HEIDENHAIN

NC Solutions

Description of NC Program 8245

English (en)
5/2019

1.1 Description of NC program 8245_en.h

NC program for ascertaining compensation values for the 3D-ToolComp option (option 92) with the touch probe, and writing them in the 3DTC table.

This NC program uses the angular values that you specified in the 3DTC table.

Requirement

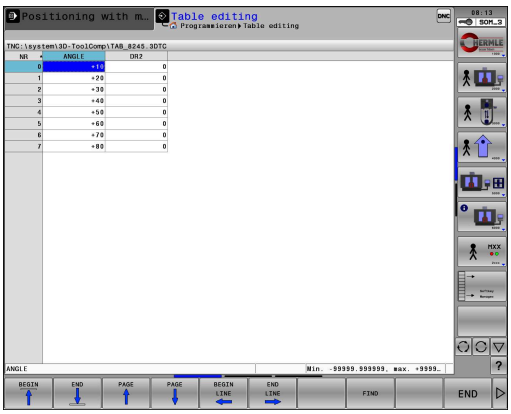
Compensate a tool three-dimensionally with the help of 3D-ToolComp (option 92). In this process you define the angular values where the tool will be compensated.

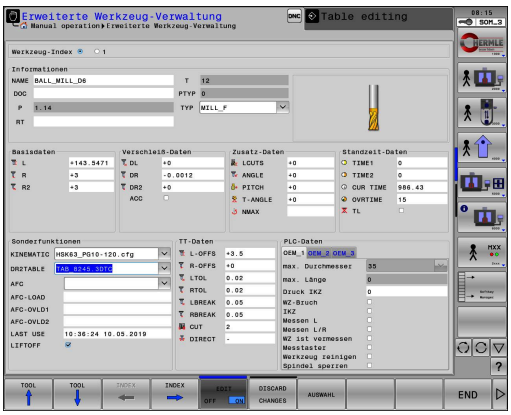
The measurement is performed with a touch probe. For this solution, you do not need a CAM system to calculate the normal vectors in the touch probe cycle.

Preparation

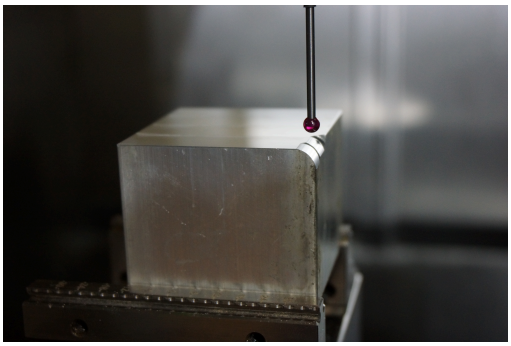
Perform the following steps before starting the NC program:

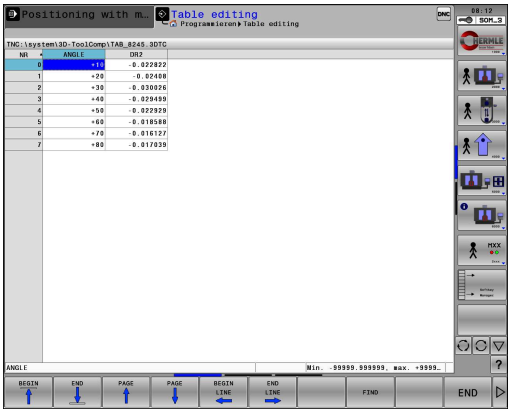
- Create a 3DTC table in **TNC:\system\3D-ToolComp**
- Insert lines in accordance with the desired number of measuring points
- Enter angular values
- Assign the 3DTC table to the tool in the tool table
- Adjust the paths and input parameters in the NC program
- Calibrate the touch probe in 3-D





Structure and program run of NC program 8245_en.h





At the beginning of the program, the control calls the NC program 82451_en.h which mills a 5 mm radius on the workpiece edge. The NC program opens the 3DTC table with FN 26.

You need to define the input parameters for the measurement. Consequently, the control calculates the number of repeats from the entered measuring points. The NC program calls the touch probe and pre-positions it.

During the subsequent program section repeat, the control reads out the angle in LBL CALC, and calculates the measuring point coordinates and the normal vectors from this. The control checks in LBL PROBE if the block scan or the Test Run operation mode is active. If they are not active, then the control performs the touch probe Cycle 444 at the calculated measuring point. The NC program converts the result parameter Q164 to the compensation value for the 3DTC table. This value is written into the table.

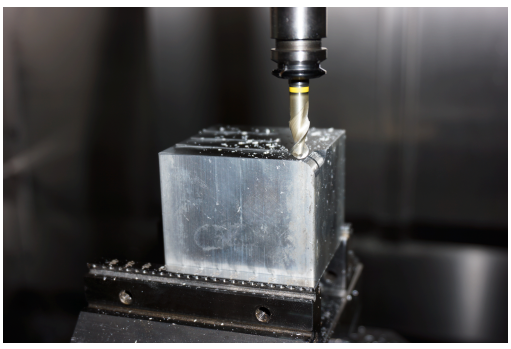
> As the touch probe cycle does not supply any values in block scan or Test Run, the 3DTC table would be overwritten with incorrect values. Therefore, the NC program only measures in the Full Sequence and Single Block operating modes.

