MEMORANDUM

To: Trian Georgeou

From: Robyn Ribet

Date: October 26, 2018

Subject: IME 335 Business Card Holder Operation #1

Introduction

CNC manufacturing has been established and developed to expedite the manufacturing process and save time and money. Compared to manual machining, CNC manufacturing allows for more complex parts, improved tool life, closer tolerances, and part repeatability. Manual programming of these machines is a basic alternative to Computer Aided Machining programs which generate the G&M code for the manufacturer. The objective of this project was to perform operation #1 of a business card holder using manual programming of G&M code. Through this project, operators gained an understanding of the basic commands to run a machine as well as differentiating between G&M codes, canned cycles, and tool changes.

This project was run on a Haas VF-2 Mill and most of the manufacturing constraints were defined in the problem statement including part dimensions, machine tools, stock size, and tooling. These constraints simplified the manufacturing challenge and focused the operator's time on controlling the CNC machine and generating the G&M code for the operation.

Throughout this project, each step in the process was well documented to demonstrate expectations of a professional environment and provide necessary information for following steps. Each of the documentation steps are explained and attached below.

Methodology

Engineering Drawing Creation

Engineering Drawings communicate part dimensions and tolerances between engineers and manufacturers to ensure parts are properly manufactured for their specific use. They can be created through different CAD programs, but all need to effectively communicate part details to achieve the desired resulting part.

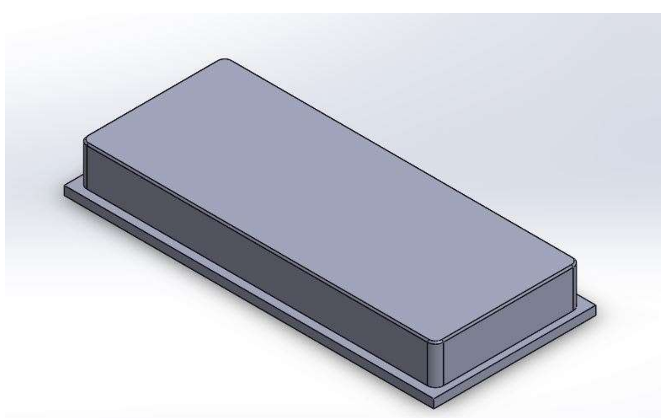


Figure 1 - SolidWorks solid model of operation #1 machined part.

The first step in this process is creating a solid model of the part in a CAD program, for this project SolidWorks was used. See **Figure 1** for a picture of the modeled part. From the solid model, an engineering drawing was created. **Attachment A** is the engineering drawing for this operation, created in SolidWorks, and shows the final part dimensions and geometric and dimensional tolerancing. The part will be inspected after manufacturing is completed to ensure that the final product has met the required GD&T.

Job Planning / Part Routing

For the complete process of making a part, several operations are required which include both the manufacturing and part inspection. Operations, tooling, and machines used are specified in the job planning or part routing documents. Specifically, for this part, there is one vertical bandsaw operation, two Haas VF-2 milling operations, and two inspection operations (See **Attachment B**). This document is critical for the general overview of the part production and operations.

The Job Planning includes selection of the raw material, operations, and tooling. The final part dimensions of 5.70” x 2.50” x .74” allow for the use of 2.50” x .75” bar stock cut to 5.70” length. Operations for the job include cutting the stock to length, milling out the outside contour, inspecting the part, milling the inside pocket, ramp, and engraving, and performing the final inspection of the part. These operations, a given constraint, were chosen to achieve the necessary part dimensions and contours. The inspections ensure that the critical dimensions and tolerances were reached before the next operation begins.

Tool Path Drawing Creation

Tool path drawings provide location information of the tool before and after every curve and straight-line path to simplify the G&M code creation process. There is one drawing for each tool being used to keep the information easy to read. They establish both lead-ins and lead-outs of tools to verify logical tool paths throughout the operation. The toolpath drawings ensure the proper tool placement with relation to the part to minimize error in depths of cut and stepover. **Attachment C-E** include the three toolpath drawings required for operation #1 of the business card holder. In these drawings is an isometric view of the part, top view of the part, and labeled circles for each step of the tool path. Finally, there is a table that lists the x and y coordinates of each tool placement relative to the G54 work coordinate system. This method makes the use of absolute (G90) coordinate systems easy for G&M code programming.

The first operation tool path, facing off the top of the part, is shown in **Attachment C**. This is a simple linear interpolation with the 3” face mill. The drawing shows the starting and ending positions of the face mill and lists the coordinates for those locations relative to the G54 work coordinate system. The second tool path drawing, **Attachment D**, includes the tool path drawing for the contour with the ¾” end mill. The tool path of the end mill is more complex and includes linear interpolation as well as circular interpolation. Using SolidWorks to create the drawing allows for easy calculation of coordinates of each position relative to the G54 work coordinate system. The radius value required for the circular interpolation is easily measured using SolidWorks which makes the G&M code creation simpler. Finally, **Attachment E** is the tool path drawing for the spot drill, which created the chamfer. Because only a certain part of the spot drill interfaced with the part, the tool path drawing for this part shows two circles: one for the diameter of the tool that cuts the part (smaller circle) and one for the actual diameter of the spot drill (larger circle). This allowed for easily seeing where the actual tool path would be as well as calculating the placement of the center of the tool.

G&M Code Creation

G&M Code is uploaded to the CNC machines to determine the position of the tool and initiate actions such as coolant, tool changes, and spindle startup. It can be either programmed manually or generated from Computer Aided Manufacturing programming. Understanding how G&M Code works is a critical step in becoming and effective and informed machinist. G codes are preparatory codes and provide information to the machine such as the coordinate system (absolute versus incremental), type of movement (rapid versus interpolation), or tool offsets. M codes are for initiating specific actions such as coolant, spindle rotation, and tool changes.

