Jose Amador

ME 340-2 UA Notes

This document will go over all the work that has been done and review the CNC setup that was used.

First, you will get 3-foot-long billets of 4”x4” aluminum. Discuss with Professor Beltran about how many of each blank you will need because it changes from year to year depending on the demand for the class. Switch the horizontal bandsaw to a coarse blade and cut each blank about 1/8” bigger than the final size. Cutting 120 blanks took me about 7 hours in front of the bandsaw.

With all the blanks cut you can now set up the VF2 for machining all the blanks. The programs are setup to maximize efficiency and use two vices and coordinate systems, G54 and G55. The setup is shown below in Figure 1. There are just two programs, one for the core and one for the cavity. Each program uses the same two coordinate systems and the same tools so you can easily switch between them by selecting the corresponding program and using the properly sized blanks.

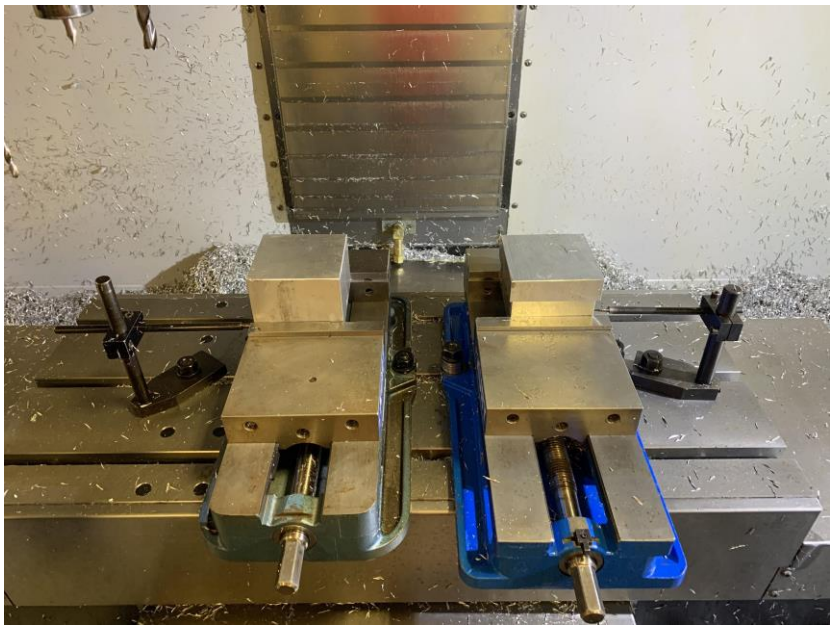

Figure 1: Vice Setup

G54

G54 is the left vice and machines a completely rough blank. When the program runs it will machine half of the blank. This half-completed blank should be moved to the right vice and a new completely rough blank should be placed in the left vice.

The XY zero is set on a corner on the left (double check the programs for specifics) and the block is set up against a stopper that sits below the top of the vice so it does not get machined as shown below in Figure 2.

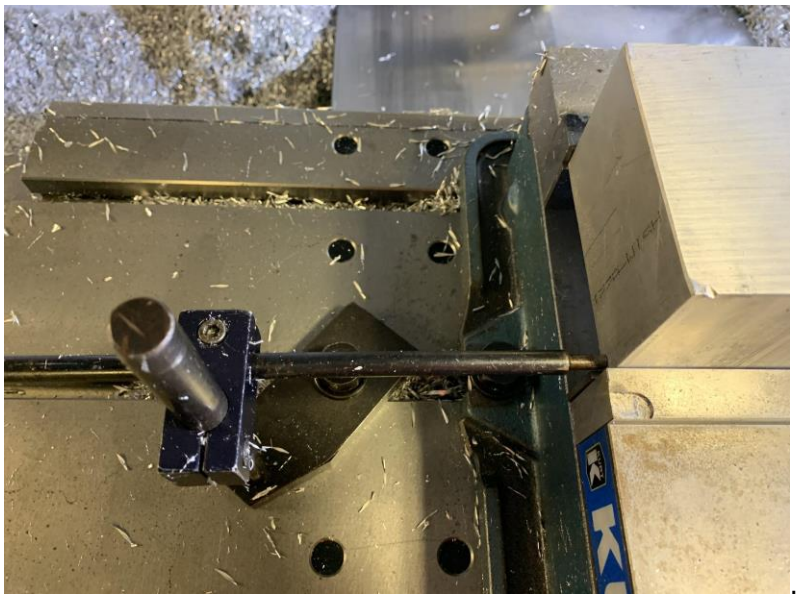

Figure 2: G54 Stopper

G55

G55 is the right vice and machines a blank that is half completed (taken from the left vice). When placing a blank in this vice PAY ATTENTION TO THE ORIENTATION so that way it is machined properly. There is one corner that is rounded and that corner should be in the top left. Furthermore, the precision placement of this block will determine how aligned the two faces of the block are. Adjust the G55 x-coordinate zero to dial this in.

The XY zero is set on a machined surface that you can't directly edge find off of (because the blank is not done yet). You can pick up the y-zero by using the stationary vice jaw. You can pick up the x-zero by setting up the stopper and using some flat piece of metal that is against the stopper (which is exactly where the blank sits).

The stopper once again sits below the top of the vice so it does not get machined, however now it goes up against a precision machined surface as shown below in Figure 3.

