

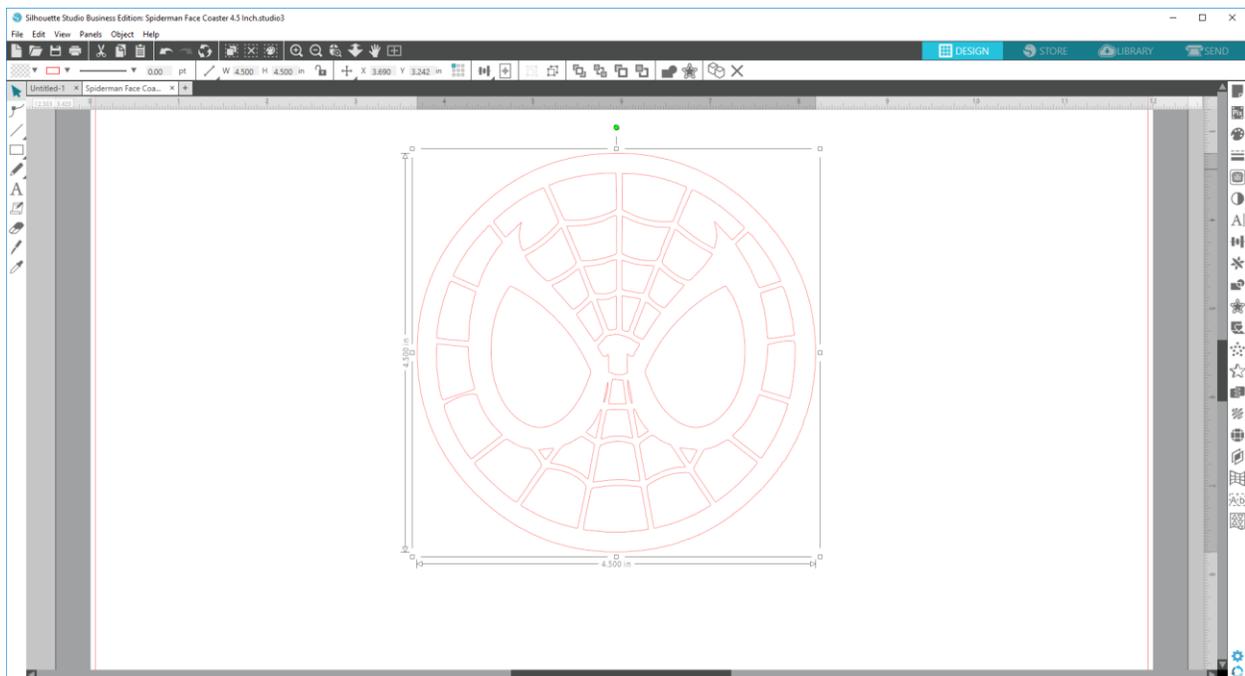
Good morning. Thank you for the positive feedback and sorry for taking a few days to respond, I was away. In terms of work flow, I'm a big fan of Winston Moy and pretty much follow his CAD, CAM, and Cut flow. Please see his video here <https://www.youtube.com/watch?v=t7FMDJciA5U>. His videos are fantastic, and I highly recommend you checking him out if you are a newbie like me.

I am still working with Carbide Create and Motion for now until I get more comfortable but have been slowly learning Fusion 360 and hope to eventually use some of the Vectric products like VCarve Desktop or Pro and move into the realm of 3D.