Several steps are involved to developing the G&M Code: feed and speed calculations, start up and ending lines, linear and circular moves, and code simulation. Using the correct feeds and speeds for tools is important to preserve tool life, minimize production time, and optimize surface finish. The feeds and speeds of the machine are determined from the tool diameter, material properties, and desired surface finish. The specific feeds and speeds for operation #1 are listed in **Figure 2**. RPM and Feed Rate are calculated using the following formulas:

$$RPM = \frac{3 * SURFACE\ SPEED}{\pi * TOOL\ DIAMTETER} \qquad FEED\ RATE = RPM * CHIP\ LOAD * \#\ OF\ TEETH$$

Surface speed (ft/min) and chip load (in/tooth) are values found in tables and are determined by tooling and part material. Tool diameter (inches) is the diameter of the surface that interfaces with the part. For example, in the case of the chamfer tool, the diameter that touches the part is 1/8”, but the full tool diameter is ½”. For the calculations, the 1/8” value is used.

Tool	**Tooling Description**	**Surface Feed (ft/min)**	**RPM (rev/min)**	**Chip Load (in/tooth)**	**Linear Feed Rate (in/min)**
T1	3" 4-Flute Carbide Insert Face Mill	1200	4100	0.004	66
T2	3/4" 3-Flute Carbide End Mill	1200	6400	0.003	58
T3	1/2" 4-Flute HSS 45° Chamfer Tool	1200	9167	0.003	110

Figure 2 - Table of Machining Feeds and Speeds

Standard start up lines are used at the beginning of each program to reset the machine and begin with the correct settings. Below is an example of the start up lines used for this operation.

CODE	**EXPLANATION**
%	**“%”** is required at the beginning of each program
O00335;	**OXXXXX** is the standard format for naming a program
T1 M06;	**T__** specifies the tool, **M06** signals the tool changes

G90 G54 G00 X-3. Y-1.25;	**G90** is for absolute coordinate system, **G54** sets the work offsets, **G00** specifies rapid movement, and the **X** and **Y** values set the table location
S4100 M03;	**S__** specifies spindle RPM, **M03** starts the spindle clockwise
G43 H01 Z0.1 M08;	**G43** is tool length compensation, **H01** specifies tool length offset, **Z__** sets the z height offset, and **M08** turns on coolant

Cutting movements are specified next in the code; in this operation we primarily used linear (G1) and circular (G2, G3) interpolation. Linear interpolation requires an X, Y, and/or Z value to travel to and will travel from the previous point to the final point in a straight line. Circular interpolation requires ending X and Y values as well as arc radius. At the start of any movement, a feed rate must be specified. Examples of linear and circular interpolation are shown below:

CODE	EXPLANATION
G1 Z-.16 F60.;	**G1** prepares for linear interpolation to a **Z** height of -.16 relative to the work coordinate system. Since **X** and **Y** values were not specified, their locations do not change. The **F__** value sets the feed rate in inches per minute.
X5.475;	**X** value gives a new value for the linear interpolation movements. Since it has not been overwritten with another command, the linear interpolation (**G1**) and feed rate (**F60.**) are still in effect.
G2 X-.275 Y-2.25 R.5;	**G2** prepares for clockwise circular interpolation, **X** and **Y** values set the final arc position, and the **R** value is the radius of the arc.

The final step in creating the G&M code is simulating it to see the visual tool path. This can be done using several computer programs or websites. **Attachment G** shows the G&M code simulation result. This is an important step in creating the code because it allows for easy trouble shooting and points out errors that could damage or crash a machine. **Attachment H** includes the sample G&M code that was used for this operation.

CNC Setup Sheet Creation

Machine setup preparation is a critical step in the CNC planning process. Proper work holding, feeds and speeds, and operation planning allows for a more repeatable and accurate parts. A CNC Setup Sheet communicates the work holding and set up intentions to the machine operator. **Attachment F** is the Setup sheet for this operation and it lists the equipment needed, tooling details, operation description, and feeds and speeds. Figure 3 is a picture of the stock setup in the machine; the work holding for this operation required parallels and a mill vise. While this is a simple operation, work holding is important to specify to ensure that the cutting forces do not shift the part or remove it from the vise. Additionally, by reviewing the Setup Sheet, the operator understands the tooling and steps necessary for the operation. This ensures that no detail or step is overlooked in the process.

Figure 3 - Setup of stock in mill vise on parallels.

First Article Inspection Report

Inspection is a critical part of the manufacturing process that ensures that manufactured parts meet the required specifications. These geometric dimensions and tolerances are determined by aesthetic, structural, and integration requirements. For this project, our tolerances were given with the original project constraints. The first step in the process was to create a First Article Inspection Report Drawing (**Attachment J**) that defined each of the tolerances and assigned a label to them. The tolerances and inspection outcomes are recorded in the First Article Inspection Document (**Attachment I**).

Figure 4 - Part set on three jack screws to measure flatness of machined surface.


Figure 5 - Dial indicator used for measuring flatness tolerance of top of machined part.

The first tolerance the part was inspected for was the flatness of the top machined part. This was to ensure that the final business card holder would sit well on the desk or other surface it would be used on. Flatness was measured by first elevating the part on three jack screws to create a flat plane, as seen in **Figure 4**. The height of the four corners of the part were measured using a test indicator on a surface plate and leveled to get an accurate flatness measurement. The test indicator was then moved across the surface of the part and displacement of the needle was observed to be no more than .001” across the part, shown by **Figure 5**.

The length, width, and step height were all measured with dial calipers to confirm the lengths and depth. These measurements are important to establish the correct outer dimensions of the final business card holder. The length and width were oversized by .004” which was due to the tool wear on the diameter of the end mill. The step height met the nominal value of .605”.