The NC program increments the line number by one, and starts the program section repeat until the given number of lines is reached.

In this example program, the control calls another two NC programs at the end of the program. The first NC program 82452_en.h finishes another surface on the radius with the compensated tool. The second NC program 82453_en.h measures the newly machined radius and documents the results.

Parameter	Name	Meaning
QL20	FIRST LINE OF THE TABLE	Line number of the first line to be read, normally 0
QL21	NUMBER OF LINES IN THE TABLE	The number of lines corresponds to the number of measuring points
QL22	RADIUS TO BE MEASURED	Radius on the workpiece edge
QL2	MEASURING POINTS IN THE Y AXIS	Y coordinate of the measuring position

Structure and program run of NC program 82451_en.h



The NC program 82451_en.h is generated from the CAM system. The machining of the radius is programmed in the NC program.

First a spherical cutter performs the roughing operation with 8 mm diameter, then a spherical cutter performs the finishing operation with 6 mm diameter. The finishing tool is the tool which is then compensated.

Structure and program run of NC program 82452_en.h

The NC program 82452_en.h is generated from the CAM system. The second machining of the radius is programmed in the NC program. This machining only takes place in order to check the compensation.

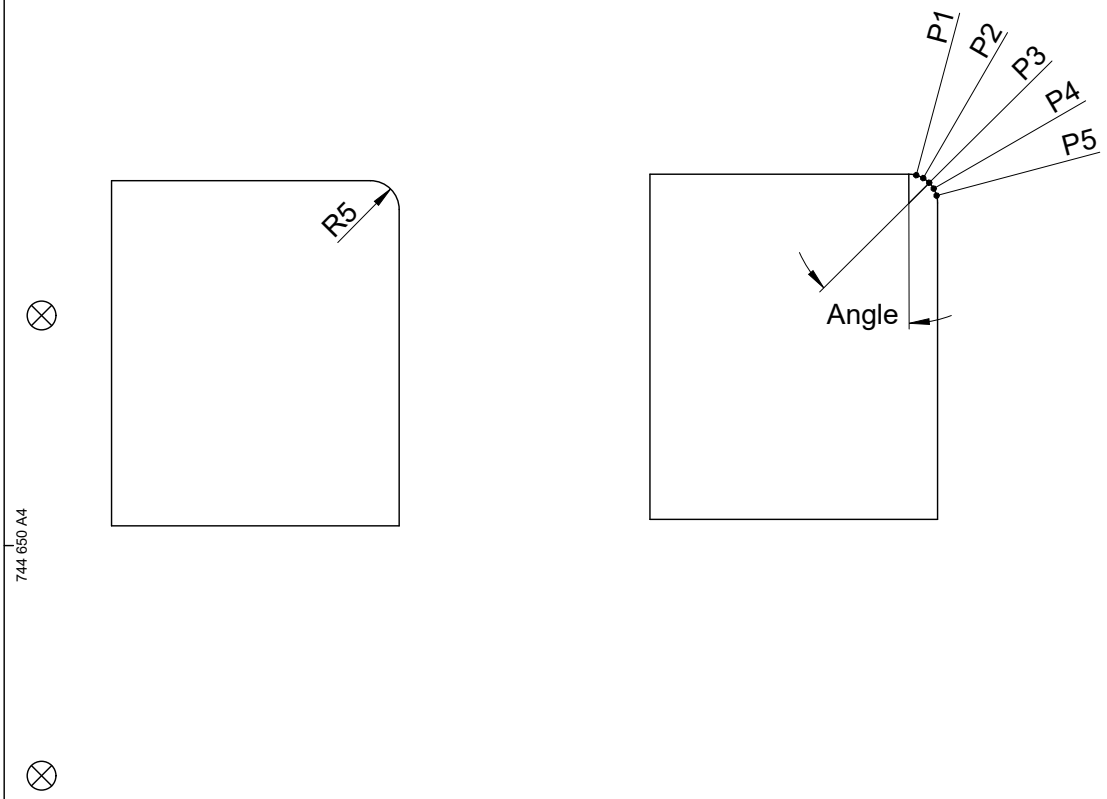
The compensated finishing tool machines another section on the radius. Roughing took place in the preceding NC program 82451_en.h.

For the 3D-ToolComp to become effective, the NC program has to be output with vectors.

Structure and program run of NC program 82453_en.h

The NC program 82453_en.h measures the machined radius at defined points. Refer to the measuring log for the measurement results.

The measuring points defined here refer to the example program. If you change the machining programs, you also have to change the coordinates of the measuring points.