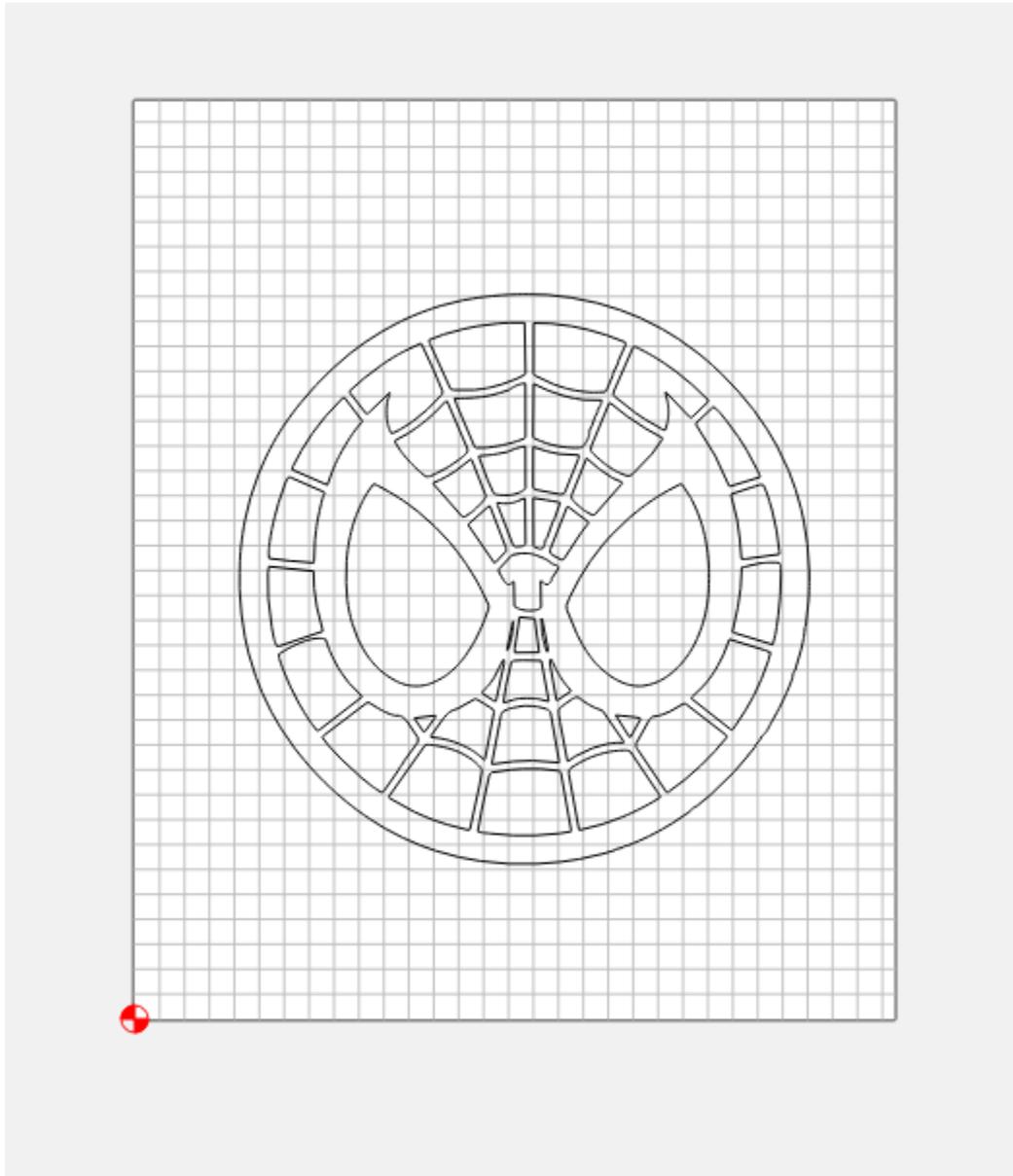
Because I am still relatively new, I have been using the sock "Speeds and Feeds" from within Carbide Create. I'm sure over time I will find settings that will work better but for now I have been getting good results with the defaults.

What I have been doing lately for my projects is creating an SVG file in another program and then importing it into Carbide Create. In my case, and this is probably a bit unorthodox, I create the SVG file using Silhouette Studio which is the software that came with my small desktop vinyl cutter. It's the software I am familiar with but I'm sure you could use a free program like GIMP or Inkscape just as easily to create a 2D image.

