Lab 1 Report

ME 597: Precision Motion Generation

Oregon State University

15 April 2019

Paresh Soni	sonip@oregonstate.edu
Bradley Anderson	anderbra@oregonstate.edu
Nizam Hakim Md Ishak	mdishakn@oregonstate.edu

Abstract

This lab demonstrates the essential operation of a Computer Numerical Control (CNC) Vertical Machining Center (VMC). The operation includes machine zeroing, part touch-off, and tool path programming.

Procedure

Before ever touching a VMC, one must have a design in mind and then use a simulator to generate the toolpath for that design. Once the G code is generated and verified in the simulator, then it is verified on the machine. Then homing of the machine to the correct coordinate system is done, and then the program is run. Once the part is machined, the machine must be cleaned up.

The stock to be machined is a 2.5″ by 4.0″ high-density polyethylene (HDPE). The maximum feed rate is 12.0 [inch/min]. The maximum depth of cut into the part is 0.5″. A minimum of 0.2″ clearance is left between the edge of the stock and the face of any cuts into the stock.

Figure 1: Blank work-piece

Design

By the requirements of the lab, the design to be manufactured must contain both linear and circular interpolations. These requirements will meet with an essential representation of the text, "Lab1".

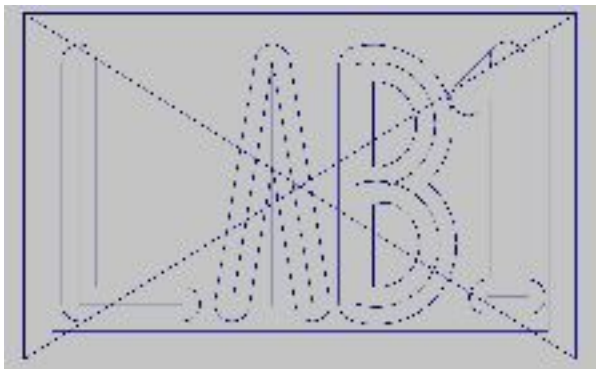

Figure 2: Design

This basic design will be converted to code. AutoCAD is used to generate coordinates of the tool paths. The tool used has a diameter of 0.250″, so slots of this width can be machined using only the centerline of the slots.

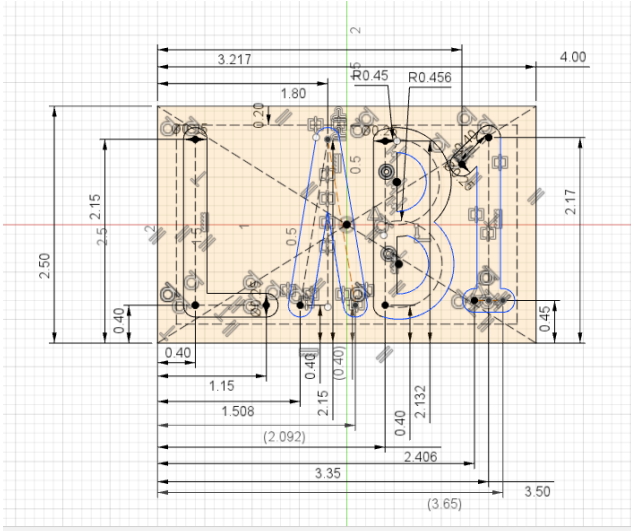

Figure 3: Design dimension

With these dimensions, the designer can start writing G code. The tool path is shown in Appendix I.

G-Code Generation

For a design as simple as 8 linear interpolations and two circular interpolations, the code can be written manually. Using a simulator to view the code while it is being written is especially helpful for simple parts.

Code is written line by line in https://ncviewer.com/ to debug syntax. The primary purpose of the G code viewer is to ensure that the G code is fundamentally correct (appendix II).

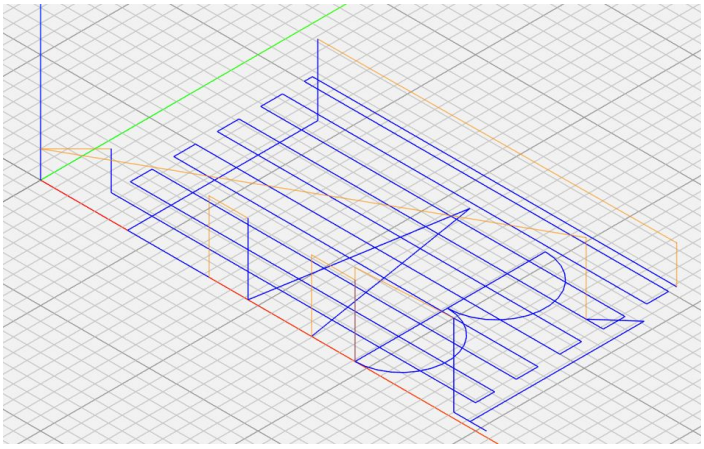

Figure 4: G-code simulation

In addition to the simple text, a series of facing passes remove the entire top surface to ensure the top surface is flat, within the precision of the machine.