The corner radii were measured in two ways: a radius gage and optical comparator. The radius gage used a visual check and demonstrated a way to quickly inspect parts on a mass production pass or fail scale. While the radius gage is not a completely accurate method of determining if the radius is the correct size, it saves time and can be used for many parts without requiring additional resources. The optical comparator took more time than the radius gage, but it provided a numerical value for the radius of the corners. To use the optical comparator, the part was aligned, the camera was focused onto the radius and the digital readout was set up to calculate a radius. Three points along the curve were located and entered into the digital readout, and the output was the diameter of the curve. The curve was slightly oversized which was due to the tool wear on the end mill.

The chamfer around the top edge was measured using an optical comparator. The x and y lengths of the chamfer were measured to be .047” in either direction, which is .022” oversized. This was due to the geometry of the spot drill; the flat tip of the spot drill gave an inaccurate “z height” for the assumed tip that was used in the height calculations.

Figure 6 - Drop indicator used to measure part height.

Finally, the overall part height was measured using a drop indicator and surface plate in **Figure 6**. The drop indicator height was set using gage blocks and zeroed at .74”. The displacement from the .74” height was measured to be .003”, which was within tolerance and likely due to tool wear and touch off error, but is within tolerance.

Summary

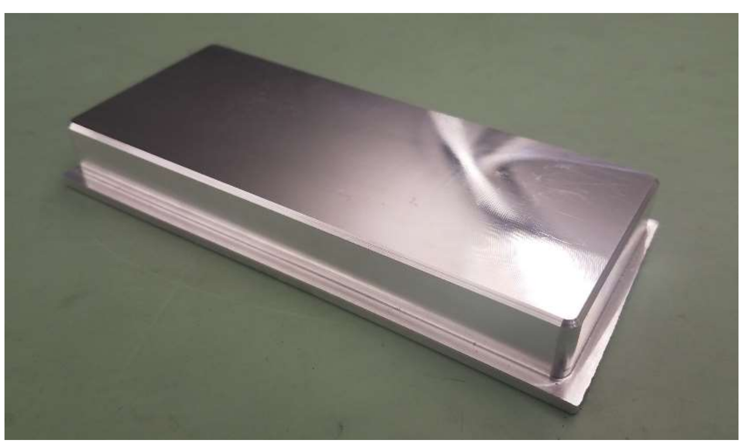


Figure 7 – Part with operation #1 machined

CNC machining is an effective way to repeatably and efficiently produce machined parts. The process of machining the first operation of the business card holder was well documented to reduce errors in the machining process and properly communicate each step. The final product is shown in **Figure 7**.

For this operation, G&M code was manually programmed to better understand how the machine works and processes information. In the future, programs such as MasterCAM or HSMWorks provide a more efficient and dynamic system for generating the G&M code for the machine. This allows for more complex geometry and quicker part production in the future.

Errors in the machining process are revealed during the part inspection operation. At this point, it is important to adjust the setup process or G&M code to correct those errors and prevent production of parts that do not meet specified requirements. During the machining of operation #1, the length and width were oversized; this could be corrected in future operations if the diameter of the tool was inspected prior to machining. The code could be adjusted accordingly to reach the correct geometry. Additionally, the chamfer size was incorrect due to the flat tip of the spot drill. To account for this in the future, it is important not to make assumptions about the tool geometry. This could be accounted for if the tool was observed under an optical comparator and the flat portion of the spot drill could be accounted for. Overall, the errors in the part were due to incorrect assumptions about tool geometry and wear. These could be fixed in the future if tooling was properly inspected prior to manufacturing.

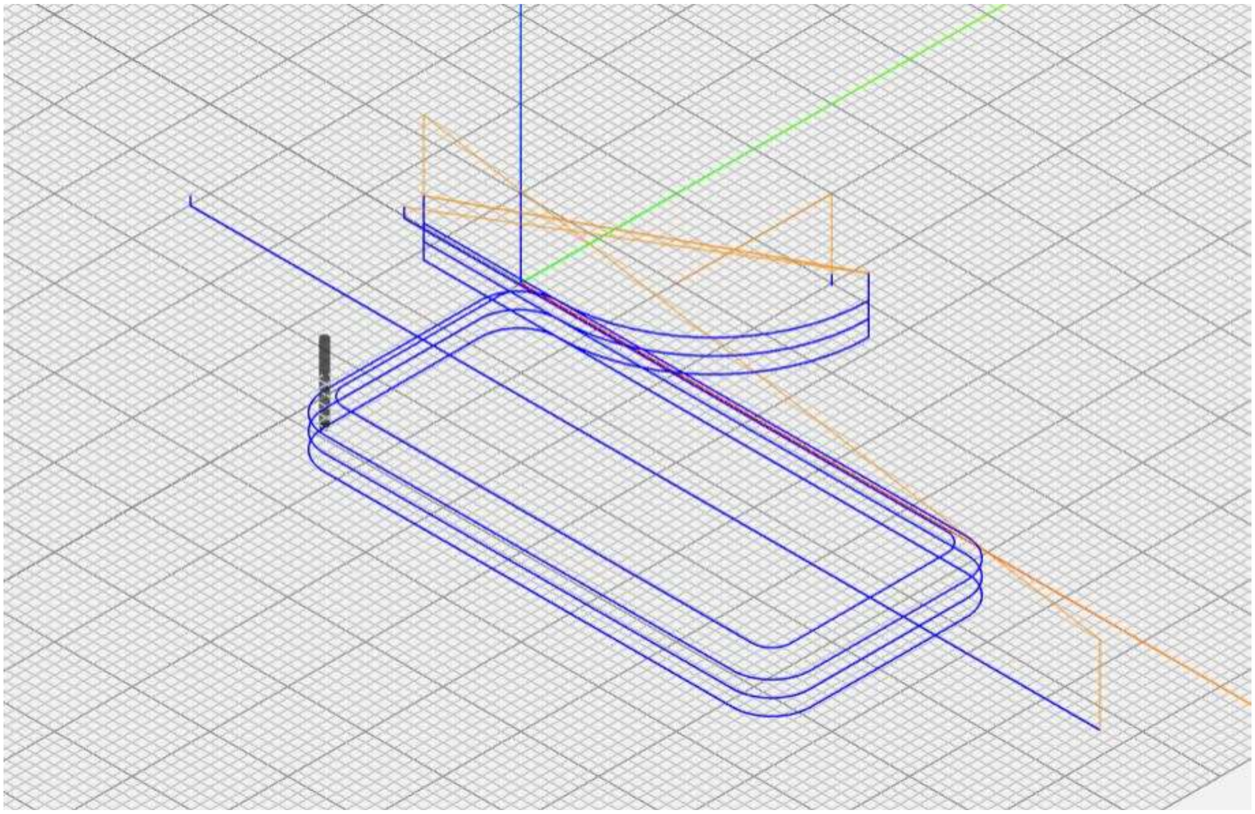
Finally, part and tool offsets were set manually using gage blocks and an edge finder. The machine setup process takes longer than necessary because of this and could be expedited using a probe on the table and in the machine. The probe on the table sets tool offsets and could be used to measure tool diameter and account for tool wear. The probe in the machine accurately locates the part and can set all three axes. Probes provide a more efficient and accurate way to locate the part and tools to eliminate operator error.

