is made
Q260=+10 ;CLEARANCE HEIGHT	Height in the touch probe axis at which the probe can traverse without collision
Q305=1 ;NO. IN TABLE	Enter center of bolt hole circle (X and Y) in the datum table
Q331=+0 ;DATUM	
Q332=+0 ;DATUM	
4 CYCL DEF 7.0 DATUM SHIFT	Use Cycle 7 to shift datum to the center of the bolt hole circle
5 CYCL DEF 7.1 #1	
6 CALL PGM 35KL7	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
0 REFERENCE PLANE Measuring a coordinate in a selectable axis	
1 POLAR DATUM PLANE Measuring a point in a probing direction	
420 MEASURE ANGLE Measuring an angle in the working plane	
421 MEASURE HOLE Measuring the position and diameter of a hole	
422 MEAS. CIRCLE OUTSIDE Measuring the position and diameter of a circular stud	
423 MEAS. RECTAN. INSIDE Measuring the position, length and width of a rectangular pocket	
424 MEAS. RECTAN. OUTSIDE Measuring the position, length and width of a rectangular stud	
425 MEASURE INSIDE WIDTH (2nd soft-key level) Measuring slot width	
426 MEASURE RIDGE WIDTH (2nd soft-key level) Measuring the width of a ridge	
427 MEASURE COORDINATE (2nd soft-key level) Measuring any coordinate in a selectable axis	
430 MEAS. BOLT HOLE CIRC. (2nd soft-key level) Measuring position and diameter of a bolt hole circle	
431 MEASURE PLANE (2nd soft-key level) Measuring the A and B axis angles of a plane	

Recording the results of measurement

For all cycles in which you automatically measure workpieces (with the exception of Cycles 0 and 1), the TNC records the results. As standard procedure the TNC 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. 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 423:

*** Measuring Log for Probing Cycle 421 Hole Measuring ***

Date: 29-11-1997
Time: 6:55:04
Measuring program: TNC:\GEH35712\CHECK1.H

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

Given limit values: Maximum limit for center in reference axis: 50.1000 Minimum limit for center in reference axis: 49.9000
Maximum limit for center in minor axis: 65.1000
Minimum limit for center in minor axis: 64.9000
Maximum dimension for hole: 12.0450
Minimum dimension for hole 12.0000

****
Actual values: Center in reference axis: 50.0810
Center in minor axis: 64.9530
Diameter: 12.0259

Deviations: Center in reference axis: 0.0810
Center in minor axis: -0.0470
Diameter: 0.0259

****
Further measuring results: Measuring height: -5.0000

***************** End of measuring log *****************

Measurement results in Q parameters

The 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).

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.

☞ The TNC also sets the status marker if you have defined no tolerance values or largest/smallest dimensions.

Tolerance monitoring

For most of the cycles for workpiece inspection you can have the TNC conduct 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 conduct 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 Q330 not equal to 0).

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 results parameter 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, 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 end input: press the ENT key
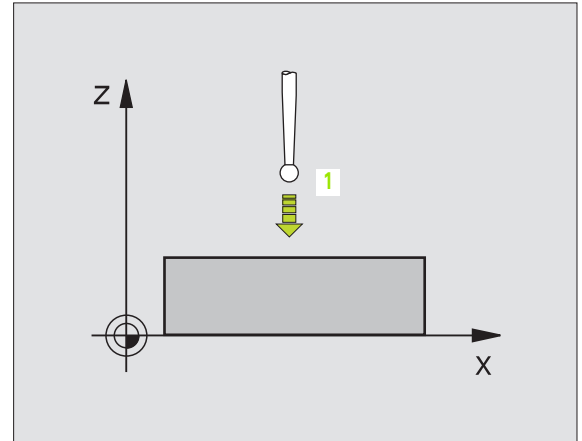
Example: NC blocks

```
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 end input: press the ENT key
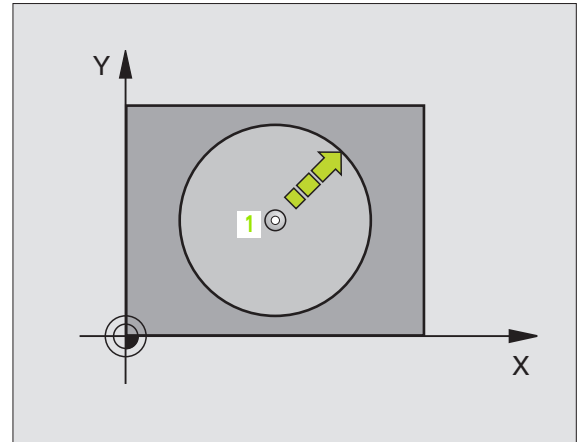
Example: NC blocks

```
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, 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 7) 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.

▶ **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 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.
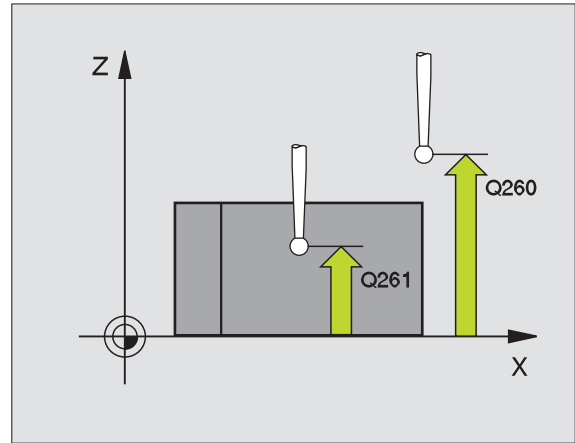
Example: NC blocks

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	;TRAVERSE TO CLEAR HEIGHT
Q281=1	;MEASURING LOG

MEASURE HOLE (touch probe cycle 421, 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 values 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 7) 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:

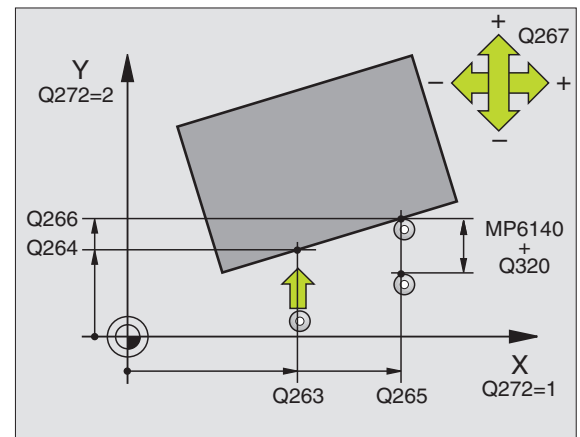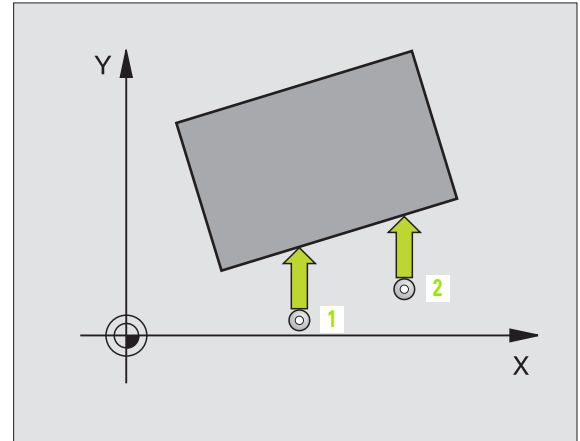| Parameter number	Meaning
Q151	Actual value of center in reference axis
Q152	Actual value of center in minor axis
Q153	Actual value of diameter
Q161	Deviation from center of reference axis
Q162	Deviation from center of minor axis
Q163	Deviation from diameter

☞ **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 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
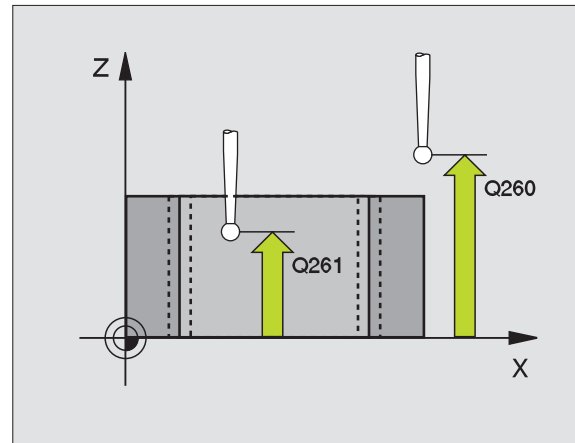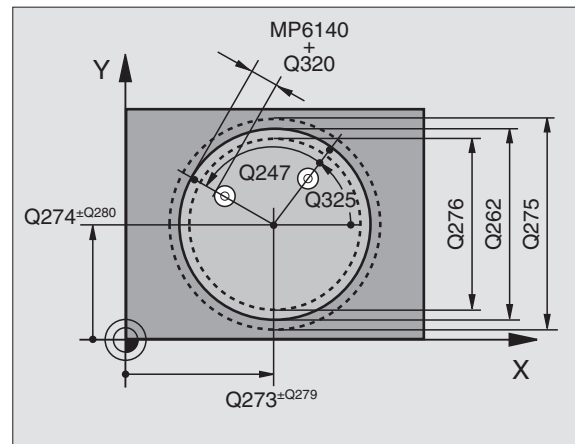
▶ **Measuring log** Q281: definition of whether the TNC is to create a measuring log:
0: No measuring log
1: Generate measuring log: with the standard setting the TNC saves the **log file TCHPR421.TXT** in the directory in which your measuring program is also stored.

▶ **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 71)
0: Monitoring not active
>0: Tool number in the tool table TOOL.T

Example: NC blocks

5	TCH PROBE 421 MEASURE HOLE	
	Q273=+50	;CENTER IN 1ST AXIS
	Q274=+50	;CENTER IN 2ND AXIS
	Q262=75	;NOMINAL DIAMETER
	Q325=+0	;STARTING ANGLE
	Q247=+60	;STEPPING ANGLE
	Q261=-5	;MEASURING HEIGHT
	Q320=0	;SET-UP CLEARANCE
	Q260=+20	;CLEARANCE HEIGHT
	Q301=1	;TRAVERSE TO CLEAR HEIGHT
	Q275=75.12	;MAXIMUM DIMENSION
	Q276=74.95	;MINIMUM DIMENSION
	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 CIRCLE OUTSIDE (touch probe cycle 422, 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 values in system parameters.
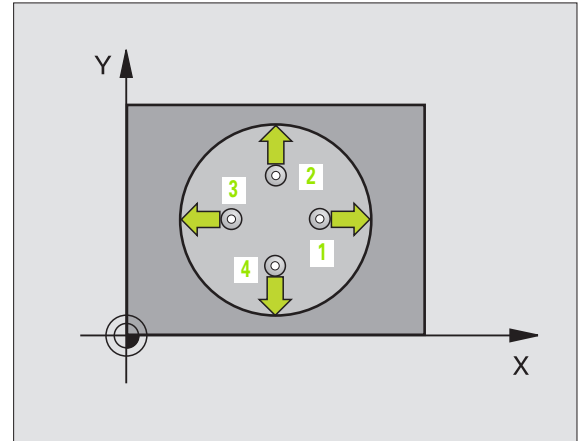
1 The TNC positions the touch probe to the starting points at rapid traverse (value from MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 7) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance 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°.
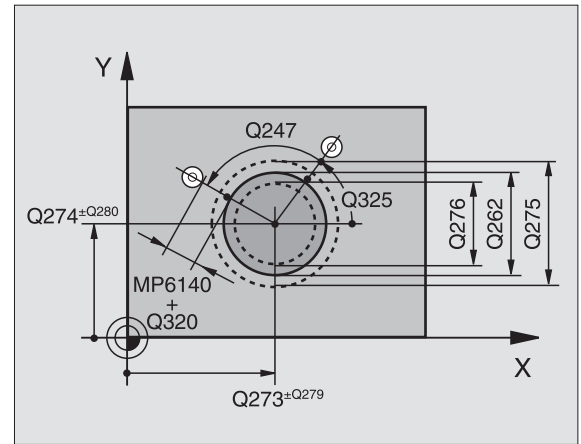
☞ The smaller the angle, the less accurately the TNC can calculate the stud dimensions. Minimum input value: 5°

▶ **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 stud** Q275: maximum permissible dimension for the stud.

