



HEIDENHAIN



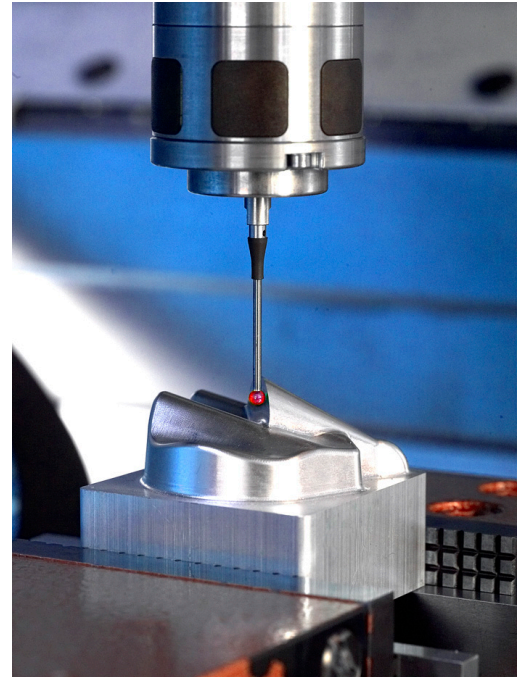
NC Solutions

Description of NC Program 8090

English (en)
02/2020

1 Description of NC programs 8090_en.h and 8095_en.h

With the NC program 8090_en.h, the control measures the respective Z coordinates in a grid of X/Y coordinates and saves all three coordinates in a log file. Subsequently, it is possible to convert the saved files into an NC program. You can run this NC program with the NC program 8095_en.h and create a copy of the measured surface.



Requirement:

In order to create a copy of a 3-D area, the Z coordinate must be determined in a grid of X/Y positions. The points measured are then used for the NC program of the copy.

Solution:

Define the distances between the measuring points in the X axis and the Y axis in the NC program. Then the control approaches the individual positions in the plane and measures the Z coordinate with the 3-D touch probe.

The control saves all three coordinates in a log. The entry is made in the HEIDENHAIN Klartext syntax for a linear block.

You can convert the created log into an NC program by copying it. With this NC program, you can machine a copy of the surface.

Surface definition and accuracy

The surface definition of the surface to be created mainly depends on the density of the grid. The narrower the rows and columns are defined, the better the surface definition. On the other hand, the machining time for measuring and milling increases because of the greater number of points.

Furthermore, the contour accuracy of the copy depends on the diameter of the tool. In order to achieve the highest possible accuracy, use a ball-nose cutter with the same radius as the ball tip radius for finishing operations on the copy.

Preparations:

Before starting the NC program, you must prepare a setting and a few files.

The control writes the points determined during the measuring program into a log file. You can define the storage location of this file in advance. Proceed as follows:

- ▶ Select the Programming/Editing or Test Run operating mode
- ▶ Press the **MOD** key
- ▶ Press the **RS232 RS422 SETUP** soft key
- ▶ For "Print" and "Print test", enter the path of the directory in which the control saves the log file
- ▶ Press the **END** soft key

Furthermore, you need to define three mask files for the log file. The control uses the FN16 function from the NC program to write the content of these masks into the log file.

The entries in the log file must correspond to the HEIDENHAIN Klartext syntax. The following three masks are required:

- BeginPGM.a file

```
"BEGIN PGM MACHINING MM";
```

```
[END]
```

- measure.a file

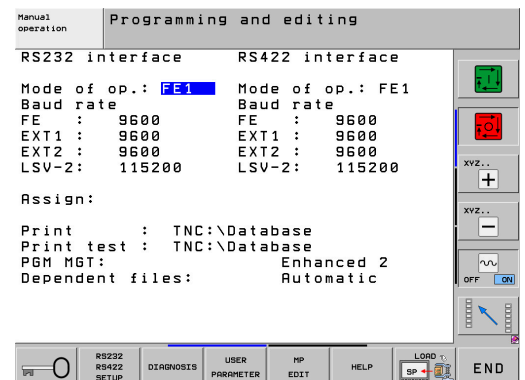
```
"L X%6.4LF Y%6.4LF Z%6.4LF R0",Q1,Q2,Q3;
```

```
[END]
```

- ENDPGM.a file

```
"END PGM MACHINING MM";
```

```
[END]
```



NC program 8090_en.h:

In the NC program 8090_en.h, the control calculates the measuring positions, measures the respective Z coordinate, and writes the log file.

First of all, the command FN16 is defined in the NC program. With this command, the control writes the content of the mask file BeginPGM.a into the log file. In this command, you must adapt the path of the mask file and the name of the log file to be created.

Then the control calls the 3-D touch probe.

Subsequently, define the start coordinate of the grid in the X axis in the Q parameter Q1. This is not the first measuring point but the position for the pre-position. For the first measuring point, an offset to the measuring point distance is still required.

Then define the Y coordinate of the first measuring row in the Q parameter Q2. Then the LBL 1 jump label is defined in the NC program.

In the first step of the repetition, the control adds the increment in X to the X position Q1. Define this increment directly in the function. Then the test system cycle is defined. The control measures the Z coordinate with this cycle at the calculated X position and the defined Y position. The control saves the measurement result in the Q parameter Q3.