In conclusion, manually writing G&M code and setting up the machine produced a well-machined part, but technology provides better manufacturing solutions. Taking advantage of computer aided manufacturing programs for code generation and probes for tool and part location could produce better quality part and a more efficient manufacturing system.

Attachments

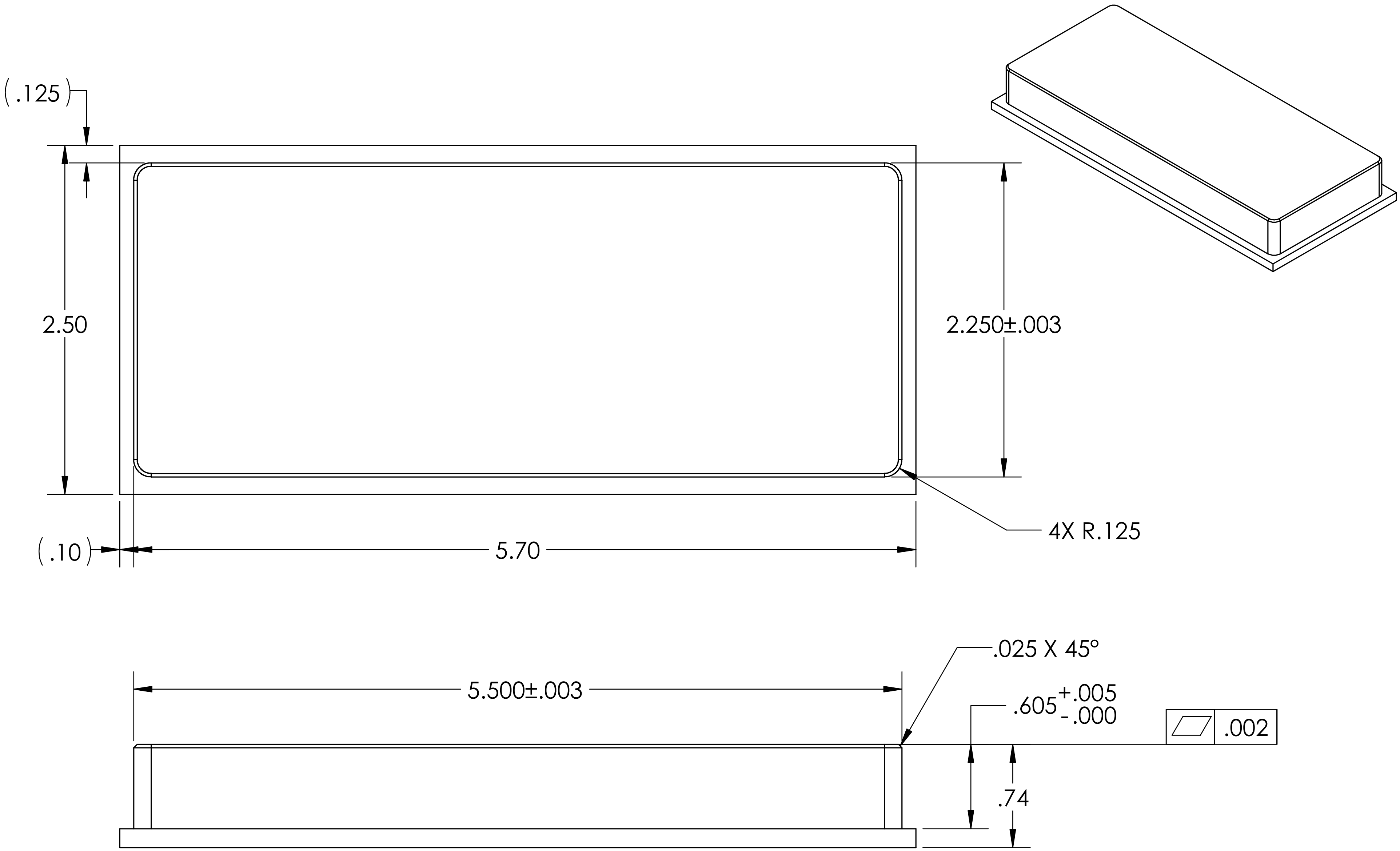
A. Part Drawing
B. Job Planning / Part Routing
C. Tool Path Drawing – 3” Face Mill
D. Tool Path Drawing – ¾” End Mill
E. Tool Path Drawing – ½” Spot Drill
F. Setup Sheet
G. G&M Code Sample
H. G&M Code Simulation
I. First Article Inspection Document
J. First Article Inspection Drawing

Attachment G - Simulated code using online code simulator, NC Viewer.