▶ **Minimum dimension of size for the stud** Q276: minimum permissible dimension for the stud.

▶ **Tolerance value for center 1st axis** Q279: permissible position deviation in the reference axis of the working plane.

▶ **Tolerance value for center 2nd axis** Q280: permissible position deviation in the minor axis of the working plane
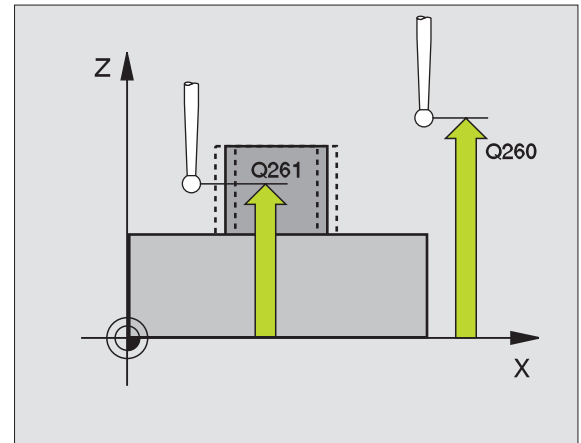
▶ **Measuring log** Q281: definition of whether the TNC is
to create a measuring log:
0: No measuring log
1: Generate measuring log: with the standard setting
the TNC saves the **log file TCHPR422.TXT** in the
directory in which your measuring program is also
stored.

▶ **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 71):
0: Monitoring not active
>0: Tool number in the tool table TOOL.T

Example: NC blocks

```
5 TCH PROBE 422 MEAS. CIRCLE OUTSIDE
     Q273=+20   ;CENTER IN 1ST AXIS
     Q274=+30   ;CENTER IN 2ND AXIS
     Q262=35    ;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     ;TRAVERSE TO CLEAR HEIGHT
     Q275=35.15 ;MAXIMUM DIMENSION
     Q276=34.9  ;MINIMUM DIMENSION
     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
```

3.3 Automatic Workpiece Measurement

MEASURE RECTANGLE FROM INSIDE
(touch probe cycle 423, 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 values in system parameters.
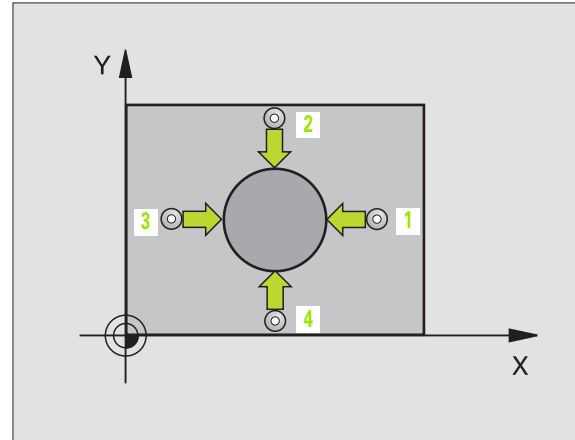
1 The TNC positions the touch probe to the starting points at rapid traverse (value from MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 7) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance 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.
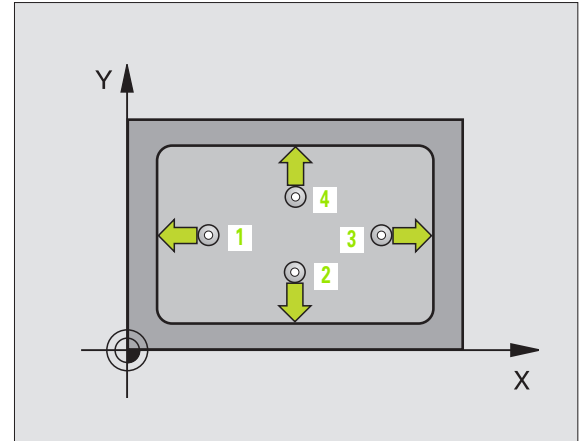
▶ **Center in 1st axis** Q273 (absolute value): Center of the pocket in the reference axis of the working plane.

▶ **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
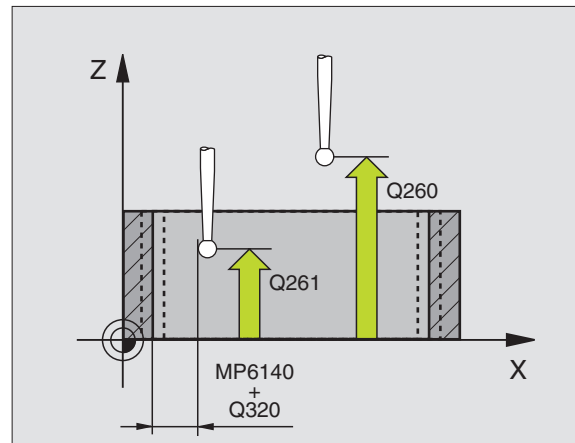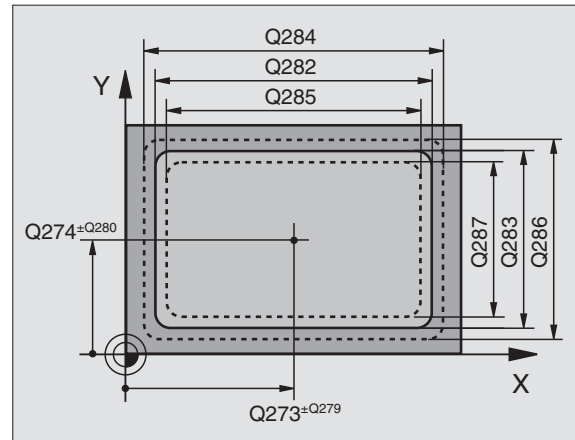
▶ **Measuring log** Q281: definition of whether the TNC is to create a measuring log:
0: No measuring log
1: Generate measuring log: with the standard setting the TNC saves the **log file TCHPR423.TXT** in the directory in which your measuring program is also stored.

▶ **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 71)
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 IN 1ST AXIS
Q274=+50	;CENTER IN 2ND AXIS
Q282=80	;1ST SIDE LENGTH
Q283=60	;2ND SIDE LENGTH
Q261=-5	;MEASURING HEIGHT
Q320=0	;SET-UP CLEARANCE
Q260=+10	;CLEARANCE HEIGHT
Q301=1	;TRAVERSE TO CLEAR HEIGHT
Q284=0	;MAX. LIMIT 1ST SIDE
Q285=0	;MIN. LIMIT 1ST SIDE
Q286=0	;MAX. LIMIT 2ND SIDE
Q287=0	;MIN. LIMIT 2ND SIDE
Q279=0	;TOLERANCE 1ST CENTER
Q280=0	;TOLERANCE 2ND CENTER
Q281=1	;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 values 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 7) 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.

▶ **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

3 Touch Probe Cycles for Automatic Workpiece Inspection
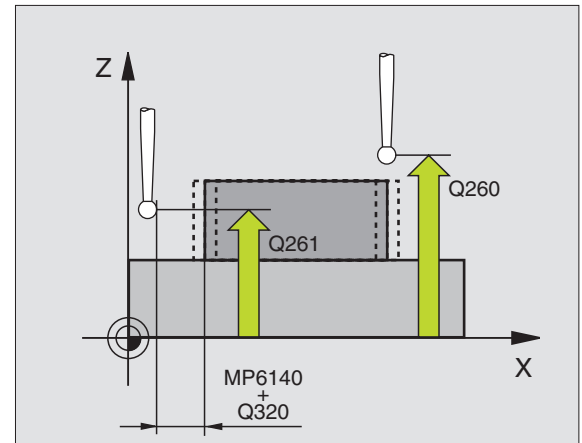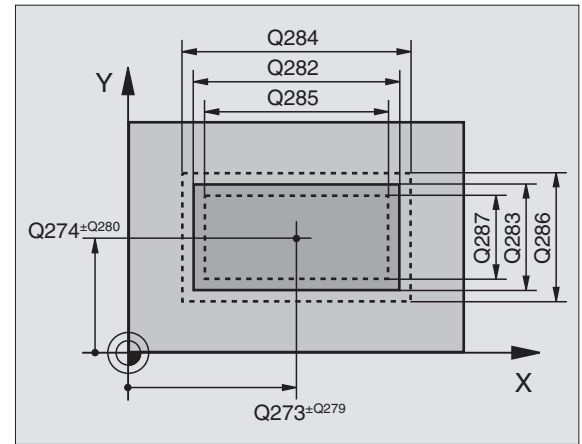
▶ **Measuring log** Q281: definition of whether the TNC is to create a measuring log:
0: No measuring log
1: Generate measuring log: with the standard setting the TNC saves the **log file TCHPR424.TXT** in the directory in which your measuring program is also stored.

