



HEIDENHAIN



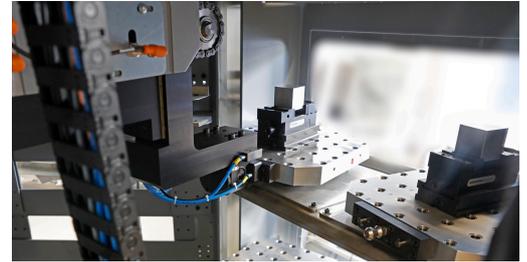
NC solutions

Description of NC program 8200

English (en)
10/2019

1 Description of NC program 8200_en.h

NC program for measuring a Z coordinate and automatically calling a further NC program. The control selects the NC program to be called based on the Z coordinate to be measured.



Requirement:

Your task is machining five different workpieces. The workpiece blanks differ only in the Z coordinate and are delivered unsorted.

A machine tool with a pallet changer is available for accomplishing the task. To save time, the machine operator does not need to sort the workpiece blanks. The machine operator simply loads the different workpiece blanks onto the pallets and starts machining.

Depending on the workpiece blank, the control is to select the appropriate NC program and start the machining operation.

In addition, errors caused by inaccurate workpiece blanks are eliminated by machining only workpiece blanks that are within a defined tolerance range.

Solution:

After inserting the pallet into the working space, the control measures the Z coordinate of the workpiece blank with a 3-D touch probe.

Depending on the Z coordinate measured, the control selects the NC program that you previously assigned to this workpiece blank size.

Then the control checks whether the workpiece blank is within the defined tolerance range. If so, then the control starts the machining operation. If the workpiece blank is outside the tolerance, then the control initiates the pallet change to the next pallet without starting the machining operation.



You do not need a pallet system for this application. To implement this application, you need to modify the NC program such that—instead of a pallet change—the operator is enabled to open the machine door and exchange the workpiece.

NC program 8200_en.h:

The NC program 8200_en.h starts with a program jump. The control uses this program jump to skip the next program section.

The second program section, which is first skipped, consists of five subprograms. A **CALL PGM** is defined in each of these subprograms. In these commands, you need to adjust the path to the individual machining programs for the components. Start with the NC program 1 describing the component with the largest Z coordinate. Then continue defining the NC programs in descending order down to NC program 5 that describes the component with the smallest Z coordinate.

The control skips this program section, because the program calls are executed only if necessary. The section was structured in this way to make sure that all of the necessary settings are summarized at the beginning of the program.

An LBL 201 is defined after the five subprograms. This is the jump label for skipping the program definition.

In the next program section, you define five Q parameters with the values of the different Z coordinates. Here again, it is important to begin with the largest coordinate. Then continue with defining the further coordinates successively. You need to enter the values such that the mean tolerance range is defined.

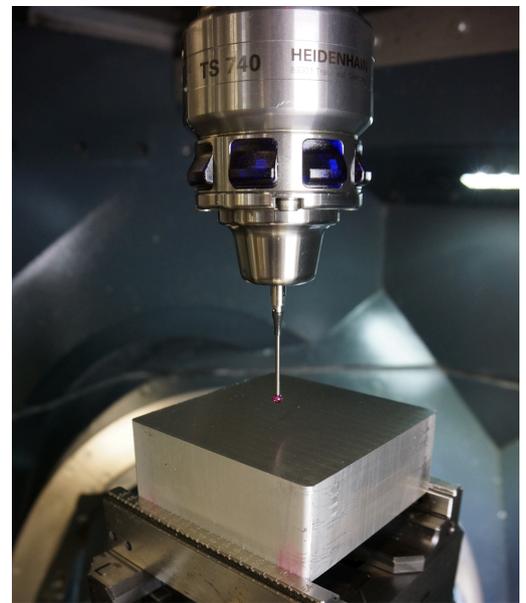
Then you define the permissible tolerance for the workpiece blanks. The control checks the workpiece blanks for this tolerance in both the positive and the negative directions relative to the defined Z coordinate.

The last parameter you define is the number of program section repeats.

The jump label for the program loop is defined next. The call of NC program 82006_en.h is the first step in the loop. The pallet change is programmed in this NC program. After the NC program called has been run, the control resets the working plane with the function **PLANE RESET**. A tool call for the 3-D touch probe is programmed next.

In the next step, the control activates the preset. In the example program, this preset is at the center of the workpiece in the X/Y plane, and at the bottom of the workpiece in the Z axis.

The control then pre-positions the 3-D touch probe. After this, a measuring cycle is defined. With this cycle, the control measures the Z coordinate of the workpiece surface. You may need to modify the parameters in this cycle. The control saves the measurement result to the parameter Q90.



In the example program, a slash (/) is used to hide the cycle. When running the NC program, you have to remove the slash (/) or ensure that the hide function is inactive.

In Test Run mode, the measuring cycle does not provide a measurement result. A definition of Q90 is programmed following the measuring cycle so that the NC program can be checked in Test Run mode. You can enter different values for simulating the measurement result.



During NC program run, the control overwrites the measurement result with the function. The function therefore needs to be hidden or deleted from the NC program.

Then the control starts with selecting the appropriate NC program. The control first calculates the minimum Z coordinate of program 1 by subtracting the tolerance from the specified mean value. Then the control checks whether the measured coordinate is greater than the minimum Z coordinate calculated for program 1.

If the measured value is greater, the control jumps to the subprogram PROG_1. If the measured value is smaller, the control applies the same procedure to the programs 2 to 5. If the control cannot assign an NC program to the measured value, it jumps to the "end" label.

In the subprograms PROG_1 to PROG_5, the control jumps to another subprogram where it checks whether the measured value is lower than the upper tolerance of the respective workpiece blank.

If the measured value is outside the tolerance, the control jumps to the "end" label. If not, the control jumps to the subprogram where it uses the function **CALL PGM** to call the corresponding NC program.

After running the part program, the control jumps to the "end" label. The call of the program section repeat is programmed after the jump label. If the number of repeats has not been reached, the control jumps to the program call for the pallet change. If the number of repeats has been reached, the control uses the command M30 to end the NC program.

NC programs 82001_en.h to 82005_en.h

In this example, the NC programs 82001_en.h to 82005_en.h are the programs for machining the different workpiece blanks. Since workpiece machining is not essential for the actual program run, the programs only contain different BLK forms and a programmed stop. This stop helps you verify in Test Run mode or on a programming station that the jumps were correctly performed.

NC program 82006_en.h

If a pallet needs to be changed in the main program, the control calls the corresponding NC program. The pallet change varies depending on the machine and cannot be shown on the programming station. This is why you only need to program a stop in the NC program.