Attachment H – Sample G&M Code for operation #1

```
%
O00335 (Ribet.nc);
T1 M06;
G90 G54 G00 X-3. Y-1.25;
S4100 M03;
G43 H01 Z0.1 M08;
G1 Z-.01 F66.;
X8.70;
G00 Z1. M09;
T2 M06;
G90 G54 G00 X-1.5 Y.250;
S6400 M03;
G43 H2 Z0.1 M8;
G1 Z-.205 F57.;
X5.475;
G2 X5.975 Y-.250 R.5;
G1 Y-2.250;
G2 X5.475 Y-2.75 R.5;
G1 X.225;
G2 X-.275 Y-2.25 R.5;
G1 Y-.250;
G2 X.225 Y.250 R.5;
G3 X2.225 Y2.250 R2.;
G00 Z.1;
X-1.5 Y.250;
G1 Z-.410 F57.;
X5.475;
G2 X5.975 Y-.250 R.5;
G1 Y-2.250;
G2 X5.475 Y-2.75 R.5;
G1 X.225;
G2 X-.275 Y-2.25 R.5;
G1 Y-.250;
G2 X.225 Y.250 R.5;
G3 X2.225 Y2.250 R2.;
G00 Z.1;
X-1.5 Y.250;
G1 Z-.615 F57.;
X5.475;
G2 X5.975 Y-.250 R.5;
G1 Y-2.250;
G2 X5.475 Y-2.75 R.5;
G1 X.225;
G2 X-.275 Y-2.25 R.5;
G1 Y-.250;
G2 X.225 Y.250 R.5;
G3 X2.225 Y2.250 R2.;
G00 Z.1 M09;
T3 M06;
G90 G54 G00 X-1.5 Y0.;
S10000 M03;
G43 H3 Z0.1 M8;
G1 Z-.16 F60.;
X5.475;
G2 X5.725 Y-.25 R.25;
G1 Y-2.25;
G2 X5.475 Y-2.5 R.25;
G1 X.225;
G2 X-.025 Y-2.25 R.25;
G1 Y-.25;
G2 X.225 Y0. R.25;
G3 X2. Y2. R.25;
G00 Z1. M09;
T1 M6;
G28 Y0.;
M30;
%
```

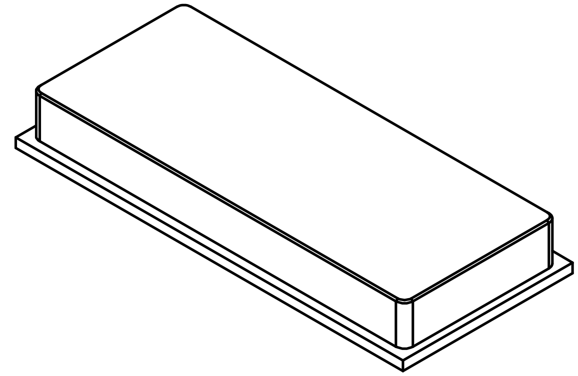


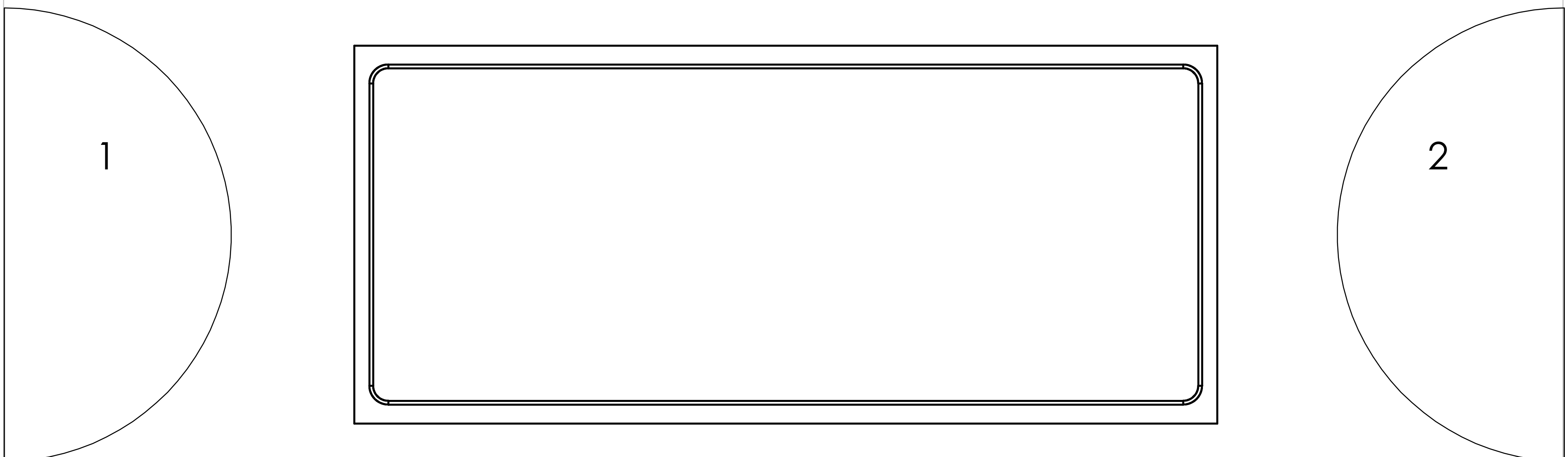
UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN INCHES TOLERANCES: ONE PLACE DECIMAL ± .1 TWO PLACE DECIMAL ± .01 THREE PLACE DECIMAL ± .005	INTERPRET DRAWING PER ASME Y14.5 2009	CAL POLY Manufacturing Engineering DATE: 9/24/2018	MATERIAL: 6061-T6 ALUMINUM DRAWN BY: ROBYN RIBET	TITLE: IME 335 CNC PROJECT #1 SHEET 1 OF 1	SCALE: 1:1	REV B	SIZE A