Machine Verification

Once the G code is generated and verified on a simulator, it can be verified on the machine (appendix III). This is important because many simulators are extremely robust and will accept any formats and syntax. However, the machine that does the milling is often more particular about how the code is formatted.

CNC machines typically have a keyboard and screen where G code can be displayed how it is interpreted by that specific machine before running the program with physical tools. The G code is plotted on the machine, and any errors in the auto plotter are corrected.

The primary purpose of machine verification is to ensure that the machine interprets the G code correctly (Appendix IV).

Machine Zeroing

Once it is verified that the machine interprets the G code correctly, the operator must ensure that the machine will execute the commands correctly.

When the machine is powered off, it forgets its absolute positioning. Ensuring that the machine knows where the part is and where the tool is in physical space are essential steps before any machining can occur.

Typically, the operator will measure the length of each tool being used and save that as a Tool Offset, as well as the height of the fixture and material to be machined and save that as a Fixture Offset. However, when only using one tool, the operator can simply measure tell the machine the distance from the end of the tool to the origin of the part from the G code.

For this lab, the tool center point of the tool is moved to the origin of the G code, relative to the part to be machined, and the machine coordinates are set to <x, y, z> = <0,0,0>. This enables the machine to follow the G code with the correct reference frame.

Machine Operation

Before operating the machine, the operator is positioned to Run Ready Mode, where one thumb rests on the Hold button and one thumb rests on the Start button, just in case the operator needs to pause the machine for any reason.

This specific machine does not have to control over the spindle speed - only feed rates. The operator turns on the spindle and turns the dial to the appropriate speed.

Once the G code is prompted into the machine, the operator presses the Start button to tell the machine to start cutting.

While the machine runs, the operator can adjust the feed overrides to allow the machine to feed at various factors of the programmed feed rate, from 1% to 100%. This allows the operator to closely follow the operation of the machine and ensure the tool and the part are not damaged.

Results

Several issues were encountered during this lab. A few syntax errors needed to be fixed in the G code. The machine was not properly zeroed to the part. A couple of adjustments could be made to the G code itself.

First, the G code did not work immediately with the VMC. When the G code was written, the feed rate was specified in G00 commands. This was not an issue in the simulator, but the machine threw an error. All feed rates were removed from G00 commands and moved to G01 commands. Also, in circular interpolations, the radius and Y coordinate were specified, but not the X coordinate because the end position should not change in the X. However, the machine required both X and Y coordinates for the final position, even if it was the same as the original position.

Second, machine zeroing is typically obtained with an edge finder. However, for this lab, the tool was lined up with the origin of the part by eye. This introduced large error into the position of the cutting into the stock. This can be observed by comparing the clearance between the cuts into the part and the edge of the stock on the right and left sides of the finished part.

Third, the finish of the part could be improved by G code. The surface finish of the part clearly shows burrs from the surfacing pass. Also, every notch is left on the left side of the, “L”, and on the right side of the, “1”. These notches are from the method of surfacing.

Figure 5: Finished work-piece

Conclusion

Several points can be drawn from this lab. G code can be interpreted in many ways when the syntax is not overly explicit. Therefore, it can save time to be overly explicit with every command so that the compatibility with various machines is maximized. Also, when precise positioning of cuts into stock is important, and edge finder and 1x2x3 block should be used to dial in tool and fixture offsets. Finally, special attention should be paid to the way G-Code is written. Facing and finishing passes are important and easy to introduce errors in manufacturing. One should ensure that every tool path is necessary and consider the width of the tool when deciding if any passes are missing.

Appendix I: Tool path form drawing

Points	X	Y
P01	0.325	0.325
P02	3.675	0.325
P03	3.675	0.5
P04	0.325	0.5
P05	0.325	0.7
P06	0.3675	0.7
P07	0.3675	0.9
P08	0.325	0.9
P09	0.325	1.1
P10	0.3675	1.1
P11	0.3675	1.3
P12	0.325	1.3
P13	0.325	1.5
P14	0.3675	1.5
P15	0.3675	1.7
P16	0.325	1.7
P17	0.325	1.9
P18	0.3675	1.9
P19	0.3675	2.1
P20	0.325	2.1
P21	0.325	2.175
P22	0.3675	2.175
P23	0.4	2.15
P24	0.4	0.4
P25	1.15	0.4
P26	1.508	0.4
P27	1.8	2.15
P28	2.092	0.4
P29	2.492	0.4
P30	2.492	1.25
P31	2.492	2.15
P32	2.492	0.4
P33	3.35	0.45
P34	3.65	0.45
P35	3.5	0.45
P36	3.5	2.05
P37	3.217	1.8

Points	X	Y	Circle center
P30	2.492	1.25	c(2.492, 0.825)
P31	2.492	2.15	c(2.492, 1.7)