▶ **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 71):
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 IN 1ST AXIS
    Q274=+50   ;CENTER IN 2ND AXIS
    Q282=75    ;1ST SIDE LENGTH
    Q283=35    ;2ND SIDE LENGTH
    Q261=-5    ;MEASURING HEIGHT
    Q320=0     ;SET-UP CLEARANCE
    Q260=+20   ;CLEARANCE HEIGHT
    Q301=0     ;TRAVERSE TO CLEAR HEIGHT
    Q284=75.1  ;MAX. LIMIT 1ST SIDE
    Q285=74.9  ;MIN. LIMIT 1ST SIDE
    Q286=35    ;MAX. LIMIT 2ND SIDE
    Q287=34.95 ;MIN. LIMIT 2ND SIDE
    Q279=0.1   ;TOLERANCE 1ST CENTER
    Q280=0.1   ;TOLERANCE 2ND CENTER
    Q281=1     ;MEASURING LOG
    Q309=0     ;PGM-STOP IF ERROR
    Q330=0     ;TOOL NUMBER
```

MEASURE INSIDE WIDTH (touch probe cycle 425, 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.
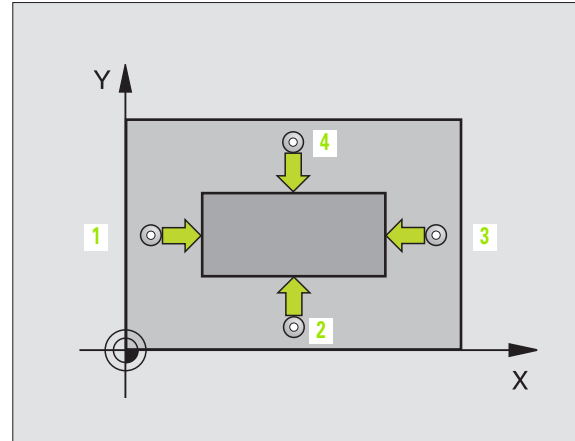
1 The TNC positions the touch probe to the starting points at rapid traverse (value from MP6150 or MP6361) following the positioning logic (see "Running touch probe cycles" on page 7) to the starting point **1**. The TNC calculates the probe starting points from the data in the cycle and the safety clearance 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). 1. The first probing is always in the positive direction of the programmed axis.

3 If you enter an offset for the second measurement, the TNC then moves the touch probe paraxially to the next starting point **2** and probes the second touch point. If you do not enter an offset, the TNC measures the width in the exact opposite direction.

4 Finally the TNC returns the touch probe to the clearance height and saves the actual values and the deviation in the following Q parameters:

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

☞ **Before programming, note the following:**

Before a cycle definition you must have programmed a tool call to define the touch probe axis.
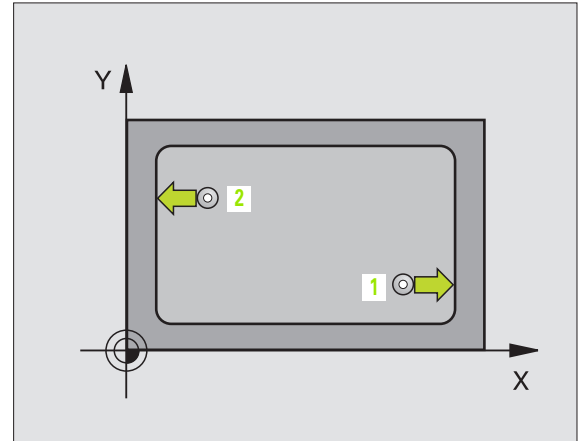
▶ **Starting point in 1st axis** Q328 (absolute): starting point for probing in the reference axis of the working plane.

▶ **Starting point in 2nd axis** Q329 (absolute): starting point for probing in the minor axis of the working plane.

▶ **Offset for 2nd measurement** Q310 (incremental): distance by which the touch probe is displaced before the second measurement. If you enter 0, the TNC does not displace the touch probe.

▶ **Measuring axis** Q272: axis in the working plane in which the measurement is to be made:
1: Reference axis = measuring axis
2: Minor axis = measuring axis

▶ **Measuring height in the touch probe axis** Q261 (absolute): coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.

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

▶ **Nominal length** Q311: nominal value of the length to be measured.

▶ **Maximum dimension** Q288: maximum permissible length.

▶ **Minimum dimension** Q289: minimum permissible length

▶ **Measuring log** Q281: definition of whether the TNC is to create a measuring log:
0: No measuring log
1: Generate measuring log: with the standard setting the TNC saves the **log file TCHPR425.TXT** in the directory in which your measuring program is also stored.

▶ **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 71):
0: Monitoring not active
>0: Tool number in the tool table TOOL.T
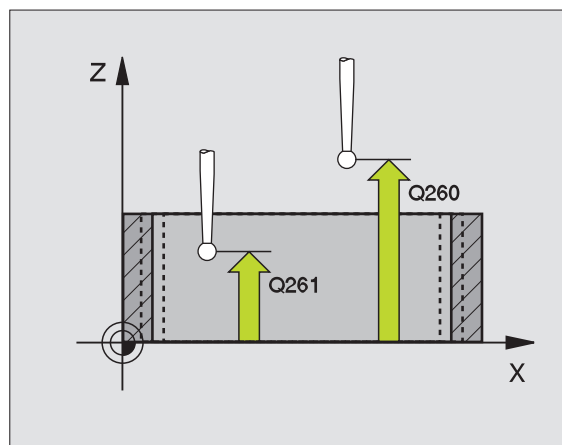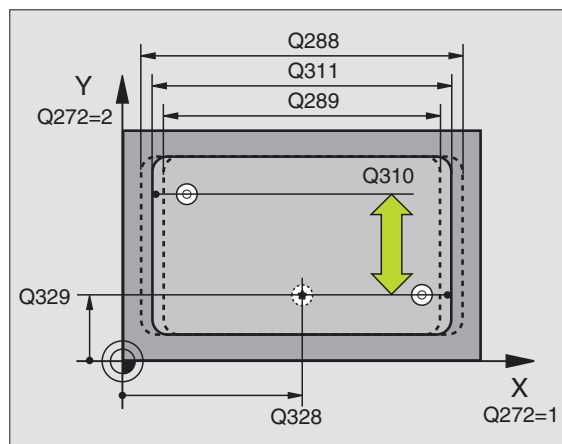
Example: NC blocks

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;MAXIMUM DIMENSION
Q289=25   ;MINIMUM DIMENSION
Q281=1   ;MEASURING LOG
Q309=0   ;PGM-STOP IF ERROR
Q330=0   ;TOOL NUMBER

MEASURE RIDGE WIDTH (touch probe cycle 426, 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 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 7) 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). 1. 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.

▶ **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 71)
0: Monitoring not active
>0: Tool number in the tool table TOOL.T
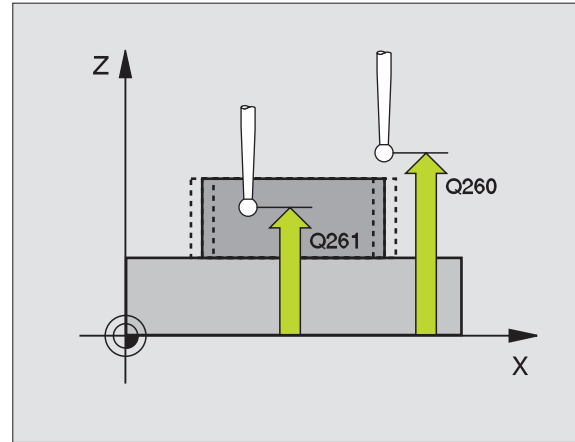
Example: NC blocks

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	;MAXIMUM DIMENSION
Q289=44.95	;MINIMUM DIMENSION
Q281=1	;MEASURING LOG
Q309=0	;PGM-STOP IF ERROR
Q330=0	;TOOL NUMBER

MEASURE COORDINATE (touch probe cycle 427, 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 7) 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:

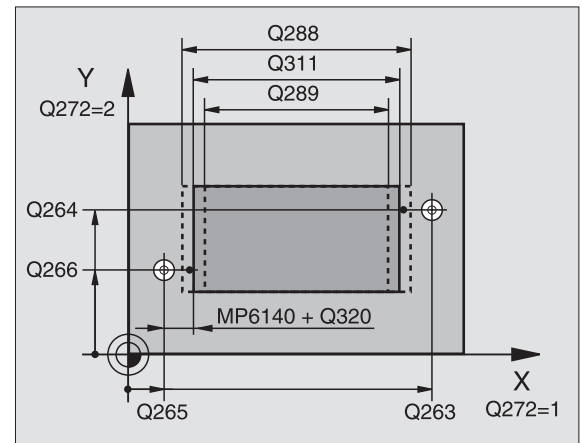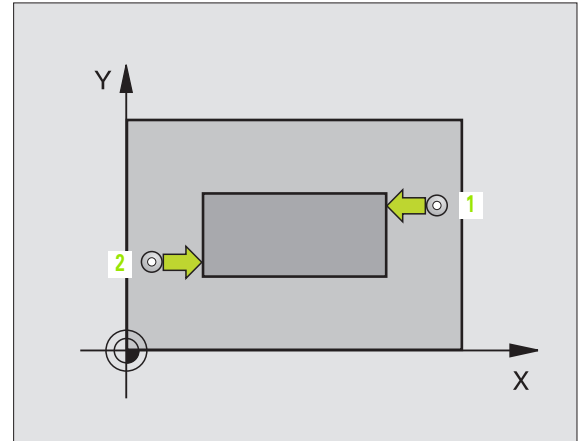| Parameter number	Meaning
Q160	Measured coordinate

☞ **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.

▶ **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 71):
0: Monitoring not active
>0: Tool number in the tool table TOOL.T
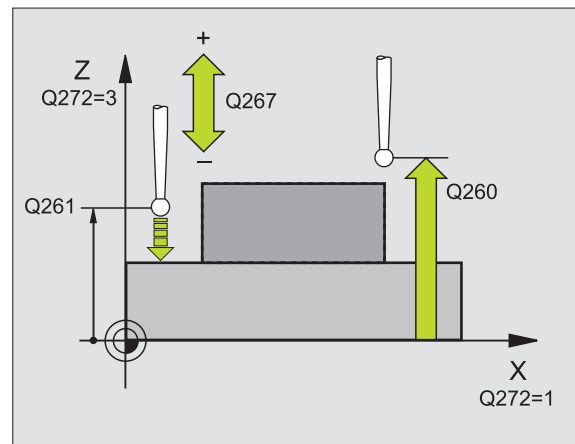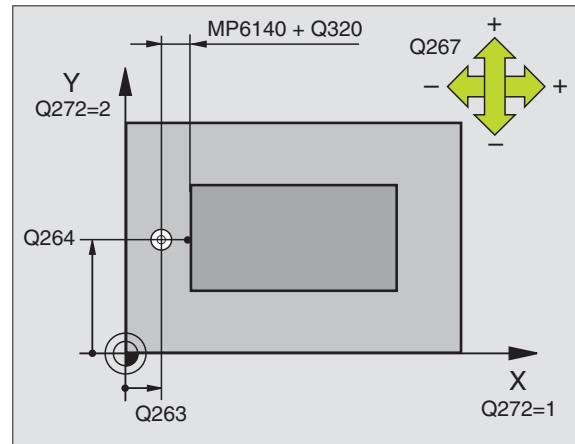
Example: NC blocks