Attachment B - Job Planning / Part Routing

CAL POLY SAN LUIS OBISPO		PART ROUTING/JOB PLANNER	
		NAME:	Robyn Ribet
		PART:	CNC Business Card Holder
		DRAWING REV:	A
		MATERIAL:	6061-T6
Notes: Deburr all edges after every operation			
OP #	**Operation Description**	**Machine Tool or Cell**	**Tooling and Fixtures Required**
10	Cut .75"x2.50" 6061-T6 Aluminum bar stock to length of 5.70 inches	Horizontal Bandsaw	Dial caliper, file
20	CNC Operation #1	Haas VF-2	Mill vise, 1 5/8" parallels, edge finder, test indicator, 3" face mill, 3/4" End mill, 1/2" 90 degree spot drill, dial caliper
30	First Article Inspection Report Op #1	Metrology Lab	Dial caliper, test indicator and stand, surface plate, 3 jack screws, optical comparitor, radius gage
40	CNC Operation #2	Haas VF-2	Mill vise, soft jaw set, test indicator and holder, 3" 4 flute face mill, 1/2" 4 flute End Mill, 3/4" 3 flute end mill, 1/4" 3 flute end mill, 1/2" 90 degree spot drill, 1/16" 2 flute ball end mill, tool setter, OMP 40 Probe
50	First Article Inspection Report Op #2	Metrology Lab	Coordinate-measuring machine
60	Wash, Polish, Package		Shop Rag, Polish

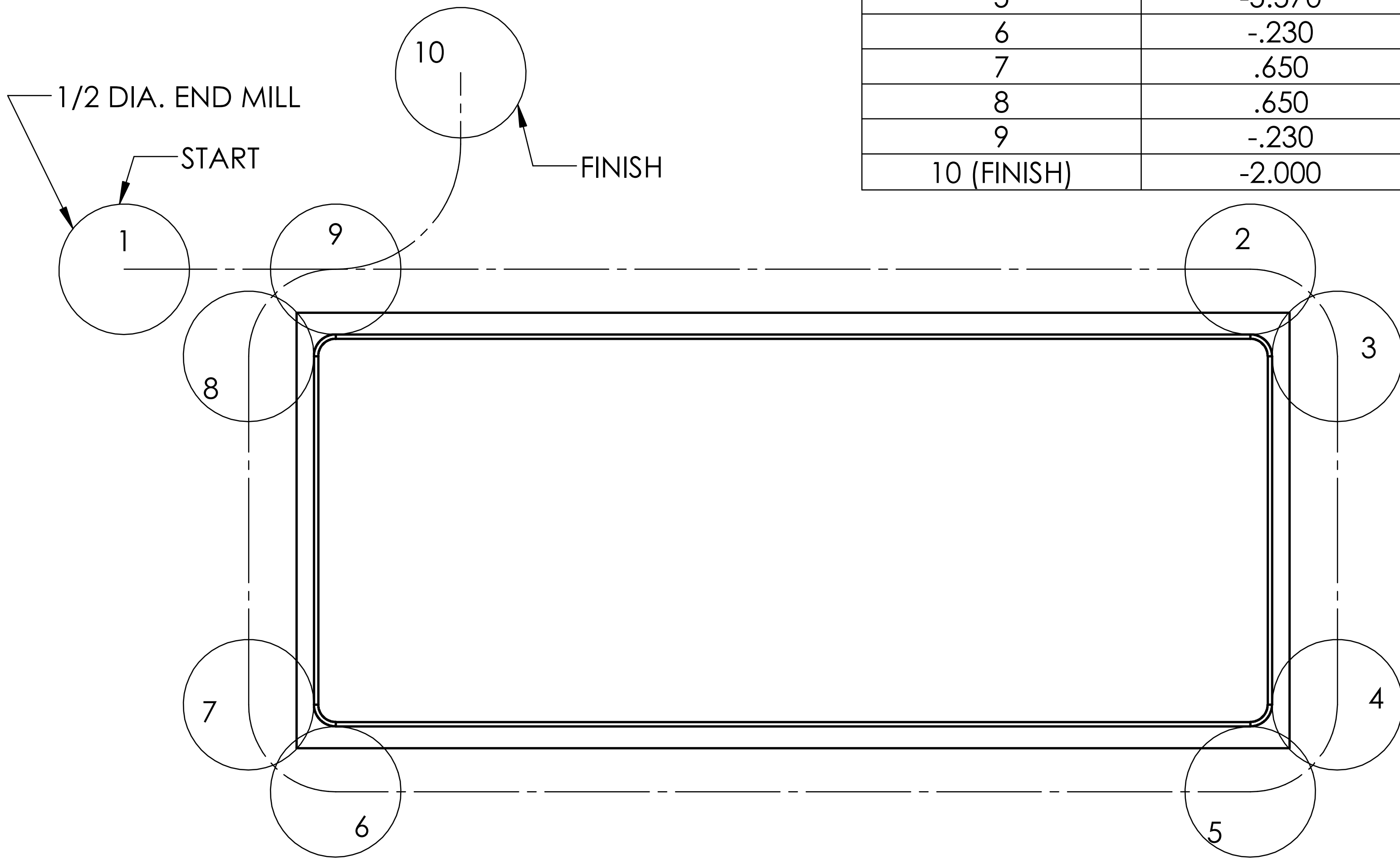
POSITION	X	Y
1	3.00	-1.25
2	8.70	-1.25





UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN INCHES TOLERANCES: ONE PLACE DECIMAL ± .1 TWO PLACE DECIMAL ± .01 THREE PLACE DECIMAL ± .005	INTERPRET DRAWING PER ASME Y14.5 2009	CAL POLY Manufacturing Engineering DATE: 10/5/2018	MATERIAL: 6061-T6 ALUMINUM DRAWN BY: ROBYN RIBET	TITLE: IME 335 CNC PROJECT #1 SHEET 1 OF 2	SCALE: 1	REV B	SIZE A