Subsequently, an FN16 function is defined again. With this function, the control writes the content of the measure.a mask file into the log file. While doing so, the control fills the placeholders in the mask file with the values that are saved in the Q parameters Q1 to Q3. In the FN16 function, you need to adapt the path of the mask file and the name of the log file again. It is important to define the same log file as in the first NF16 function.

The next step in the NC program is a program section repeat. Thereby the control repeats the steps "calculate, measure, and write a new X position". The numbers of repetitions must be defined in a way that the sum of steps in the X axis corresponds to the measured surface. In the example, +5 mm is defined for each step in X. The control repeats this step 18 times, so the last measured X coordinate lies at X+95.

The first row of the grid is measured when the defined repetitions have been executed. Then the Y coordinate for the second row is calculated. For this purpose, the control adds the Y step to the current Y coordinate. Define also this increment directly in the function.

Subsequently, the control measures the Z coordinate at the newly calculated position and writes all of the coordinated into the log. Then another step is executed in the X axis, the value of which you also need to enter into the function. In a loop, the control repeats the measurement, the writing into the log, and the X step. You must define the number of repetitions in a way that the last calculated X coordinate corresponds to the first measured X coordinate.

When the number of repetitions has been reached, another step in Y direction is defined in the NC program. Enter the increment directly into the calculation again. Then another program section repeat is defined so that the control scans the entire workpiece in a meandering tool path. For this purpose, you must define the

number of repetitions in a way that the Y coordinate corresponds to the workpiece.

After the control has measured all positions and written all coordinates into the log, it enters the content of the ENDPGM.a mask file into the log. In this FN16 command, you must also adapt the path of the mask file and the name of the log file. The control then ends the NC program.



When the NC program 8090_en.h starts and the log file exists, the control adds all of the entries to the existing file at the end.



If no .TXT file is defined in the FN16 command, the control writes the data into the %FN16RUN.A file during program run and into the %FFN16SIM.A file during test run.

Log file surface_data.txt

After executing the complete NC program, the log file created by the NC program 8090_en.h contains all of the determined coordinates as well as the syntax for the program start and the program end.

```

01011: surface_data.txt          Z010: 12      SpN10: 1      NBER:
BEGIN PGM MACHINING MM
L X5.0000  V0.5000  Z-1.4956  R0 F300 M13
L X10.0000 V0.5000  Z-3.4347  R0 F300 M13
L X15.0000 V0.5000  Z-5.3606  R0 F300 M13
L X20.0000 V0.5000  Z-7.2882  R0 F300 M13
L X25.0000 V0.5000  Z-9.1574  R0 F300 M13
L X30.0000 V0.5000  Z-11.0635 R0 F300 M13
L X35.0000 V0.5000  Z-13.0096 R0 F300 M13
L X40.0000 V0.5000  Z-15.0368 R0 F300 M13
L X45.0000 V0.5000  Z-16.9741 R0 F300 M13
L X50.0000 V0.5000  Z-18.9075 R0 F300 M13
L X55.0000 V0.5000  Z-20.8426 R0 F300 M13

```

Converting a log file into an NC program

In order to convert the log file into an NC program, proceed as follows:

- ▶ Press the **PGM MGT** key
- ▶ Select log file
- ▶ Press the **COPY** soft key
- ▶ Enter the program name with the extension .H into the pop-up window (MACHINING.h in the example)
- ▶ Press the **OK** soft key
- ▶ The control saves the NC program as a new file.

```

0  BEGIN PGM MACHINING MM
1  L  X+5  Y+0.5  Z-1.4956  R0 F300 M13
2  L  X+10 Y+0.5  Z-3.4347  R0 F300 M13
3  L  X+15 Y+0.5  Z-5.3606  R0 F300 M13
4  L  X+20 Y+0.5  Z-7.2882  R0 F300 M13
5  L  X+25 Y+0.5  Z-9.1574  R0 F300 M13
6  L  X+30 Y+0.5  Z-11.0635 R0 F300 M13
7  L  X+35 Y+0.5  Z-13.0096 R0 F300 M13
8  L  X+40 Y+0.5  Z-15.0368 R0 F300 M13
9  L  X+45 Y+0.5  Z-16.9741 R0 F300 M13
10 L  X+50 Y+0.5  Z-18.9075 R0 F300 M13
11 L  X+55 Y+0.5  Z-20.8426 R0 F300 M13

```

NC program 8095-en.h

The NC program 8095_en.h serves to machine a workpiece with the coordinates measured and saved in the NC program MACHINING.h. In the NC program 8095-en.h, the control first calls the milling cutter. In this example, the milling cutter is a ball-nose cutter with the same diameter as the ball tip of the touch probe that was used to measure the coordinates.

Then Cycle 30 **RUN CAM DATA** is defined. In this cycle, the NC program MACHINING.h is defined as an NC program to be called. Further entries in the cycle must be adapted to your workpiece accordingly. Subsequently, the control calls the cycle and machines the workpiece. Then the NC program ends.

NC program 8095-en-callpgm.h

In the NC program 8095-en-callpgm.h, a second possibility exists to execute the NC program MACHINING.h that resulted from the sequence of measurement. Again, the control first calls the milling cutter. Alternatively to execution with the cycle, the control calls the **CALL PGM** command in this NC program. After the called NC program has been executed, the NC program 8095-en-callpgm.h ends.