```
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   ;MAXIMUM DIMENSION
    Q289=4.95  ;MINIMUM DIMENSION
    Q309=0     ;PGM-STOP IF ERROR
    Q330=0     ;TOOL NUMBER
```

MEASURE BOLT HOLE CIRCLE (touch probe cycle 430, 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 in system parameters.

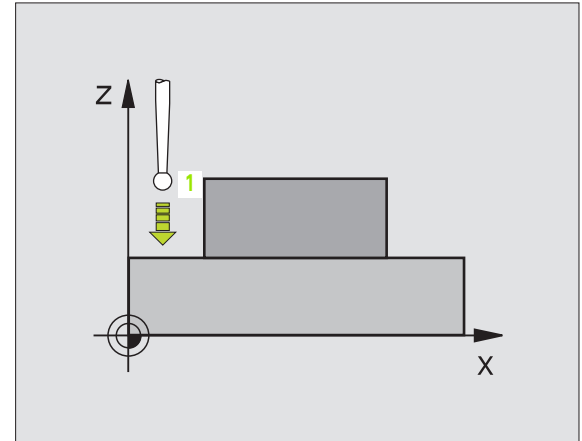**1** Following the positioning logic (see "Running touch probe cycles" on page 7), the TNC positions the touch probe at rapid traverse (value from MP6150 or MP6361) to the point entered as center of the first hole **1**.

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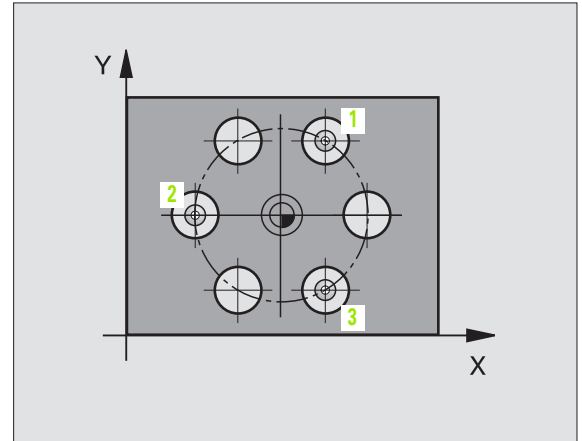
▷ **Center of 1st axis** Q273 (absolute): bolt hole circle center (nominal value) in the reference axis of the working plane.

▷ **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
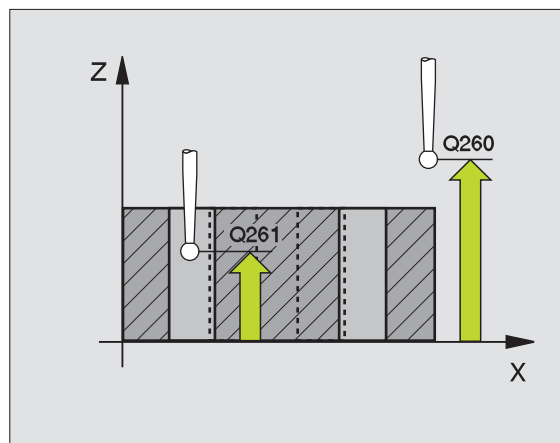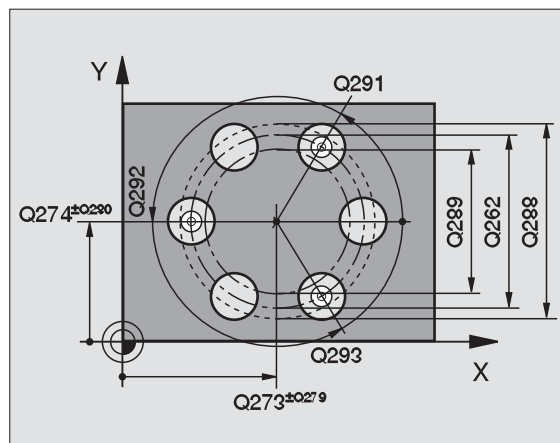
▶ **Measuring log** Q281: definition of whether the TNC is to create a measuring log:
0: No measuring log
1: Generate measuring log: with the standard setting the TNC saves the **log file TCHPR430.TXT** in the directory in which your measuring program is also stored.

▶ **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 71):
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 IN 1ST AXIS
Q274=+50	;CENTER IN 2ND AXIS
Q262=80	;NOMINAL DIAMETER
Q291=+0	;ANGLE OF 1ST HOLE
Q292=+90	;ANGLE OF 2ND HOLE
Q293=+180	;ANGLE OF 3RD HOLE
Q261=-5	;MEASURING HEIGHT
Q260=+10	;CLEARANCE HEIGHT
Q288=80.1	;MAXIMUM DIMENSION
Q289=79.9	;MINIMUM DIMENSION
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, 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 7), 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	Angle of the A axis
Q159	Angle of the B axis
Q170	Rotation about the A axis
Q171	Rotation about the B axis
Q172	Rotation about the C 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.

As of NC software 280 476-12, parameters Q170 to Q172 find the angle of the rotary axes that are needed in the "tilted working plane with spatial angle" function. With the first two measuring points you also specify the direction of the reference axis when tilting the working plane.
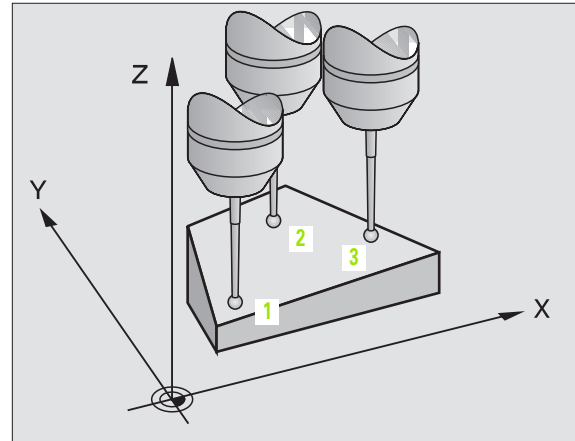
▶ **First measuring point in the 1st axis** Q263 (absolute): coordinate of the first touch point in the reference axis of the working plane.

▶ **First measuring point in the 2nd axis** Q264 (absolute): coordinate of the first touch point in the minor axis of the working plane.

▶ **First measuring point in the 3rd axis** Q294 (absolute): coordinate of the first touch point in the touch probe axis.

▶ **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

▶ **Second measuring point in the 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.

▶ **Third measuring point in the 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.
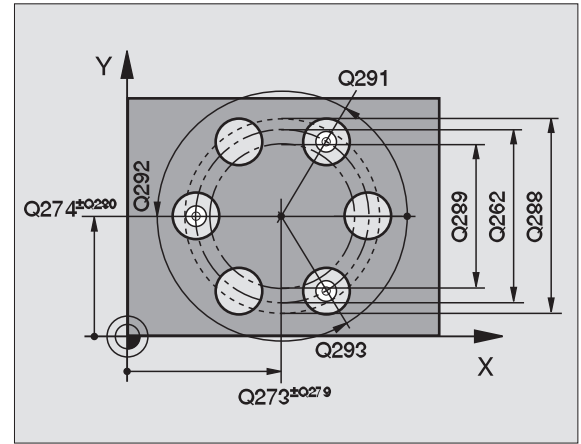
Example: NC blocks

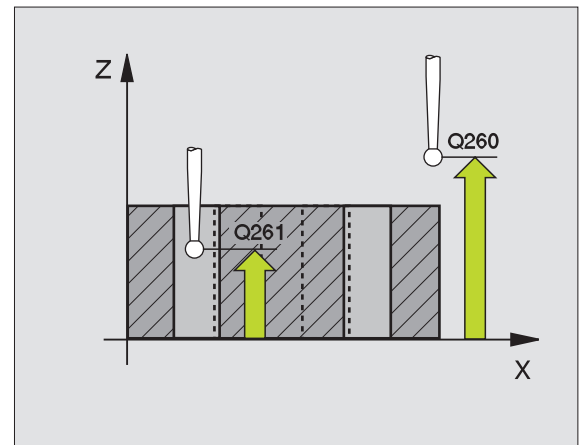| 5 TCH PROBE 431 MEASURE PLANE |
|---|
| Q263=+20   ;1ST POINT 1ST AXIS |
| Q264=+20   ;1ST POINT 2ND AXIS |
| Q294=-10   ;1ST POINT 3RD AXIS |
| Q265=+50   ;2ND POINT 1ST AXIS |
| Q266=+80   ;2ND POINT 2ND AXIS |
| Q295=+0   ;2ND POINT 3RD AXIS |
| Q296=+90   ;3RD POINT 1ST AXIS |
| Q297=+35   ;3RD 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
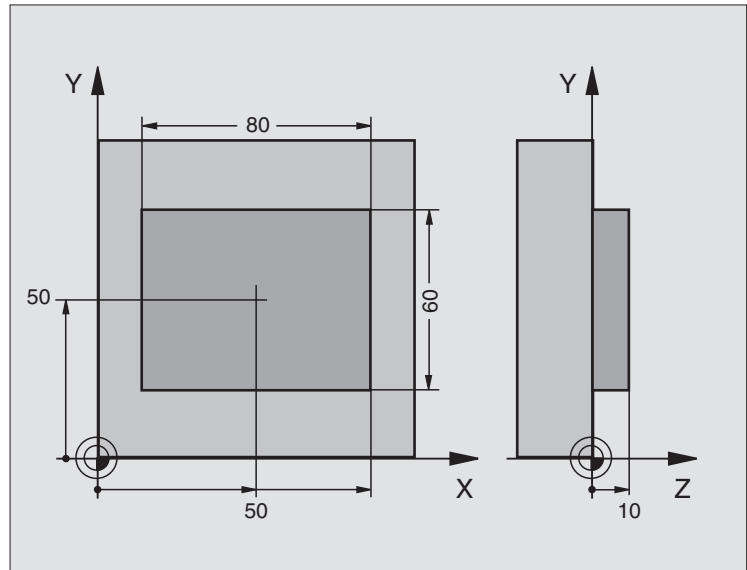
0 BEGIN PGM BEAMS MM	
1 TOOL CALL 0 Z	Prepare tool call
2 L Z+100 R0 F MAX	Retract the tool
3 FN 0: Q1 = +81	Pocket length in X (roughing dimension)
4 FN 0: Q2 = +61	Pocket length in Y (roughing dimension)
5 CALL LBL 1	Call subprogram for machining
6 L Z+100 R0 F MAX M6	Retract the tool, change the tool
7 TOOL CALL 99 Z	Call the touch probe
8 TCH PROBE 424 MEAS. RECTAN. OUTS.	Measure the rough-milled rectangle
Q273=+50 ;CENTER IN 1ST AXIS	
Q274=+50 ;CENTER IN 2ND AXIS	
Q282=80 ;1ST SIDE LENGTH	Nominal length in X (final dimension)
Q283=60 ;2ND SIDE LENGTH	Nominal length in Y (final dimension)
Q261=-5 ;MEASURING HEIGHT	
Q320=0 ;SET-UP CLEARANCE	
Q260=+30 ;CLEARANCE HEIGHT	
Q301=0 ;TRAVERSE TO CLEAR HEIGHT	
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 F MAX M6`	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 F MAX 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 IN 1ST AXIS`	
`Q217=+50  ;CENTER IN 2ND AXIS`	
`Q218=Q1   ;1ST SIDE LENGTH`	Length in X variable for roughing and milling
`Q219=Q2   ;2ND SIDE LENGTH`	Length in Y variable for roughing and milling
`Q220=0    ;CORNER RADIUS`	
`Q221=0    ;ALLOWANCE IN 1ST AXS`	
`17 CYCL CALL M3`	Call the cycle
`18 LBL 0`	End of subprogram
`19 END PGM BEAMS MM`	

Example: Measuring a rectangular pocket and recording the results
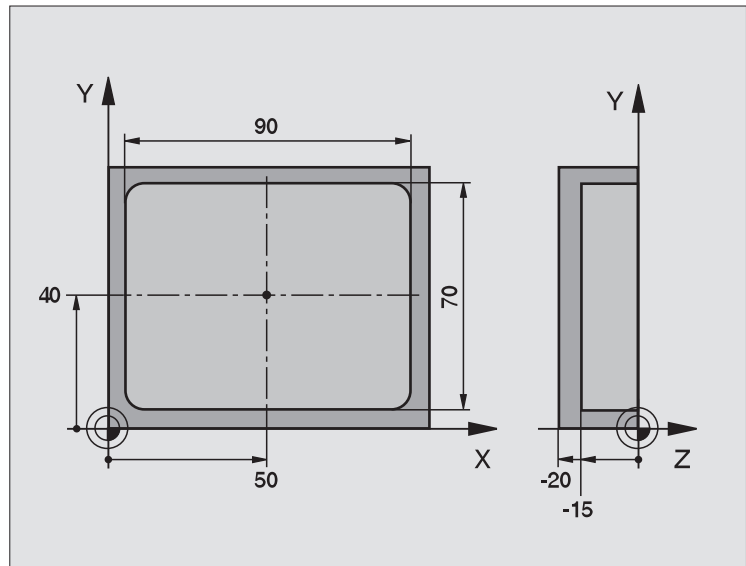
0 BEGIN PGM BSMEAS MM	
1 TOOL CALL 1 Z	Tool call for touch probe
2 L Z+100 R0 F MAX	Retract the touch probe
3 TCH PROBE 423 MEAS. RECTAN. INSIDE	
Q273=+50 ;CENTER IN 1ST AXIS	
Q274=+40 ;CENTER IN 2ND AXIS	
Q282=90 ;1ST SIDE LENGTH	Nominal length in X
Q283=70 ;2ND SIDE LENGTH	Nominal length in Y
Q261=-5 ;MEASURING HEIGHT	
Q320=0 ;SET-UP CLEARANCE	
Q260=+20 ;CLEARANCE HEIGHT	
Q301=0 ;TRAVERSE TO CLEAR HEIGHT	
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	Transmit the measuring log
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 F MAX M2	Retract in the tool axis, end program
5 END PGM BSMEAS MM	

Measuring log (file TCPR423.TXT)

```
--------------------------------------------------------------------------------
****************** MEASURING LOG FOR PROBING CYCLE 423: RECTANGULAR POCKET MEASURING ******************
DATE: 29-09-1997
TIME:  8:21:33
MEASURING PROGRAM: TNC:\BSMESS\BSMES.H
--------------------------------------------------------------------------------
NOML. VALUES:          CENTER IN REF. AXIS:              50.0000
                       CENTER IN MINOR AXIS:             40.0000