Appendix II: Pre-lab G-code

```
N10 G90 G80 G40 G70
N20 G53 Z0.0
N30 G54
N40 M6 H1
N50 G00 X0.0 Y0.0 Z0.25
N60 G00 X0.325 Y0.325
N70 F2.0 S1000 M3
N80 G01 Z-0.1
N90 X3.675 F12
N91 Y0.5
N92 X0.325
N93 Y0.7
N94 X3.675
N95 Y0.9
N96 X0.325
N97 Y1.1
N98 X3.675
N99 Y1.3
N100 X0.325
N101 Y1.5
N102 X3.675
N103 Y1.7
N104 X0.325
N105 Y1.9
N106 X3.675
N107 Y2.1
N108 X0.325
N109 Y2.175
N110 X3.675
N120 G00 Z0.25 F2N121 X0.4 Y2.15
N121 X0.4 Y2.15
N122 G01 Z-0.4
N123 Y0.4 F12
N124 X1.15
N130 G00 Z0.25 F2
N131 X1.508
N132 G01 Z-0.4
N133 X1.8 Y2.15 F12
N134 X2.092 Y0.4
N140 G00 Z0.25 F2
N141 X2.492
N142 G01 Z-0.4
N143 G03 Y1.25 R0.425 F12
N144 G03 Y2.15 R0.45
N145 G01 Y0.4
N150 G00 Z0.35 F2
N151 X3.35 Y0.45
N152 G01 Z-0.4
N153 G01 X3.65
N154 G01 X3.50
N155 G01 Y2.05
N156 G01 X3.217 Y1.8
N160 G00 Z0.25 F2
N161 M05
N162 G53  Z0.25
N163 G00 X0.0 Y0.0
N170 M30
```

Appendix III: G-code before machine corrections

```
N10 G90 G80 G40 G70
N20 G53 Z0.0
N30 G54
N40 M6 H1
N50 G00 X0.0 Y0.0 Z0.25
N60 G00 X0.325 Y0.325
N70 F2.0 S1000 M3
N80 G01 Z-0.1
N90 X3.675 F12
N91 Y0.5
N92 X0.325
N93 Y0.7
N94 X3.675
N95 Y0.9
N96 X0.325
N97 Y1.1
N98 X3.675
N99 Y1.3
N100 X0.325
N101 Y1.5
N102 X3.675
N103 Y1.7
N104 X0.325
N105 Y1.9
N106 X3.675
N107 Y2.1
N108 X0.325
N109 Y2.175
N110 X3.675
N120 G00 Z0.25 F2
N121 X0.4 Y2.15
N122 G01 Z-0.4
N123 Y0.4 F12
N124 X1.15
N130 G00 Z0.25 F2
N131 X1.508
N132 G01 Z-0.4
N133 X1.8 Y2.15 F12
N134 X2.092 Y0.4
N140 G00 Z0.25 F2
N141 X2.492
N142 G01 Z-0.4
N143 G03 Y1.25 R0.425 F12    (No X)
N144 G03 Y2.15 R0.45    (No X)
N145 G01 Y0.4
N150 G00 Z0.35 F2
N151 X3.35 Y0.45
N152 G01 Z-0.4
N153 G01 X3.65
N154 G01 X3.50
N155 G01 Y2.05
N156 G01 X3.217 Y1.8
N160 G00 Z0.25 F2
N161 M05
N162 G53  Z0.25
N163 G00 X0.0 Y0.0
N170 M30
```

Appendix IV: Final G-code

```
N10 G90
N11 T1
N50 G00 X0.0 Y0.0 Z0.25
N60 G00 X0.325 Y0.325
N70 F2.0
N80 G01 Z-0.1
N90 X3.675 F12
N91 Y0.5
N92 X0.325
N93 Y0.7
N94 X3.675
N95 Y0.9
N96 X0.325
N97 Y1.1
N98 X3.675
N99 Y1.3
N100 X0.325
N101 Y1.5
N102 X3.675
N103 Y1.7
N104 X0.325
N105 Y1.9
N106 X3.675
N107 Y2.1
N108 X0.325
N109 Y2.175
N110 X3.675
N120 G00 Z0.25
N121 G00 X0.4 Y2.15
N122 G01 Z-0.4 F10.
N123 G01 Y0.4 F12.
N124 X1.15
N130 G00 Z0.25
N131 G00 X1.508
N132 G01 Z-0.4
N133 G01 X1.8 Y2.15 F12
N134 X2.092 Y0.4
N140 G00 Z0.25
N141 X2.492

N142 G01 Z-0.4
N143 G03 X2.492 Y1.25 R0.425 F12
N144 G03 X2.492 Y2.15 R0.45
N145 G01 Y0.4
N150 G00 Z0.35
N151 X3.35 Y0.45
N152 G01 Z-0.4
N153 G01 X3.65
N154 G01 X3.50
N155 G01 Y2.05
N156 G01 X3.217 Y1.8
N160 G00 Z0.25
N162  Z0.25
N163 G00 X0.0 Y0.0
N170 M30
```