744 650 A4

				ID number
Text:				Change No. C000941-10 Phase: Nicht-Serie
	Original drawing Scale / Format	Platte Plate Einzelteilzeichnung / Component Drawing		Werkstoff: Material:
RoHS	1:1 / A4			
Maße in mm / Dimensions in mm				●blanke Flächen/Blank surfaces
Werkstückkanten nach ISO 13715 Workpiece edges ISO 13715 -0.3 +0.3		Allgemeintoleranzen ISO 2768-mH ≦6mm: ±0,2 General tolerances ISO 2768-mH ≦6mm: ±0,2	Tolerierung nach ISO 8015 Tolerances as per ISO 8015 Oberflächenbehandlung: Surface treatment:	Oberflächen nach ISO 1302 Surfaces as per ISO 1302

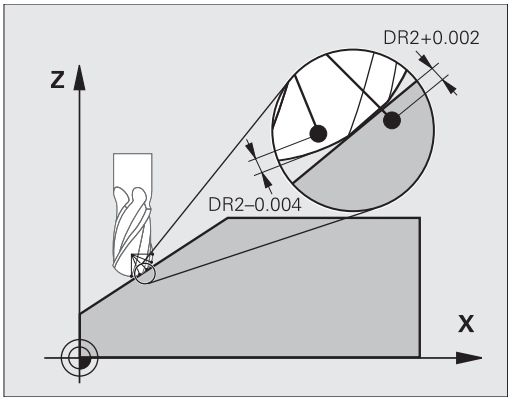
	Created	Responsible	Released	Version	Revision	Sheet	Page
HEIDENHAIN DR. JOHANNES HEIDENHAIN GmbH 83301 Traunreut, Germany	Feichtner 01.04.2019			D1284822-00-A-01 Document number			1 of 1

2

Excerpt from the User's Manual:

2.1 3-D radius compensation depending on the tool's contact angle (option 92)

Application



The effective sphere radius of a radius cutter deviates from the ideal form owing to the production process. The maximum form inaccuracy is defined by the machine tool builder. Common deviations lie between 0.005 mm and 0.01 mm.

The form inaccuracy can be saved in the form of an compensation value table. This table contains angle values and the deviation from the nominal radius **R2** measured on the respective angle value.

The **3D-ToolComp** software option (option 92) enables the control to compensate the value defined in the compensation value table depending on the actual contact point of the tool.

3-D calibration of the touch probe can also be carried out with the **3D-ToolComp** software option. During this process the deviations determined during touch probe calibration are saved to the compensation value table.

Further information: User's Manual for Setup, Testing and Running NC Programs

Requirements

To be able to use the software option **3D-ToolComp** (option 92) the control requires the following preconditions:

- Option 9 is enabled
- Option 92 is enabled
- The **DR2TABLE** column in the TOOL.T tool table is enabled
- The name of the compensation value table (without its extension) is entered in the **DR2TABLE** column for the tool to be compensated
- 0 is entered in the **DR2** column
- NC program with surface normal vectors (LN blocks)

Error compensation table

If you create the compensation value table yourself, proceed as follows:

PGM MGT

- In the file manager open the path **TNC:\system \3D-ToolComp**

NEW FILE

- Press the **NEW FILE** soft key
- Enter the file name with extension .3DTC

> The control opens a table containing the required columns for a compensation value table.

The compensation value table contains three columns:

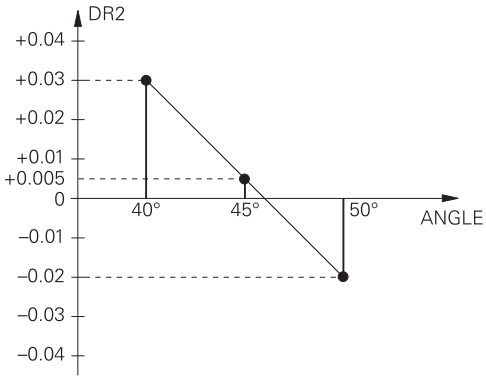
- **NR**: Consecutive line number
- **ANGLE**: Measured angle in degrees
- **DR2**: Radius deviation from the nominal value

The control evaluates a maximum of 100 lines in the compensation value table.

Function

If you are executing an NC program with surface-normal vectors and have assigned a compensation value table (DR2TABLE column) to the active tool in the tool table (TOOL.T), the control uses the values from the compensation value table instead of the compensation value DR2 from TOOL.T.

In doing so, the control takes the compensation value from the compensation value table defined for the current contact point of the tool with workpiece into account. If the contact point is between two compensation points, the control interpolates the compensation value linearly between the two closest angles.



Angle value	Compensation value
40°	0.03 mm (measured)
50°	-0.02 mm (measured)
45° (contact point)	+0.005 mm (interpolated)

Operating and programming notes:

- If the control cannot interpolate a compensation value, it displays an error message.
- **M107** (suppress error message for positive compensation values) is not required, even if positive compensation values are determined.
- The control uses either DR2 from TOOL.T or a compensation value from the compensation value table. If required, you can define additional offsets, such as a surface oversize, via DR2 in the **TOOL CALL** block.

NC program

The software option **3D-ToolComp** (option 92) only functions with NC programs containing surface normal vectors.

Pay attention when creating the CAM program how you measure the tools:

- NC program output at the south pole of the sphere requires tools measured on the tool tip
- NC program output at the center of the sphere requires tools measured on the tool center