                       SIDE LENGTH IN REF. AXIS:         90.0000
                       SIDE LENGTH IN MINOR AXIS:        70.0000
--------------------------------------------------------------------------------
GIVEN LIMIT VALUES:    MAX. FOR CENTER IN REF. AXIS:     50.1500
                       MIN. FOR CENTER IN REF. AXIS:     49.8500

                       MAX. FOR CENTER IN MINOR AXIS:    40.1000
                       MIN. FOR CENTER IN MINOR AXIS:    39.9000

                       MAX. IN REFERENCE AXIS:           90.1500
                       MINIMUM DIMENSION IN REFERENCE AXIS: 89.9500

                       MAXIMUM SIDE LENGTH IN MINOR AXIS:   70.1000
                       MINIMUM SIDE LENGTH IN MINOR AXIS:   69.9500
********************************************************************************
ACTUAL VALUES:         CENTER IN REF. AXIS:              50.0905
                       CENTER IN MINOR AXIS:             39.9347

                       SIDE LENGTH IN REF. AXIS:         90.1200
                       SIDE LENGTH IN MINOR AXIS:        69.9920
--------------------------------------------------------------------------------
DEVIATIONS:            CENTER IN REF. AXIS:              0.0905
                       CENTER IN MINOR AXIS:             −0.0653

                       SIDE LENGTH IN REF. AXIS:         0.1200
                       SIDE LENGTH IN MINOR AXIS:        −0.0080
********************************************************************************
FURTHER MEASURING RESULTS: MEASURING HEIGHT: −5.0000
********************************* END OF MEASURING LOG *********************************
```

3.4 Special Cycles

Overview

The TNC provides three cycles for the following special purposes:

Cycle	Soft key
2 CALIBRATE TS Calibrate touch probe	
3 MEASURING Cycle for defining OEM cycles	
440 MEASURING AXIS SHIFT	

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 the
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
```

MEASURING (touch probe cycle 3, available as of NC software 280 474-xx)

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 and feed rate directly. The touch probe does not retract automatically after determining the measured value.

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. You define the number of the first parameter in the cycle.

3 If necessary, you must program the retraction of the touch probe in a separate traversing block.

☞ **Before programming, note the following:**

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

With the non-modal function **M141** (available as of NC software No. 280 476-06) you can switch off tool monitoring in order to be able to retract the tool in a traversing block. Make sure that you have selected the correct disengaging direction, otherwise the touch probe may be damaged.

▶ **Parameter number for result**: Enter the number of the Q parameter to which you want the TNC to assign the first coordinate (X).

▶ **Probe axis**: Enter the reference axis of the working plane (X for tool axis Z, Z for tool axis Y, and Y for tool axis X), and confirm with ENT.

▶ **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 may move. Confirm with ENT.

▶ **Feed rate**: Enter the measuring feed rate.

▶ To end input: press the ENT key

Example: NC blocks

```
5 TCH PROBE 3.0 MEASURING
6 TCH PROBE 3.1 Q1
7 TCH PROBE 3.2 X ANGLE: +15
8 TCH PROBE 3.3 DIST +10 F100
```

MEASURE AXIS SHIFT (touch probe cycle 440, ISO: G440, available as of NC software 280 476-xx)

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 the control measurement.

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

▶ **Direction: 0=Calibrate, 1=Measuring?**: Definition of whether you want to calibrate or make a control 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

4

**Touch Probe Cycles for
Automatic Tool Measurement**

4.1 Tool Measurement with the TT Tool Touch Probe

Overview

> The TNC and the machine tool must be set up by the machine tool builder for use of the TT touch probe.
>
> Some cycles and functions may not be provided on your machine tool. Refer to your machine manual.

In conjunction with the TNC's tool measurement cycles, the tool touch probe enables you to measure tools automatically. The compensation values for tool length and radius can be stored in the central tool file TOOL.T and are accounted for in the next tool call. 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 60 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

Display the results of measurement

You can display the results of tool measurement in the additional status display by pressing the STATUS TOOL PROBE soft key (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.

```
Program run, full sequence              Test run

0  BEGIN PGM FK1 MM                 Tool data T 5
1  BLK FORM 0.1 Z X+0 Y+0 Z-20              MIN
2  BLK FORM 0.2 X+100 Y+100 Z+0             MAX
3  TOOL CALL 1 Z                            DYN
4  L Z+250 R0 F MAX
5  L X-20 Y+30 R0 F MAX
6  L Z-10 R0 F1000 M3
7  APPR CT X+2 Y+30 CCA90 R+5 RL F250
8  FC DR- R18 CLSD+ CCX+20 CCY+30

                   0% S-IST 16:59
                   2% S-MOM LIMIT 1
   X      +0.224 Y    +295.720 Z      +24.447
  +B      -0.477 +c        +8.439
                                   S    114.014
ACTL.              T 5    Z S 1300    F 0      M 5/9

STATUS   STATUS   STATUS   STATUS    STATUS   STATUS OF
PGM      POS.     TOOL     COORD.    TOOL     M FUNCT.
                           TRANSF.   PROBE
```

4.2 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

☞ Cycles 480 to 483 are available as of NC software 280 476-xx.

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

☞ 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 is calibrated automatically with the measuring cycle TCH PROBE 30 or TCH PROBE 480. 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 TT CALIBRATE
8 TCH PROBE 30.1 HEIGHT: +90

Example: NC blocks in new format

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

Measuring the tool length

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.

The tool length is measured with the cycle TCH PROBE 31 TOOL LENGTH. 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 tool toward the measuring surface of the TT (without rotation of the spindle) until the tool contacts 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.

Define cycle

▶ **Measure tool=0 / Check tool=1**: Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool length L in the central tool file TOOL.T by the delta value DL = 0. If you wish to inspect a tool, the TNC compares the measured length with the tool length L that is stored in TOOL.T. It then calculates the positive or negative deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation can also be used for Q parameter Q115. If the delta value is greater than the permissible tool length tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T).

▶ **Parameter number for result?**: Parameter number in which the TNC stores the status of the measurement:
0.0: Tool is within the tolerance
1.0: Tool is worn (**LTOL** exceeded)
2.0: Tool is broken (**LBREAK** exceeded). If you do not wish to use the result of measurement within the program, answer the dialog prompt with NO ENT.

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

▶ **Cutter measurement? 0=No / 1=Yes**: Choose whether or not the TNC is to measure the individual 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

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.

The tool radius is measured with the cycle TCH PROBE 32 TOOL RADIUS. 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 as of NC software 280 476-xx. 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 or not the TNC is to measure the individual 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 tool length and radius

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 MEASURE TOOL. 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 as of NC software 280 476-xx. 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 or not the TNC is to measure the individual 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

5

Digitizing

# 5.1 Digitizing with Triggering or Measuring Touch Probe (Option)

Overview

The digitizing option enables you to reduce a three-dimensional part into discrete digital information by scanning it with a touch probe.

The following components are required for digitizing:

- touch probe
- "Digitizing option" software module in the TNC
- SUSA evaluation software from HEIDENHAIN for further processing of the data digitized in the MEANDER cycle.

The touch probe systems provide four digitizing cycles

Cycle	Soft key
5 RANGE cuboid, triggering and measuring touch probe: Defining a digitizing range	5 MAX MIN
6 MEANDER, triggering touch probe: Digitizing in back-and-forth parallel lines	6
7 CONTOUR LINES, triggering touch probe: Digitizing in upwardly successive levels	7
8 LINE, triggering touch probe: Digitizing parallel lines in one direction	8
15 RANGE Point table, measuring touch probe: Defining a digitizing range	15 PNT
16 MEANDER, measuring touch probe: Digitizing in back-and-forth parallel lines	16
17 CONTOUR LINES, measuring touch probe: Digitizing in upwardly successive levels	17
18 LINE, measuring touch probe: Digitizing parallel lines in one direction	18

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

☞ The touch probe must be calibrated before digitizing.

If you are working alternately with a triggering and a measuring touch probe, ensure that

■ You have selected the correct touch probe in MP 6200.
■ The measuring and triggering touch probes are never connected to the control at the same time.

The TNC cannot detect which probe is actually in the spindle.

Function

The touch probe scans a 3-D surface point for point in a selectable grid. The scanning speeds for triggering touch probes vary from 200 to 800 mm/min with a probe point interval (PP.INT) of 1 mm. The scanning speeds for measuring touch probes are programmed in the scanning cycle. You can enter up to 3000 mm/min.

The TNC saves the digitized positions directly to the hard disk. The interface function PRINT determines in which directory the data are stored.

If you want to mill the digitized data subsequently using a tool whose radius equals the radius of the ball tip, you can machine the digitized data directly from the file with Cycle 30 (see User's Manual, section 8.8 "Cycles for Multipass Milling").

☞ Digitizing cycles are programmed only for the main axes X, Y and Z and for the rotary axes A, B and C.

Coordinate transformations or a basic rotation must not be active during digitizing.

The TNC also transfers the **BLK FORM** to the digitizing data file. For this purpose, the TNC enlarges the workpiece blank defined in the RANGE cycle by twice the value set in MP6310 (for measuring touch probes).

5.2 Programming Digitizing Cycles

Selecting digitizing cycles

▶ press the TOUCH PROBE key.

▶ Use soft keys to select the desired digitizing cycle

▶ Answer the dialog questions on the TNC screen: Enter the appropriate values with the keyboard and confirm each input with the ENT key. When all the required data have been entered, the TNC terminates the cycle definition automatically. For information on the individual input parameters refer to the cycle descriptions in this chapter.

Defining the digitizing range

The touch probe systems provide two cycles for defining the digitizing range. Cycle 5 RANGE defines a cuboid range within which the touch probe scans the contour. If you are using a measuring touch probe, you can also select a contour point table with Cycle 15 RANGE, in which the shape of the digitizing range is defined as a polygon.

Defining a cuboid digitizing range

Similar to defining the workpiece blank, the digitizing range is programmed by entering the MIN and MAX point coordinates of the three main axes X, Y and Z (see figure at right).

▶ **PGM Name digitizing data**: Enter the name of the file in which the TNC is to store the digitizing data.

☞ In the screen menu for configuring the data interface, you must enter the complete directory path in which the TNC is to store the digitized data.

▶ **TCH PROBE axis**: Enter the touch probe axis
▶ **MIN point in range**: Lowest coordinates in the range to be digitized.
▶ **Max. point range**: Highest coordinates in the range to be digitized.
▶ **Clearance height**: Position in probe axis at which the stylus cannot collide with the model.
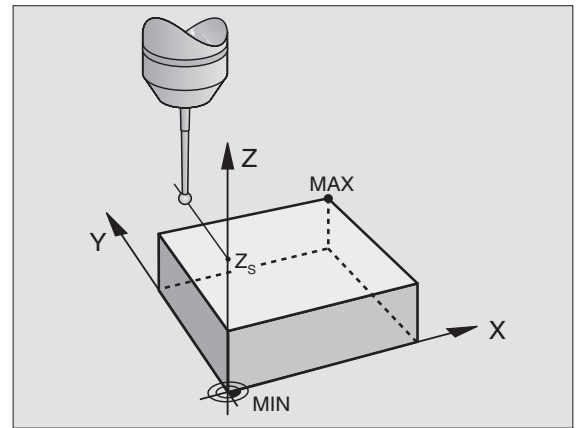

Example

50 TCH PROBE 5.0 RANGE
51 TCH PROBE 5.1 PGM NAME: DATA
52 TCH PROBE 5.2 Z X+0 Y+0 Z+0
53 TCH PROBE 5.3 Z X+10 Y+10 Z+2
54 TCH PROBE 5.4 HEIGHT: +100

Defining the digitizing range in a specific shape (only available with measuring touch probes)

☞ Digitizing cycle 15 cannot be combined with digitizing cycle 17 CONTOUR LINES.

The digitizing range is defined by a contour point table generated in the Positioning with MDI mode of operation. You can either transfer the individual points by teach-in programming or move the touch probe around the model manually and have the TNC generate them automatically (see figure at right).

