HEIDENHAIN

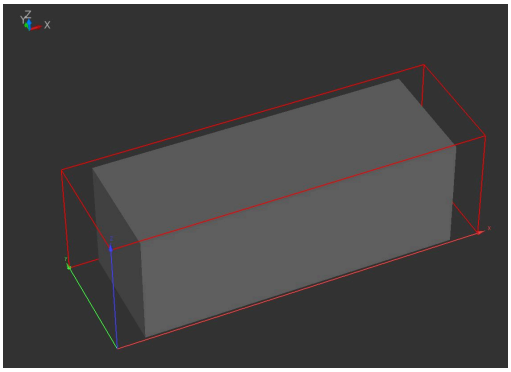
NC Solutions

Description of NC Program 4220

English (en)
3/2020

1 Description of NC program 4220_en.h

NC program for machining a cuboid from five sides.



Requirement

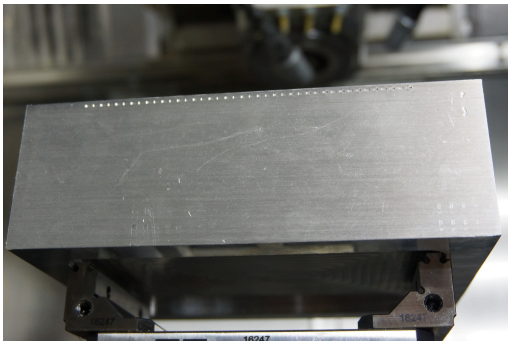
The requirement was to machine a cuboid workpiece blank from five sides. The machine operator only wants to define the workpiece blank and the machined part. The control calculates the dimensions of the individual surfaces and the allowance on the surfaces. Subsequently, the control machines the surfaces.

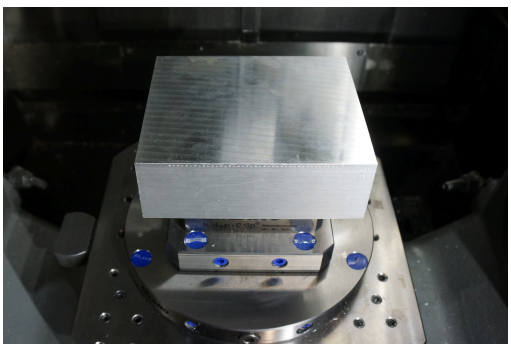
Preparation

The workpiece must be clamped on the bottom so that the control can completely machine the upper and lateral surfaces. Since an overrun is used on the surfaces, you must leave clearance for this.

The preset is at the center of the workpiece blank in the X/Y plane and on the bottom of the workpiece blank in the Z axis.

The control must work with spatial angles when tilting the working plane.





NC program 4220_en.h

At the beginning of the NC program, define all parameters required for the machining process. Then the control calculates further required values. First, it calculates the dimensions of the workpiece blank starting from the preset. Then it calculates the allowances within the individual axes and the target dimensions starting from the preset. In the last block of the calculations, the control determines the coordinates for pre-positioning in the individual axes.

The next step in the NC program is the blank form. The control defines the blank form with the workpiece blank dimensions previously calculated. Then the control calls the tool. This requirement has also been met with the Q parameter programming. This means that you conduct the tool definition in the input block at program start. Then the control switches on the spindle.

Subsequently, the control checks whether the workpiece blank is larger than the machined part in the X axis:

- If the workpiece blank is larger than the machined part, machining is required and the control jumps to the **LBL 1** subprogram.
- If the workpiece blank is not larger than the machined part, machining is not required and the NC program continues without jump

A jump label is defined after the check. This jump label is required to jump from the **LBL 1** subprogram back to the main program.

Then the control checks whether the workpiece blank is larger than the machined part in the Y axis:

- If the workpiece blank is larger than the machined part, machining is required and the control jumps to the **LBL 2** subprogram.
- If the workpiece blank is not larger than the machined part, machining is not required and the NC program continues without jump

A jump label is also defined after this check. The jump label is required to jump back from the **LBL 2** subprogram back to the main program.

In the next step, the control checks whether the workpiece blank is larger than the machined part in the Z axis:

- If the workpiece blank is larger than the machined part, machining is required and the control jumps to the **LBL 5** subprogram.
- If the workpiece blank is not larger than the machined part, machining is not required and the NC program continues without jump

For jumping back from the **LBL 5** subprogram, a jump label is defined as well.

In the last step of the main program, the control retracts the tool and ends the NC program.

Subsequently, the subprograms are defined for the machining processes in the individual axes.

The **LBL 1** and **LBL 2** subprograms for machining in the X axis and the Y axis have the same structure and run as follows:

- Shift datum in positive direction to the finished part edge
- Move to safe position in the Z axis
- Move to tilt position in the X axis and the Y axis
- Tilt the working plane
- Pre-position in the X axis
- Pre-position in the X axis and the Y axis
- Define Cycle 232 FACE MILLING with the calculated values
- Call the cycle
- Retract the tool in the tool axis
- Move to tilt position in the X axis and the Y axis
- Reset tilting the working plane function
- Shift datum in negative direction to the finished part edge
- Move to tilt position in the X axis or in the Y axis
- Tilt the working plane
- Pre-position in the X axis
- Pre-position in the X axis and the Y axis
- Define Cycle 232 FACE MILLING with the calculated values
- Call the cycle
- Retract the tool in the tool axis
- Move to tilt position in the X axis and the Y axis
- Reset tilting the working plane function
- Cancel datum shift
- Jump back to the main program

The subprogram for machining in the Z axis runs as follows:

- Define Cycle 232 FACE MILLING with the calculated values
- Call the cycle
- Retract the tool in the tool axis
- Jump back to the main program

The control calculates the coordinates required for the subprograms from the parameters defined by you. If required, you must adjust the tilt positions and the tilting behavior for your machine.

Parameter	Name	Meaning
Q1	FINISHED DIMENSION IN X	Absolute length of the machined part in the X axis
Q2	FINISHED DIMENSION IN Y	Absolute width of the machined part in the Y axis
Q3	FINISHED DIMENSION IN Z	Absolute height of the machined part in the Z axis
Q4	BLANK DIMENSION IN X	Absolute length of the workpiece blank in the X axis
Q5	BLANK DIMENSION IN Y	Absolute width of the workpiece blank in the Y axis
Q6	BLANK DIMENSION IN Z	Absolute height of the workpiece blank in the Z axis
Q9	TOOL NUMBER	Number of the milling cutter to be used
Q10	SPEED RPM	Speed of the tool spindle during machining
Q11	FEED RATE FOR MILLING	Traversing speed of the tool during roughing
Q15	PLUNGING DEPTH	Dimension in the tool axis by which the control moves the tool between two machining steps.
Q16	FINISHING ALLOWANCE FOR DEPTH	Finishing allowance in the tool axis
Q17	FINISHING FEED RATE	Traversing speed of the tool during finishing operation
Q18	SET-UP CLEARANCE	Distance between tool tip and workpiece surface
Q19	CLEARANCE HEIGHT DURING PRE-POSITIONING	Safe coordinate in the Z axis that the control moves to before pre-positioning