



# HEIDENHAIN



## NC Solutions

Description of NC Program 5080

English (en)  
9/2017

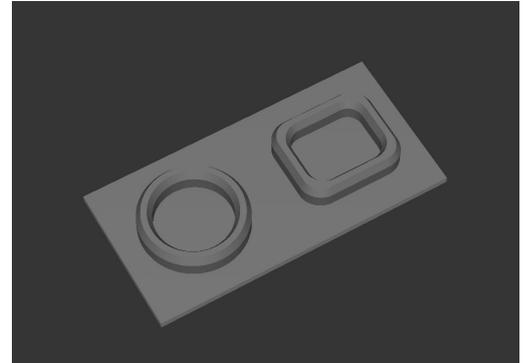
## 1 Description of NC program 5080

NC programs for machining chamfers on rectangular and circular contours.



The NC program can be run on the following controls if software option 2 (option 9) is activated:

- TNC 640
- TNC 620 as of NC software number 340 56x-03
- iTNC 530 as of NC software number 340 422-xx



### Requirement:

Use an inclined tool to mill a chamfer on a circular contour and a rectangular contour.

### Description of NC program 5080-en.h

In NC program 5080-en.h, you first define the workpiece blank and the tool. Then machining begins. Machining is defined with standard cycles. A **FACE MILLING** cycle is defined as the first machining step. Then the control machines the outside contours with an SL cycle. Three contours for roughing are defined in this cycle: A rectangular stud, a circular stud and a pocket so that the complete outside contour of the workpiece has been prepared. Then the control machines the inside contours with a **CIRCULAR POCKET** cycle and a **RECTANGULAR POCKET** cycle. The workpiece has now been completely pre-roughed.

After the workpiece has been prepared, the control shifts the datum into the center of the circle. To machine the chamfers on the circle, the control calls the NC program 50801-en.h.

The control subsequently shifts the datum into the center of the rectangle. It then calls the NC program 50802-en.h to machine the chamfers on the rectangle.

When all chamfers have been machined, the control resets the datum and ends the NC program.

### Description of NC program 50801-en.h

In NC program 50801-en.h, you first define all parameters required for the first chamfer. After that, a **TOOL CALL** block follows. Only a **DL** is defined in this tool call. The definition of the **DL** enables you to influence whether and how far the control positions the cutting edge beyond the lower edge of the chamfer.

The control then calls a subprogram. The control jumps to another subprogram—the type of subprogram depends on whether you select inside machining or outside machining in Q4. In these subprograms, **FUNCTION TCPM** is defined first. The control subsequently carries out a few calculations. It then pre-positions the tool to the calculated starting position. The control subsequently inclines the tool by the defined chamfer angle and approaches the first contour point. The control divides the circular path required for the chamfer into four circle segments. For every segment, the control moves along a circular path with an angular length of 90 degrees and also rotates the C angle by 90 degrees in the same NC block.

After the circular path has been completed, the control returns the tool to the starting point and retracts in the Z axis. Then it resets **FUNCTION TCPM**.

In the example program, a definition of the parameters and call of the subprogram follow again after the return jump to the main program, in order to execute outside machining in addition to inside machining.

After the second chamfer has also been machined, the control jumps to the end of the program. It ends the NC program with the **END PGM** NC block and returns to the calling program.

Parameter	Name	Meaning
Q1	CHAMFER LENGTH	Length of the chamfer section with respect to the circle radius
Q2	CHAMFER ANGLE	Inclination angle of the tool with respect to the Z axis
Q3	DIAMETER CIRCLE	Circle diameter
Q4	1=OUTSIDE 2=INSIDE PROCESSING	Selecting the type of machining <ul style="list-style-type: none"> <li>■ 1 = Machining on the inside</li> <li>■ 2 = Machining on the outside</li> </ul>

### Description of NC program 50802-en.h

At the beginning of the program you define the required parameters. After that, a **TOOL CALL** block is defined. Only a **DL** is defined in this tool call. The definition of the **DL** enables you to influence whether and how far the control positions the cutting edge beyond the lower edge of the chamfer.

The control then calls a subprogram. In this subprogram, the control jumps to another subprogram—the type of subprogram depends on whether inside machining or outside machining has been defined. In these subprograms, **FUNCTION TCPM** is defined first. The control subsequently carries out a few calculations. It then pre-positions the tool to the calculated starting position. The control subsequently inclines the tool by the defined chamfer angle and approaches the first contour point. The control uses linear paths to create the contour. The control calculates the end points for the straight lines of the rectangle at the beginning of the subprogram. A circular arc with an angular length of 90 degrees and a rotation of the C axis by 90 degrees are defined for each of the corner radii.

After the contour has been completely machined, the control returns the tool to the starting point and retracts it in the Z axis. Then it resets **FUNCTION TCPM**.

In the example program, a definition of the parameters and call of the subprogram follow again after the return jump to the main program, in order to execute outside machining in addition to inside machining.

After the second chamfer has also been machined, the control jumps to the end of the program. It ends the NC program with the **END PGM** NC block and returns to the calling program.

Parameter	Name	Meaning
Q1	CHAMFER LENGTH	Length of the chamfer section
Q2	CHAMFER ANGLE	Inclination angle of the tool with respect to the Z axis
Q13	LENGTH X	Length of the rectangle in the X axis
Q3	WIDTH Y	Width of the rectangle in the Y axis
Q5	CORNER RADIUS	Corner radius of the rectangle
Q4	1=OUTSIDE 2=INSIDE PROCESSING	Selecting the type of machining <ul style="list-style-type: none"> <li>■ 1 = Machining on the inside</li> <li>■ 2 = Machining on the outside</li> </ul>