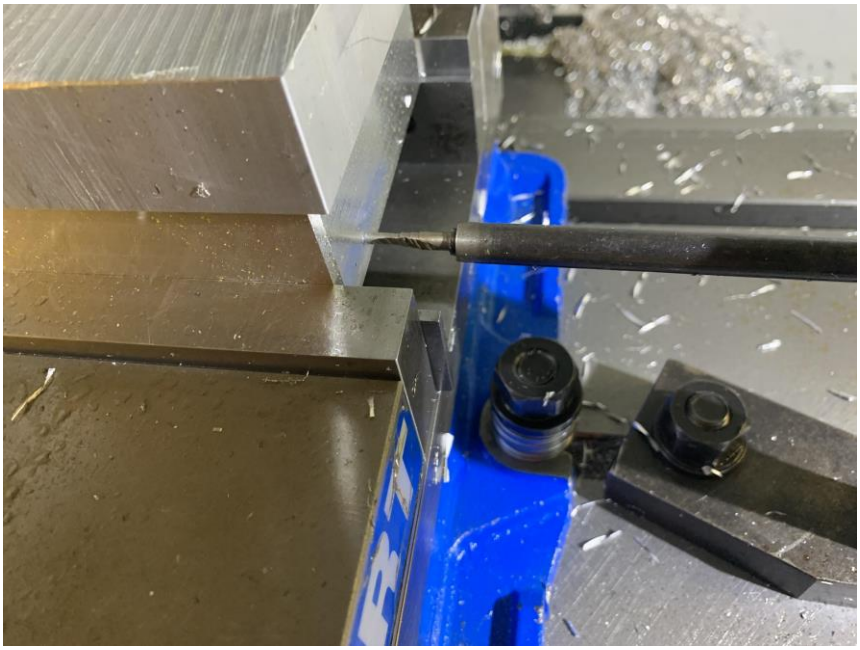

Figure 3: G55 Stopper

Tool Offsets

The way the programs are written, the z-zero plane is on the bottom of the blocks/on the top edge of the parallels. To successfully measure the tool offsets to this plane, you can do a wide variety of things. In my case I measured tool offsets to the bottom of the G54 vice and then manually adjusted them to the bottom of the parallels (which are 1.249” tall). You can also do tool offsets to the bottom of the vice, and use the Z-offset for the coordinate system to accomplish the same task (probably a little easier). Here is a link that explains the difference between tool offsets and positive/negative z-offsets.

Furthermore, the vices might not be the exact same height so I added a 0.0021 z-offset to G55 to compensate for the difference since the top of the parallels in each vice are not level with each other. An image of the tool offsets is shown in Figure 4.

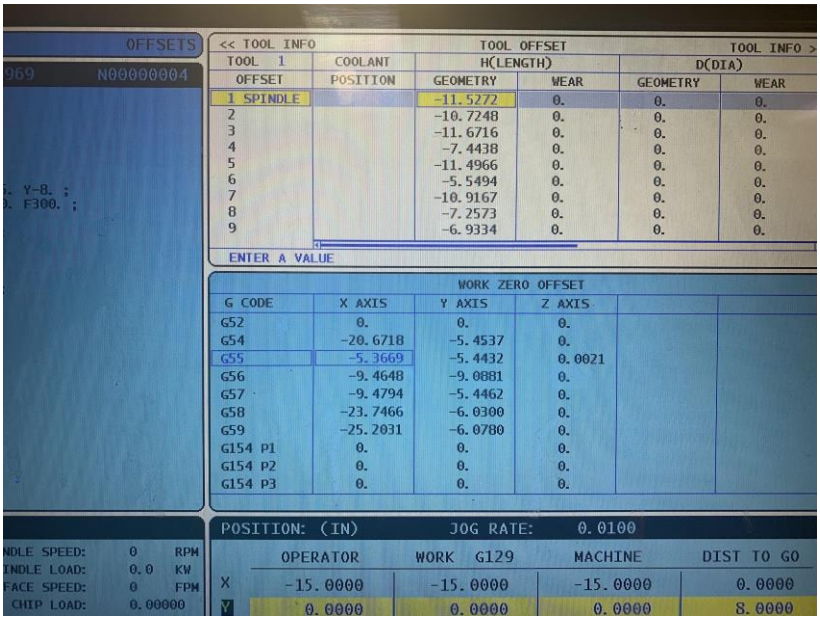

Figure 4: Tool Offset Example

Both blocks take about 15 mins each to run, with the thick block being slightly shorter. Each time you run the program you get a full block (except the first time). Below is a list of tooling and supplies I used. Tools 8-10 are only used for cavity manufacturing. All the tools should be in the blue Lista cart.

Tooling used:
Two vices
Two stoppers (so the blocks end up in the same places)
4 x 1.249” parallels
Tool height setter (get a new one from Professor Beltran because all the old ones are ruined)
Tool #1 - 4.5” Face Mill
Tool #2 - 3 Flute 5/8 Carbide End Mill
Tool #3 - Center Drill
Tool #4 - 9/32 Drill
Tool #5 - 1/4 Ball Mill
Tool #6 - 7/16 Drill
Tool #7 - 7/16 EM (Find or buy a longer one because I had a lot of stick out)
Tool #8 - 1/4 Taper pin reamer
Tool #9 - 1/4 Drill
Tool #10 - 1/4 Flat Carbide EM

What to improve upon for next year and general warnings:

- You need a special post processor to output the program with G54 and G55, check the folder containing all the files.
- Some system to track what teams have which blanks/need more of them (as opposed to just getting random emails and texts from people in the class).

- Better stopper system for more repeatability and accurate parts - some of them were noticeably off and it caused some issues when students went to machine their parts
- Change speeds of the drill bits because some of them squeal very loudly
- Make sure you watch the face mill do its first pass. If your blank from the bandsaw is too big then the face mill will not cut it and it will crash. After that it *should* be good to walk away from (assuming you've watched it before) however still be careful.