▶ **PGM Name digitizing data**: Enter the name of the file in which the TNC is to store the digitizing data.

☞ In the screen menu for configuring the data interface, you must enter the complete directory path in which the TNC is to store the digitized data.

▶ **TCH PROBE axis**: Enter the touch probe axis
▶ **PGM Name range data**: Enter the name of the point table in which you have defined the range.
▶ **Min point TCH PROBE axis**: Lowest probe axis coordinate in the DIGITIZING range.
▶ **Max point TCH PROBE axis**: Highest probe axis coordinate in the DIGITIZING range.
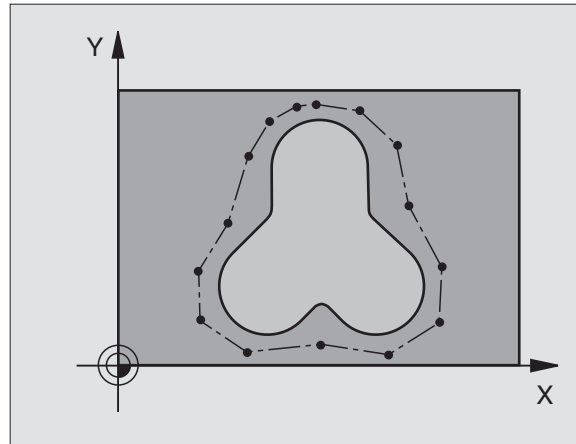▶ **Clearance height**: Position in probe axis at which the stylus cannot collide with the model.

Example

50 TCH PROBE 15.0 RANGE
51 TCH PROBE 15.1 PGM DIGIT: DATA
52 TCH PROBE 15.2 PGM RANGE: TAB1
53 TCH PROBE 15.3 MIN: +0 MAX: +10 HEIGHT: +100

Point Tables

When you are working with a measuring touch probe, you can generate point tables in the Positioning with MDI mode of operation. You can use these tables to define the digitizing range in a specific shape or to program a contour in a specific shape and then machine the contour with Cycle 30. This feature requires the "digitizing with measuring touch probe" software option from HEIDENHAIN.

You can transfer points in the following two ways

■ manually by TEACH-IN programming
■ automatically by the TNC

☞ The TNC can store up to 893 points in a point table that is to be used for the digitizing range. You can have the TNC monitor the number of points stored in the point table by setting the TM:RANGE/CONTOUR DATA soft key to TM:RANGE.

The individual points are connected to form a progression of straight lines that define the digitizing range. The TNC automatically connects the last point in the table with the first point by a straight line.

To generate a point table

After inserting the measuring touch probe in the spindle and locking it mechanically, you can select a point table with the PNT soft key:

PNT ▶ Press the PNT soft key in the Positioning with MDI mode of operation. The TNC then displays soft-key rows with the following soft keys:

Function	Soft key
Transfer points manually	PROBE MAN
Transfer points automatically	PROBE AUTO
Select between digitizing range and contour	TM:RANGE CONTOUR DATA
Store/Do not store X coordinate	X OFF / ON
Store/Do not store Y coordinate	Y OFF / ON
Store/Do not store Z coordinate	Z OFF / ON

▶ To select input of contour (CONTOUR DATA) or digitizing range (TM RANGE), set the TM:RANGE CONTOUR DATA soft key to the desired function.

If you want to transfer the points manually by teach-in programming, proceed as follows:

▶ To select manual transfer, press the PROBE MAN   soft key. The TNC then displays the following soft keys:

Function	Soft key
Feed rate at which the touch probe is to respond to a deflection	F
Store position in the point table with ACTUAL-POSITION-CAPTURE	┼

▶ Select the feed rate at which the touch probe is to respond to a deflection by pressing the F soft key and entering the feed rate.

▶ Define whether or not you want the TNC to transfer the coordinates in specific axes by setting the X OFF/ON, Y OFF/ON and Z OFF/ON soft keys to the selected functions.

▶ To move the touch probe to the first point of the range to be digitized or to the first contour point, deflect the touch probe manually in the desired direction.

▶ Press the ACTUAL-POSITION-CAPTURE soft key. The TNC transfers the coordinates of the selected axes to the point table. The TNC uses the coordinates of the working plane only for defining the digitizing range.

▶ Move the touch probe to the next point and capture the actual position. Repeat this process until the whole range is defined.

If you want to have the TNC generate the points automatically, proceed as follows:

▶ To transfer points automatically, press the PROBE AUTO soft key. The TNC then displays the following soft keys:

Function	Soft key
Feed rate at which the touch probe is to respond to a deflection	F
Define the probe point interval for automatic transfer	⊹

▶ Select the feed rate at which the touch probe is to respond to a deflection by pressing the F soft key and entering the feed rate.

▶ To transfer points automatically, press the PROBE AUTO soft key. The TNC displays additional soft keys.

▶ Select the feed rate at which the touch probe is to respond to a deflection by pressing the F soft key and entering the feed rate.

▶ Define the spacing between points at which the TNC transfers the points by pressing the PROBE POINT INTERVAL soft key and entering a value. After entry of the probe point interval, the TNC displays the START soft key.

▶ To move the touch probe to the first point of the range to be digitized or to the first contour point, deflect the touch probe manually in the desired direction.

▶ To start transfer, press the START soft key.

▶ Deflect the touch probe manually in the desired direction. The TNC transfers the coordinates in the programmed probe point interval.

▶ To terminate transfer, press the STOP soft key.

5.3 Types of Digitizing

Meander digitizing

■ Triggering touch probe: Digitizing Cycle 6 MEANDER
■ Measuring touch probe: Digitizing Cycle 16 MEANDER

The MEANDER cycle scans and digitizes a 3-D surface in a back-and-forth series of parallel lines. This method is best suited for digitizing relatively flat surfaces. If you want to use the SUSA evaluation software to further process the data, you must use the MEANDER cycle.

Before starting the digitizing process, select an axis in the working plane. During digitizing, the touch probe moves in the positive direction of the selected axis, starting at the MIN point of the working plane. When it reaches the range limit, it moves by the line spacing in the positive direction of the other axis and then travels back along this line. At the other end of this line, the touch probe moves once again by the line spacing. This process is repeated until the entire range has been scanned.

Once the entire range has been scanned, the touch probe returns to the clearance height.

When you are digitizing with the measuring touch probe, the TNC stores positions at which sharp changes in direction have occurred. Up to 1000 positions per line can be stored. In the next line, the TNC automatically reduces the probe feed rate in the vicinity of such positions. This behavior improves the scanning results.

Starting position
■ MIN point coordinates from Cycle 5 RANGE or Cycle 15 RANGE in the working plane, probe axis coordinate = clearance height.
■ Automatically move to the starting position: First in the probe axis to clearance height, then in the working plane.

Contour approach
The touch probe moves in the negative probe axis direction toward the model. When it makes contact, the TNC stores the position coordinates.

☞ Before the MEANDER cycle, the part program must have a range defined in the RANGE cycle.
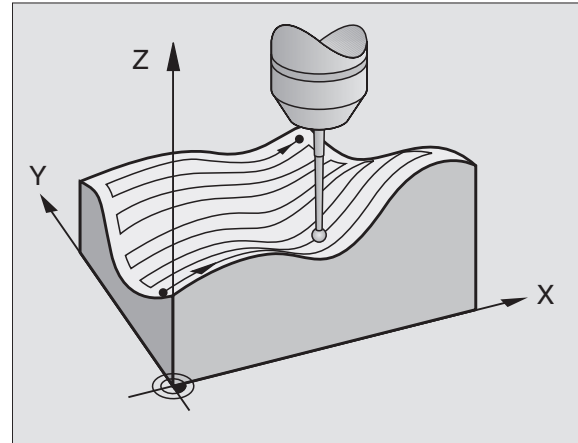
Digitizing parameters

The parameters marked with an **(M)** apply to measuring touch probes, those marked with an **(S)** apply to triggering touch probes:

▶ `Line direction` **(M, S)**: Coordinate axis in whose positive direction the touch probe moves from the first contour point.

▶ `Limit in normal lines direction` **(S)**: Distance by which the touch probe lifts off the model after each deflection of the stylus during scanning. Input range: 0 to 5 mm. Recommended input value: Enter an input value between 50% and 100% of the probe point interval. The smaller the ball-tip radius, the larger the Limit in Normal lines direction should be programmed.

▶ `Scanning direction` **(M)**: Traversing direction of the touch probe referenced to the line direction. Input range: –90° to +90°

▶ `Feed rate F` **(M)**: Enter the digitizing speed. Input range: 1 to 3 000 mm/min. Note that the higher the digitizing speed, the less accurate the resulting data will be.

▶ `MIN. feed rate` **(M)**: Probe feed rate for the first line. Input range: 1 to 3000 mm/min

▶ `Min. line spacing` **(M)**: If you enter an input value that is smaller than the programmed `line spacing`, the TNC decreases the spacing between the scanned lines to the programmed minimum for digitizing flat sections of a contour. This ensures that the density of digitized positions remains constant even if complex surface structures are scanned. Input range: 0 to 20 mm **(M)**, 0 to 5 mm **(S)**

▶ `Line spacing` **(M, S)**: The offset by which the probe moves in the probe axis at the end of each line before scanning the next line. Input range: 0 to 20 mm **(M)**, 0 to 5 mm **(S)**

▶ `Max. probe point interval` **(M, S)**: Maximum spacing between consecutive digitized positions. The TNC also accounts for important points that define the contour of the model, for example at inside corners. Input range: 0.02 to 20 mm **(M)**, 0.02 to 5 mm **(S)**

▶ `Tolerance` **(M)**: The TNC stores only those digitized positions whose distance from a straight line defined by the last two probe points exceeds the programmed tolerance. This ensures a high density of digitized positions for contours with sharply curved surfaces, and as low a density as possible for contours with flat surfaces. An input value of zero means that the digitized positions are output in the programmed probe point interval. Input range: 0 to 0.9999 mm

▶ `Feed rate reduction at edges` **(M)**: Answer the dialog question with NO ENT. The TNC enters a value automatically.

☞ The feed rate reduction is effective only if the number of points at which the feed rate must be reduced does not exceed 1000 points in a digitizing line.

Example: NC blocks for triggering touch probes

```
60 TCH PROBE 6.0 MEANDER
61 TCH PROBE 6.1 DIRECTN: X
62 TCH PROBE 6.2 TRAVEL: 0.5 L.SPAC: 0.2
    PP.INT: 0.5
```

Example: NC blocks for measuring touch probes

```
60 TCH PROBE 16.0 MEANDER
61 TCH PROBE 16.1 DIRECTN: X
    ANGLE: +0
62 TCH PROBE 16.2 F1000 FMIN500
    MIN.L.SPAC: 0.2 L.SPAC: 0.5
    PP.INT: 0.5 TOL: 0.1 DIST: 2
```

Contour line digizing

- Triggering touch probe: Digitizing Cycle 7 CONTOUR LINES
- Measuring touch probe: Digitizing cycle 17 CONTOUR LINES

The CONTOUR LINES cycle scans a 3-D contour by circling around the model in a series of upwardly successive levels. This cycle is particularly useful for digitizing steep contours, such as the gating of injection molding dies, or for digitizing a single contour line, such as the contour of a cam disk.

After first making contact with the form, the probe starts scanning at constant height around the form. After it has orbited the form and returned to the first probe point, it then moves in the positive or negative direction of the probe axis by the value of the LINE SPACING. It then once again orbits the form at constant height. This process is repeated until the entire range has been scanned.

Once the entire range has been scanned, the touch probe returns to the clearance height and the programmed starting point.

When you are digizing with the measuring touch probe, the TNC stores positions at which sharp changes in direction have occurred. Up to 1000 positions per line can be stored. In the next line, the TNC automatically reduces the probe feed rate in the vicinity of such positions. This behavior improves the scanning results.