Prior to creating the image in Silhouette Studio, I figure out what my dimensions are going to be and scale it accordingly in the software. The intent for the coaster design was to have a diameter larger than most standard coasters so it would accommodate my larger coffee mugs, so I opted for a 4.5" size. I then used the software, (Silhouette Studio, see pic below) to trace a Spidey image from the web that I liked and centered it in the circle I had previously made. The next step is to group the elements and save it as an SVG.



I then open Carbide Create and import the SVG file. Once the SVG file is in Carbide Create, its relatively easy to set up the job and toolpaths, see screenshot below. The one thing I find is key is to make sure you spend the time doing your setup and really thinking about the parameters of the job. I used my digital calipers to take a reading at multiple locations on the piece of aluminum stock that I was using to determine what the maximum thickness was.



So, my stock size, thickness and everything else was entered into Carbide Create. The 10 mm retract height is something that I forgot to change and is far higher than is necessary on a job like this.

The image shows a settings dialog box in Carbide Create, organized into several sections:

- Stock Size:** Includes a diagram of a rectangle with width 'x' and height 'y'. The **Width (X)** is set to 6.000 in and the **Height (Y)** is set to 7.250 in.
- Stock Thickness:** Includes a diagram of a rectangle with thickness 'z'. The **(Z)** is set to 0.130 in. A dropdown menu is set to **Top**.
- Toolpath Zero:** Includes a diagram of a rectangle with a red dot at the bottom-left corner. A dropdown menu is set to **Lower-left**.
- Material:** A dropdown menu is set to **Aluminum - Metal**.
- Machine:** A dropdown menu is set to **Shapeoko XXL**. The **Retract Height** is set to 10.000 mm.
- Units:** Radio buttons for **Inches** (selected) and **MM**.
- Document Background:** A green button labeled **Edit**.
- Clear Drawing:** A green button labeled **Clear Drawing**.
- Ok** and **Cancel** buttons at the bottom.

There are only two tool paths in this job and below are the parameters for Tool Path 1 or the face carving portion of the job. To keep things simple, I have purchased all of my bits from Carbide 3D initially because they are all built into the software and seem to be good quality, but I am definitely not an endmill expert at this point. I used a #112 0.063in Square endmill because it was the largest bit that I had that had the ability to reach the majority of the areas to be milled. However, you will notice that the tool path was not able to reach all of the areas as indicated by the orange slivers that do not have blue tool paths in them. The squares in the images below are just my attempt to make sure I was going to provide myself with spacing I required on my particular piece of aluminum stock.

Tool

#112 0.063in

[Edit](#)

Depth Per Pass 0.010 in

Stepover 0.028 in

Feedrate 14.300 in

Plungerate 1.787 in

RPM 9375.000

Cutting Depth

Start Depth in
[Use Stock Top](#)



Max Depth in
[Use Stock Bottom](#)