POSITION	X	Y
1 (START)	1.500	.245
2	-5.570	.245
3	-6.075	-.260
4	-6.075	-1.980
5	-5.570	-2.745
6	-.230	-2.745
7	.650	-1.980
8	.650	-.260
9	-.230	.245
10 (FINISH)	-2.000	2.000



UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN INCHES TOLERANCES: ONE PLACE DECIMAL ± .1 TWO PLACE DECIMAL ± .01 THREE PLACE DECIMAL ± .005	INTERPRET DRAWING PER ASME Y14.5 2009	CAL POLY Manufacturing Engineering DATE: 10/5/2018	MATERIAL: 6061-T6 ALUMINUM DRAWN BY: ROBYN RIBET

TITLE: IME 335 CNC PROJECT #1			
SHEET 2 OF 2	SCALE: 1:1	REV B	SIZE A

Attachment F - CNC Setup Sheet

CAL POLY
SAN LUIS OBISPO

CNC OPERATION SETUP SHEET			
NAME:	Robyn Ribet	**ACHINE TOOL:**	Haas VF-2
PART:	Business Card Holder	**PRINT:**	Business Card Holder Stock
DWG REV:	A	**TY MACHINED:**	1
ATERIAL:	2.50" X 0.75" X 5.70" L.O.C. 6061-T6	**GR APPROVAL:**	

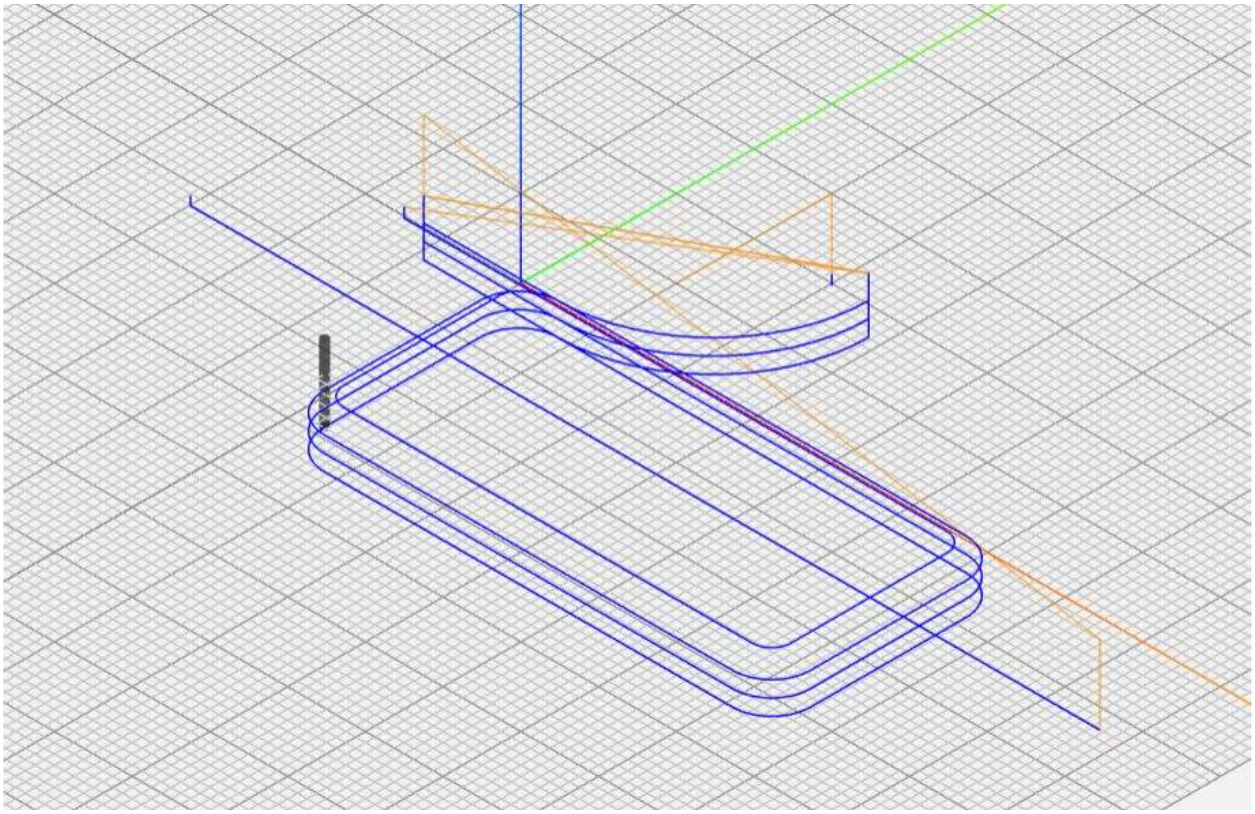
Tooling Information: Mill Vice, Parallels (1 5/8”), Edge Finder, Test Indicator, 3” Face Mill, ¾” End Mill, 45 degree chamfer, dial caliper, 1-2-3 block
Inspection Equipment: Dial Calipers, Surface Plate, Jack Screws, Test Indicator and Stand, Optical Comparator, Radius Gage
Personal Protection Required (PPE): Safety Glasses, Closed Toed Shoes

OP #	Machining Operation Description
20	Begin operation with squaring the mill vice using the test indicator, and a mallet. Next, load the edge finder, 3" Face Mill (T1), 3/4" End Mill (T2), 1/2" Chamfer Tool (T3), and Edge Finder (T4) into the machine and reference them respectively in the G-code. Make sure stock is deburred and place it on 1 5/8" parallels in the vice, tighten the vice to secure the part. Next, use the edge finder to set the work coordinate system as the back left corner of the stock. Next use the 1-2-3 block to touch off each tool and set their heights. Make sure to set the offset from the 1-2-3 block in the work coordinate system. Ensure coolant properly flows from machine before running the program. The Operation 1 program will face the top of the part, profile the sides, and chamfer the edge.