Requirements for the scanning range
- In the touch probe axis: The defined RANGE must be lower than the highest point of the 3-D model by at least the radius of the probe tip.
- In the plane perpendicular to the touch probe: The defined RANGE must be larger than the 3-D model by at least the radius of the probe tip.

Starting position
- Probe axis coordinate of the MIN point from Cycle 5 RANGE if the line spacing is entered as a positive value.
- Probe axis coordinate of the MAX point from Cycle 5 RANGE if the line spacing is entered as a negative value.
- Define the working plane coordinates in the CONTOUR LINES cycle.
- Automatically move to the starting position: First in the probe axis to clearance height, then in the working plane.

☞ Before the CONTOUR LINES cycle, the part program must have a range defined in the RANGE cycle.

Digizing cycle 17 cannot be combined with digizing cycle 15.
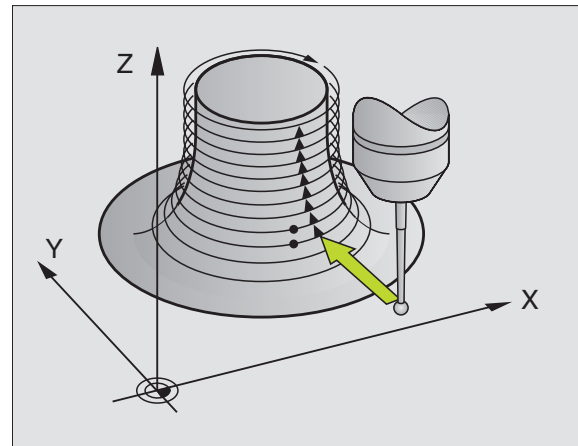
Digitizing parameters

The parameters marked with an **(M)** apply to measuring touch probes, those marked with an **(S)** apply to triggering touch probes:

▶ **Time limit (M, S)**: Time within which the touch probe must orbit the model and reach the first touch point. You can define a maximum distance from the end point to the starting point with machine parameter MP 6390. If the entered time limit is exceeded, the TNC interrupts the digitizing process. Input range: 0 to 7200 seconds. The input value 0 means there is no time limit.

▶ **Starting point (M, S)**: Coordinates of the starting position in the working plane.

▶ **Axis and direction of approach (M, S)**: Coordinate axis and direction in which the touch probe approaches the model.

▶ **Starting probe axis and direction (M, S)**: Coordinate axis and direction in which the probe begins scanning the model. Defining the direction at the same time determines whether the subsequent machining operation is performed by up-cut or climb milling.

▶ **Feed rate F (M)**: Enter the digitizing speed. Input range: 1 to 3 000 mm/min. Note that the higher the digitizing speed, the less accurate the resulting data will be.

▶ **MIN. feed rate (M)**: Probe feed rate for the first line. Input range: 1 to 3000 mm/min

▶ **Min. line spacing (M)**: If you enter an input value that is smaller than the programmed **line spacing**, the TNC decreases the spacing between the scanned lines to the programmed minimum for digitizing flat sections of a contour. This ensures that the density of digitized positions remains constant even if complex surface structures are scanned. Input range: 0 to 20 mm **(M)**, 0 to 5 mm **(S)**

▶ **Limit in normal lines direction (S)**: Distance by which the touch probe lifts off the model after each deflection of the stylus during scanning. Input range: 0 to 5 mm. Recommended input value: Enter an input value between 50% and 100% of the probe point interval. The smaller the ball-tip radius, the larger the Limit in Normal lines direction should be programmed.

▶ **Line spacing and direction (M, S)**: The offset by which the touch probe moves to start a new contour line after completing the previous one. The algebraic sign determines the direction in which the probe is offset. Input range: -20 to +20 mm **(M)**, -5 to +5 mm **(S)**

☞ If you only want to digitize a single contour line, enter zero for both the min. line spacing and the line spacing.

▶ **Max. probe point interval (M, S)**: Maximum spacing between consecutive digitized positions. The TNC also accounts for important points that define the contour of the model, for example at inside corners. Input range: 0.02 to 20 mm **(M)**, 0.02 to 5 mm **(S)**

Example: NC blocks for triggering touch probes

```
60 TCH PROBE 7.0 CONTOUR LINES
61 TCH PROBE 7.1 TIME:  0  X+0  Y+0
62 TCH PROBE 7.2 ORDER: Y- / X-
63 TCH PROBE 7.3 TRAVEL: 0.5 L.SPAC: +0.2
   PP.INT: 0.5
```

Example: NC blocks for measuring touch probes

```
60 TCH PROBE 17.0 CONTOUR LINES
61 TCH PROBE 17.1 TIME:  0  X+0  Y+0
62 TCH PROBE 17.2 ORDER: Y- / X-
63 TCH PROBE 17.2 F1000 FMIN500
   MIN.L.SPAC: 0.2 L.SPAC: +0.5
   PP.INT: 0.5 TOL: 0.1 DIST: 2
```

▶ **Tolerance (M)**: The TNC stores only those digitized positions whose distance from a straight line defined by the last two probe points exceeds the programmed tolerance. This ensures a high density of digitized positions for contours with sharply curved surfaces, and as low a density as possible for contours with flat surfaces. An input value of zero means that the digitized positions are output in the programmed probe point interval. Input range: 0 to 0.9999 mm

▶ **Feed rate reduction at edges (M)**: Answer the dialog question with NO ENT. The TNC enters a value automatically.

☞ The feed rate reduction is effective only if the number of points at which the feed rate must be reduced does not exceed 1000 points in a digitizing line.

Unidirectional line digitizing

■ Triggering touch probe: Digitizing cycle 8 LINE
■ Measuring touch probe: Digitizing Cycle 18 LINE

The LINE cycle scans and digitizes a 3-D contour in a single direction in a series of parallel lines.

With measuring touch probes, this cycle is used mainly for digitizing with a rotary axis. See "Digitizing with a Rotary Axis."

With triggering touch probes, this cycle is best suited for digitizing relatively flat surfaces which are to be consistently up-cut or climb milled without prior evaluation of the digitized data.

In this cycle, the touch probe moves in the positive direction of the axis entered under DIRECTION. When the touch probe reaches the MAX coordinate of this axis, it returns to clearance height. It then moves at rapid traverse back to the beginning of the next line and from there in the negative probe axis direction to the height for feed rate decrease. From this point it continues traveling at the probing feed rate until it contacts the 3-D contour. This process is repeated until the entire range has been scanned. The paths of traverse in the LINE cycle are illustrated in the figure at lower right.

Once the entire range has been scanned, the touch probe returns to the clearance height.

When you are digitizing with the measuring touch probe, the TNC stores positions at which sharp changes in direction have occurred. Up to 1000 positions per line can be stored. In the next line, the TNC automatically reduces the probe feed rate in the vicinity of such positions. This behavior improves the scanning results.
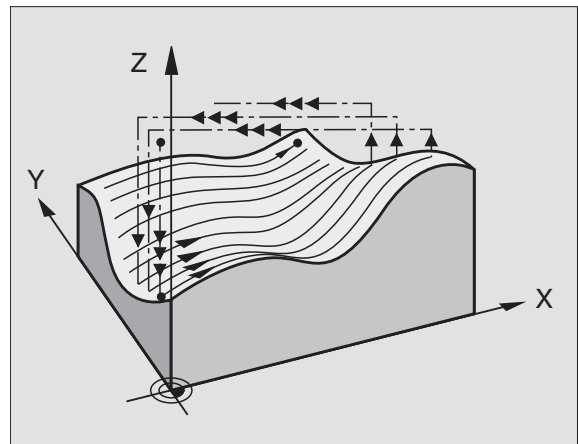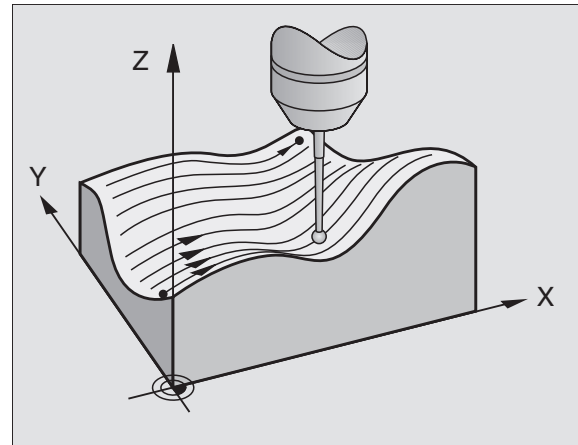
Starting position

- Positive or negative range limit of the programmed line direction (depending on the digitizing direction).
- MIN point coordinates from Cycle 5 RANGE or Cycle 15 RANGE in the working plane, probe axis coordinate = clearance height.
- Automatically move to the starting position: First in the probe axis to clearance height, then in the working plane.

Contour approach

The touch probe moves in the negative probe axis direction toward the model. When it makes contact, the TNC stores the position coordinates.

☞ Before the LINE cycle, the part program must have a range defined in the RANGE cycle.

Digitizing parameters

The parameters marked with an **(M)** apply to measuring touch probes, those marked with an **(S)** apply to triggering touch probes:

▶ **Line direction (M, S)**: Coordinate axis of the working plane parallel to which the touch probe scans the model. Defining the direction at the same time determines whether the subsequent machining operation is performed by up-cut or climb milling.

▶ **Scanning direction (M)**: Traversing direction of the touch probe referenced to the line direction. By combining line direction and scanning angle you can set the digitizing direction as desired. Input range: -90° to +90°

▶ **Height for feed rate reduction (M, S)**: Probe axis coordinate at which the touch probe feed rate is reduced from rapid traverse to the probing feed rate at the beginning of each line. Input range: -99 999.9999 to +99 999.9999

▶ **Feed rate F (M)**: Enter the digitizing speed. Input range: 1 to 3 000 mm/min. Note that the higher the digitizing speed, the less accurate the resulting data will be.

▶ **MIN. feed rate (M)**: Probe feed rate for the first line. Input range: 1 to 3000 mm/min

▶ **Min. line spacing (M)**: If you enter an input value that is smaller than the programmed **line spacing**, the TNC decreases the spacing between the scanned lines to the programmed minimum for digitizing flat sections of a contour. This ensures that the density of digitized positions remains constant even if complex surface structures are scanned. Input range: 0 to 20 mm **(M)**, 0 to 5 mm **(S)**

▶ **Limit in normal lines direction (S)**: Distance by which the touch probe lifts off the model after each deflection of the stylus during scanning. Input range: 0 to 5 mm. Recommended input value: Enter an input value between 50% and 100% of the probe point interval. The smaller the ball-tip radius, the larger the Limit in Normal lines direction should be programmed.

▶ **Line spacing (M, S)**: The offset by which the probe moves in the probe axis at the end of each line = line spacing. Input range: 0 to +20 mm **(M)**, 0 to +5 mm **(S)**

Example: NC blocks for triggering touch probes

```
60 TCH PROBE 8.0 LINE
61 TCH PROBE 8.1 DIRECTN: X HEIGHT: +25
62 TCH PROBE 8.2 TRAVEL: 0.5 L.SPAC.: 0.2
   PP.INT: 0.5
```

Example: NC blocks for measuring touch probes

```
60 TCH PROBE 18.0 LINE
61 TCH PROBE 18.1 DIRECTN: X ANGLE: 0
   HEIGHT: +25
62 TCH PROBE 18.2 F1000 FMIN500
   MIN.L.SPAC: 0.2 L.SPAC: 0.5
   PP.INT: 0.5 TOL: 0.1 DIST: 2
```

▶ **Max. probe point interval (M, S)**: Maximum spacing between consecutive digitized positions. The TNC also accounts for important points that define the contour of the model, for example at inside corners. Input range: 0.02 to 20 mm **(M)**, 0.02 to 5 mm **(S)**

▶ **Tolerance (M)**: The TNC stores only those digitized positions whose distance from a straight line defined by the last two probe points exceeds the programmed tolerance. This ensures a high density of digitized positions for contours with sharply curved surfaces, and as low a density as possible for contours with flat surfaces. An input value of zero means that the digitized positions are output in the programmed probe point interval. Input range: 0 to 0.9999 mm

▶ **Feed rate reduction at edges (M)**: Answer the dialog question with NO ENT. The TNC enters a value automatically.