Offset Direction

Pocket

Outside / Right

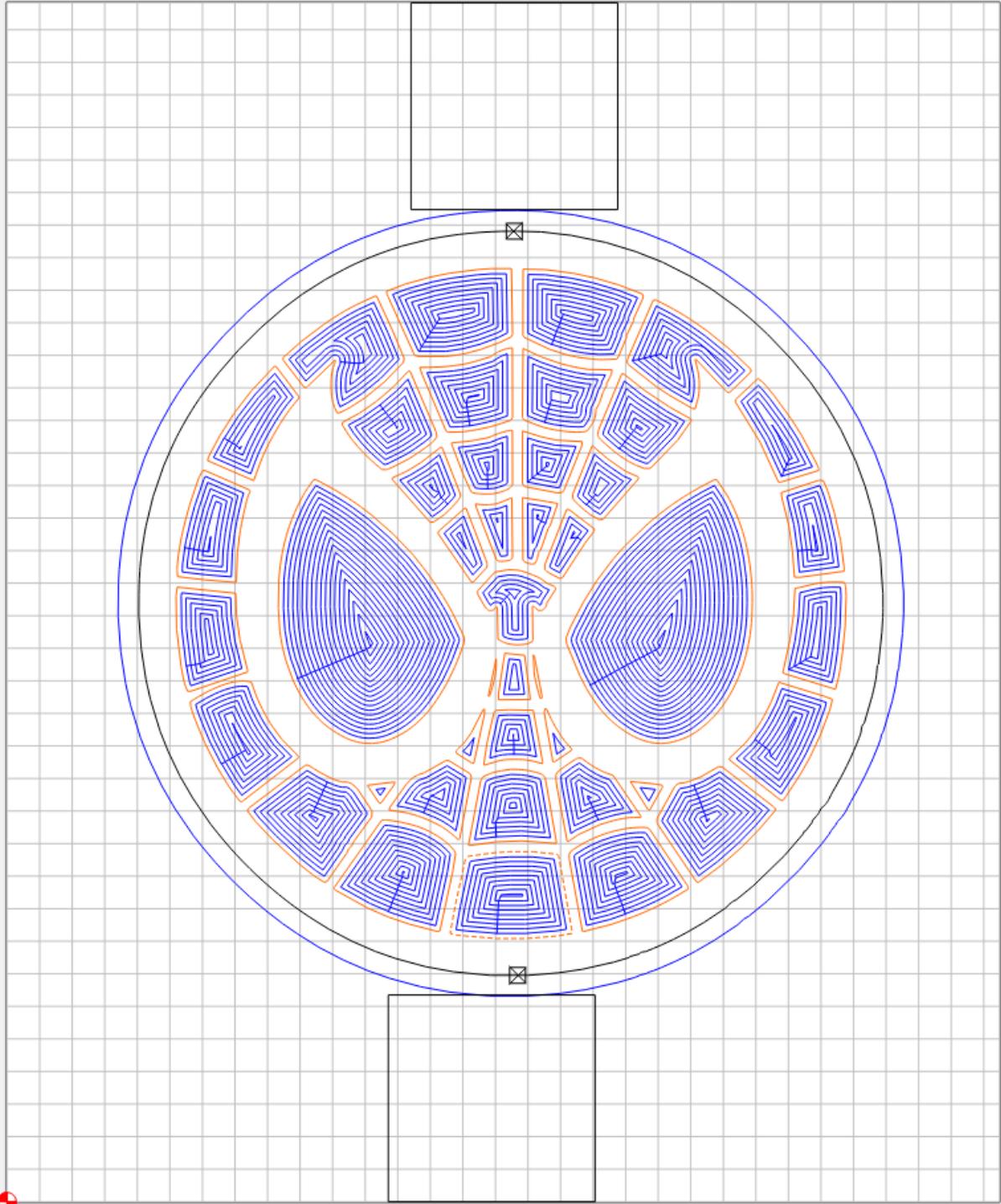
Inside / Left

No Offset



Name

[Ok](#)



Tool path 2 just consists of a few tabs and is the outer circle that cuts the job free from the stock. For this, I was able to use my #201 0.250in bit. This portion of the job was completed fairly quickly as opposed to the face carving portion. See Tool path 2 below.

Tool

#201 0.250in

[Edit](#)

Depth Per Pass 0.010 in

Stepover 0.113 in

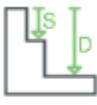
Feedrate 36.669 in

Plungerate 3.056 in

RPM 12223.000

Cutting Depth

Start Depth in
[Use Stock Top](#)



Max Depth in
[Use Stock Bottom](#)

Offset Direction

Pocket

Outside / Right 

Inside / Left

No Offset

Tabs

[Edit tabs](#)

Name

[Ok](#)

Well, that's probably more than anyone ever would need to know about making a coaster, lol but I wanted to provide you with as complete of a reply as possible. We are all constantly learning here, and I

think its important to share knowledge whenever we can. For the record, I have only had my machine for about a month and am not an expert in any way. If anyone has any comments on tips, tricks, etc. that can make what I have done here better, please feel free to share. Thank you.