T#	Tooling Description	Surface Speed (ft/min)	RPM (rev/min)	Chip Load (in/tooth)	Linear Feed Rate (in/min)
T1	3" 4-Flute Carbide Insert Face Mill	3220	4099.8	0.004	66
T2	3/4" 3-Flute Carbide End Mill	1200	6400	0.003	58
T3	1/2" 4-Flute HSS 45° Chamfer Tool	1200	9167	0.003	110
T4	Edge Finder	N/A	500	N/A	N/A

Setup Figure

Attachment G - Simulated code using online code simulator, NC Viewer.



Attachment H – Sample G&M Code for operation #1

```
%
O00335 (Ribet.nc);
T1 M06;
G90 G54 G00 X-3. Y-1.25;
S4100 M03;
G43 H01 Z0.1 M08;
G1 Z-.01 F66.;
X8.70;
G00 Z1. M09;
T2 M06;
G90 G54 G00 X-1.5 Y.250;
S6400 M03;
G43 H2 Z0.1 M8;
G1 Z-.205 F57.;
X5.475;
G2 X5.975 Y-.250 R.5;
G1 Y-2.250;
G2 X5.475 Y-2.75 R.5;
G1 X.225;
G2 X-.275 Y-2.25 R.5;
G1 Y-.250;
G2 X.225 Y.250 R.5;
G3 X2.225 Y2.250 R2.;
G00 Z.1;
X-1.5 Y.250;
G1 Z-.410 F57.;
X5.475;
G2 X5.975 Y-.250 R.5;
G1 Y-2.250;
G2 X5.475 Y-2.75 R.5;
G1 X.225;
G2 X-.275 Y-2.25 R.5;
G1 Y-.250;
G2 X.225 Y.250 R.5;
G3 X2.225 Y2.250 R2.;
G00 Z.1;
X-1.5 Y.250;
G1 Z-.615 F57.;
X5.475;
G2 X5.975 Y-.250 R.5;
G1 Y-2.250;
G2 X5.475 Y-2.75 R.5;
G1 X.225;
G2 X-.275 Y-2.25 R.5;
G1 Y-.250;
G2 X.225 Y.250 R.5;
G3 X2.225 Y2.250 R2.;
G00 Z.1 M09;
T3 M06;
G90 G54 G00 X-1.5 Y0.;
S10000 M03;
G43 H3 Z0.1 M8;
G1 Z-.16 F60.;
X5.475;
G2 X5.725 Y-.25 R.25;
G1 Y-2.25;
G2 X5.475 Y-2.5 R.25;
G1 X.225;
G2 X-.025 Y-2.25 R.25;
G1 Y-.25;
G2 X.225 Y0. R.25;
G3 X2. Y2. R.25;
G00 Z1. M09;
T1 M6;
G28 Y0.;
M30;
%
```

Attachment I - First Article Inspection Document

CAL POLY SAN LUIS OBISPO				FIRST ARTICLE INSPECTION SHEET			
				NAME:	Robyn Ribet		
				PART:	Business Card Holder		
				DATE:	10/26/2018		
				DRAWING NUMBER:	IME 335 CNC PROJECT #1 F.A.I.R DRAWING		
Dimension ID	**Description**	**Nominal Size**	**Limits**	**Actual**	**Device**	**Comments**	**Pass/Fail**
Width (A)	Width of the part measured with a dial caliper.	2.250"	± .003"	2.254"	Dial Calipers	The part is slightly oversized, this is possibly due to wear on the end mill over time.	Fail
Length (B)	Length of the part measured with a dial caliper.	5.500"	± .003"	5.504"	Dial Calipers	The part is slightly oversized, this is possibly due to wear on the end mill over time.	Fail
Step Height (C)	Step height was measured using a dial caliper to measure the depth.	.605"	+ .005" - .000"	.605"	Dial Calipers	The height of the step passed and matched the nominal size.	Pass
Flatness (D)	Flatness was measured with a test indicator on a microflat surface. The part was elevated evenly using three jack screws.	N/A	.002"	.001"	Jack screw set, test indicator, surface plate	The flatness varied by one thou over the top surface and fulfilled the form tolerance requirements.	Pass
Chamfer (E)	Chamfer was measured using optical comparitor.	.025" x 45°	± .005"	.047"	Optical Comparitor	The chamfer was significantly too large which was due to the flat tip on the spot drill. This gave an inaccurate z height for the assumed placement of the spot drill tip.	Fail
Corner Radii (F)	Corner Radii were visually checked with a radius gage.	.125"	± .005"	Visual Pass	Radius Gage	The radius gage visually fit around the corners with no obvious gaps.	Pass
Corner Radii (F)	Corner Radii were measured using the optical comparitor to get a numerical radius value.	.125"	± .005"	.129"	Optical Comparitor	The optical comparitor measured three points to create an arc. The arc radius was slightly over-sized due to tool wear.	Pass
Overall Height (G)	Height of the part measured with a drop test indicator on a microflat surface.	.74"	± .01"	.743"	Surface plate, drop indicator	The overall part height was too high by .003". This was likely caused by error in touching off tools and tool wear.	Pass