☞ The feed rate reduction is effective only if the number of points at which the feed rate must be reduced does not exceed 1000 points in a digitizing line.

Digitizing with rotary axes

If you are using a triggering touch probe, the digitizing feature supports rotary axes with the cycles MEANDER (Cycle 6), LINE (Cycle 8) or CONTOUR LINE (Cycle 7). Regardless of the digitizing cycle used, the corresponding rotary axis must be entered in the RANGE cycle. The TNC interprets the rotary axis input as values in degrees.

If you are using a measuring touch probe, the digitizing feature supports rotary axes with cycle 18 LINE only. The rotary axis must be defined as a column axis.

Surface data

The digitized data contains position information on the axes that are identified in the RANGE cycle.

The TNC does not generate a **BLK FORM** since rotary axes cannot be graphically simulated.

☞ The display mode of the rotary axis (i.e. whether the display reduces the values to below 360° or not) must be the same for digitizing as for milling.

Measuring touch probe: LINE cycle with a rotary axis

If you enter a linear axis (e.g. X) in the input parameter LINE DIRECTION, the TNC moves the rotary axis (e.g. A) entered in the RANGE cycle by the line spacing L.SPAC as soon as it reaches the end of the line. See figures at right.

Example: NC blocks

```
30 TCH PROBE 5.0 RANGE

31 TCH PROBE 5.1 PGM DIGIT: DATROT

32 TCH PROBE 5.2 Z X+0 A+0 Z+0

33 TCH PROBE 5.3 X+85 A+270 Z+25

34 TCH PROBE 5.4 HEIGHT: 50

. . .

60 TCH PROBE 18.0 LINE

61 TCH PROBE 18.1 DIRECTN: X
   ANGLE: 0 HEIGHT: 25

62 TCH PROBE 18.2 F1000
   MIN.L.SPAC: 0.2 L.SPAC: 0.5
   PP.INT: 0.5 TOL: 0.1 DIST: 2
```
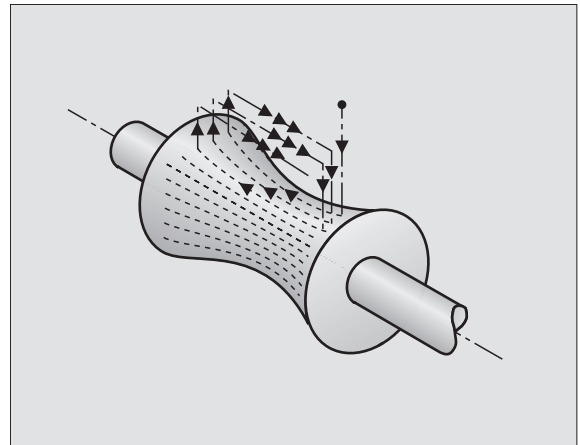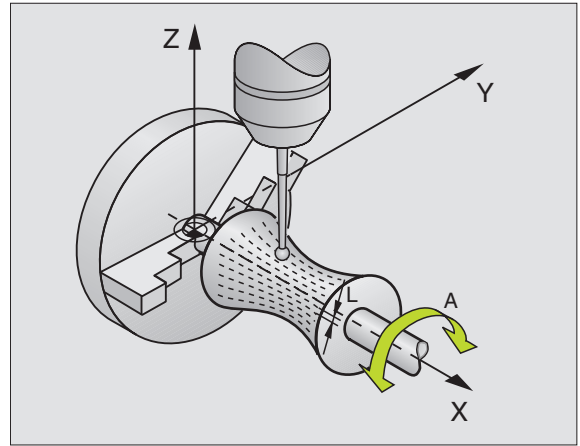
Triggering touch probe: MEANDER cycle with a rotary axis

If you enter a linear axis (e.g. X) in the input parameter for line direction, the TNC moves the rotary axis (e.g. A) entered in the RANGE cycle by the line spacing L.SPAC as soon as it reaches the end of the line. The touch probe then oscillates, for example in the Z/X plane (see figure at upper right).

If you define the line direction as a rotary axis, the TNC moves the linear axis entered in the RANGE cycle by the line spacing L.SPAC as soon as it reaches the end of the line. If the line direction is the rotary axis A, and L.SPAC is in the linear axis X, then the touch probe oscillates in the Z/A plane (see figure at center right).

Example: NC blocks

```
30 TCH PROBE 5.0 RANGE
```
```
31 TCH PROBE 5.1 PGM DIGIT: DATROT
```
```
32 TCH PROBE 5.2 Z X+0 A+0 Z+0
```
```
33 TCH PROBE 5.3 X+85 A+270 Z+25
```
```
34 TCH PROBE 5.4 HEIGHT: 100
```
```
. . .
```
```
60 TCH PROBE 6.0 MEANDER
```
```
61 TCH PROBE 6.1 DIRECTN: A
```
```
62 TCH PROBE 6.2 TRAVEL: 0.3 L.SPAC: 0.5  PP.INT: 0.5
```
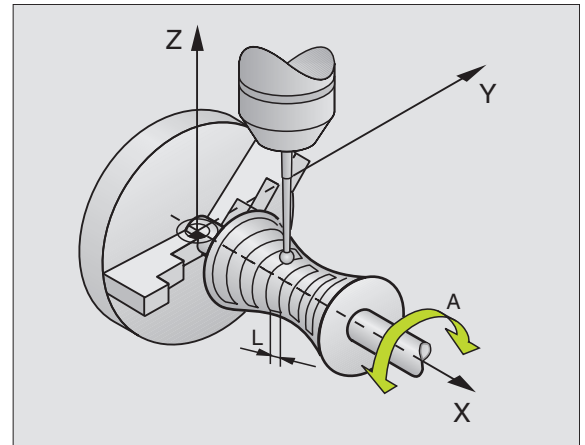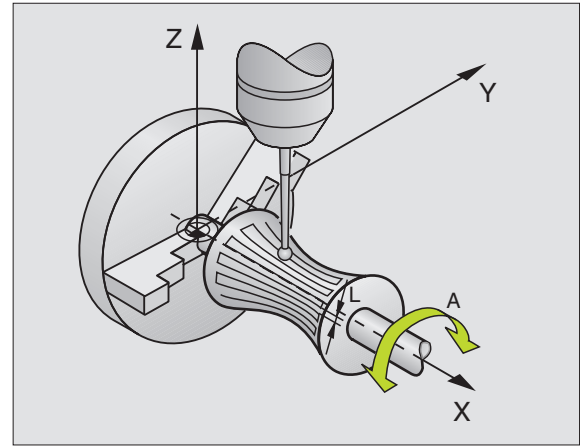
CONTOUR LINES cycle with a rotary axis

Define the starting point in a linear axis and in a rotary axis. Define also the order and direction of axis approach. The touch probe then oscillates, for example in the X/C plane. See figure at lower right.

This method also works well for machines that have only two linear axes (e.g. Z/X) and one rotary axis (e.g. C).

Example: NC blocks

```
30 TCH PROBE 5.0 RANGE
31 TCH PROBE 5.1 PGM DIGIT: DATH
32 TCH PROBE 5.2 Z X-500 C+0 Z+0
33 TCH PROBE 5.3 X+50 C+360 Z+85
34 TCH PROBE 5.4 HEIGHT: 100
. . .
60 TCH PROBE 7.0 CONTOUR LINES
61 TCH PROBE 7.1 TIME: 250 X+80 C+0
62 TCH PROBE 7.2 ORDER X-/C+
63 TCH PROBE 7.3 TRAVEL 0.3 L.SPAC: +0.5 PP.INT: 0.5
```

☞ The direction of rotation defined in Order is valid for all levels (lines). The direction at the same time defines whether the workpiece is then machined consistently in up-cut or in climb milling.
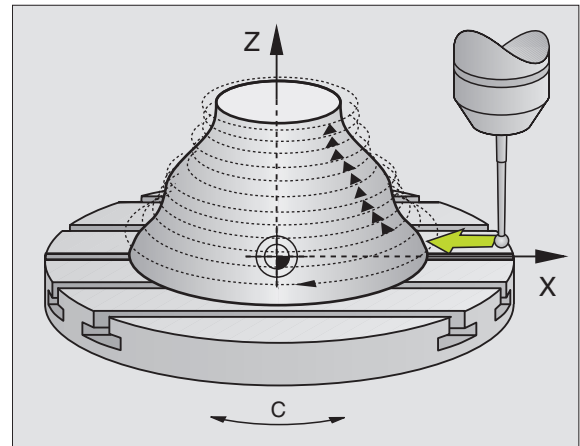
5.4 Using Digitized Data in a Part Program

Resulting NC blocks of a file containing data that were digitized with the CONTOUR LINES cycle.

`0   BEGIN PGM DATA MM`	Program name DATA is entered in the RANGE cycle
`1   BLK FORM 0.1 Z X-40 Y-20 Z+0`	Define workpiece blank: The size is determined by the TNC
`2   BLK FORM 0.2 X+40 Y+40 Z+25`	
`3   L Z+250 FMAX`	Clearance height in Z is entered in the RANGE cycle
`4   L X+0 Y-25 FMAX`	Starting point in X/Y is entered in the CONTOUR LINES cycle
`5   L Z+25`	Starting height in Z is entered in the CONTOUR LINES cycle and depends on the algebraic sign of the line spacing
`6   L X+0.002 Y-12.358`	First probed position
`7   L X+0.359 Y-12.021`	Second probed position
`...`	
`253   L X+0.003 Y-12.390`	A contour line has been completed: probe has returned to the first
`254   L Z+24.5 X+0.017 Y-12.653`	
`...`	
`2597   L X+0.093 Y-16.390`	Last probed position in the range
`2598   L X+0 Y-25 FMAX`	Back to the starting point in X/Y
`2599   L Z+250 FMAX`	Back to the clearance height in Z
`2600   END PGM DATA MM`	End of program

A surface-data file can have up to 1500 MB. This is the available memory capacity of the TNC's hard disk if no other programs are stored.

To execute a part program from digitized data, you can:

- Use fixed cycle 30 if several infeeds are required (only possible with data that were digitized in the MEANDER and LINE cycles, see the User's Manual, section 8.8 "Cycles for Multipass Milling").
- Write an auxiliary program if you only need to finish-mill:

0 BEGIN PGM MILLING MM	
1 TOOL DEF 1 L+0 R+4	Tool definition: tool radius = stylus radius
2 TOOL CALL 1 Z S4000	Tool call
3 L R0 F1500 M13	Define feed rate for milling, spindle and coolant ON
4 CALL PGM DATA	Call digitized data
5 END PGM MILLING MM	End of program

HEIDENHAIN

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

Technical support FAX +49 (8669) 31-1000
  E-Mail: service@heidenhain.de
Measuring systems ☎ +49 (8669) 31-3104
  E-Mail: service.ms-support@heidenhain.de
TNC support ☎ +49 (8669) 31-3101
  E-Mail: service.nc-support@heidenhain.de
NC programming ☎ +49 (8669) 31-3103
  E-Mail: service.nc-pgm@heidenhain.de
PLC programming ☎ +49 (8669) 31-3102
  E-Mail: service.plc@heidenhain.de
Lathe controls ☎ +49 (711) 952803-0
  E-Mail: service.hsf@heidenhain.de

www.heidenhain.de