

# G-Code with XCarve and Fusion 360

How to take a file from Fusion 360, and CNC it using Easel

## Contents

- Initial Part Set-Up (Page 2)
- Drilling Set-Up (Page 5)
- Cutting Set-Up (Page 7)
- Easel - Exporting and Cutting (Page 9)

## Robo Raiders FTC 7129

Titus 2:7-8a

Show yourself in all respects to be a model of good works, and in your teaching, show integrity, dignity, and sound speech that cannot be condemned.

**Have Questions?  
Contact Us!**

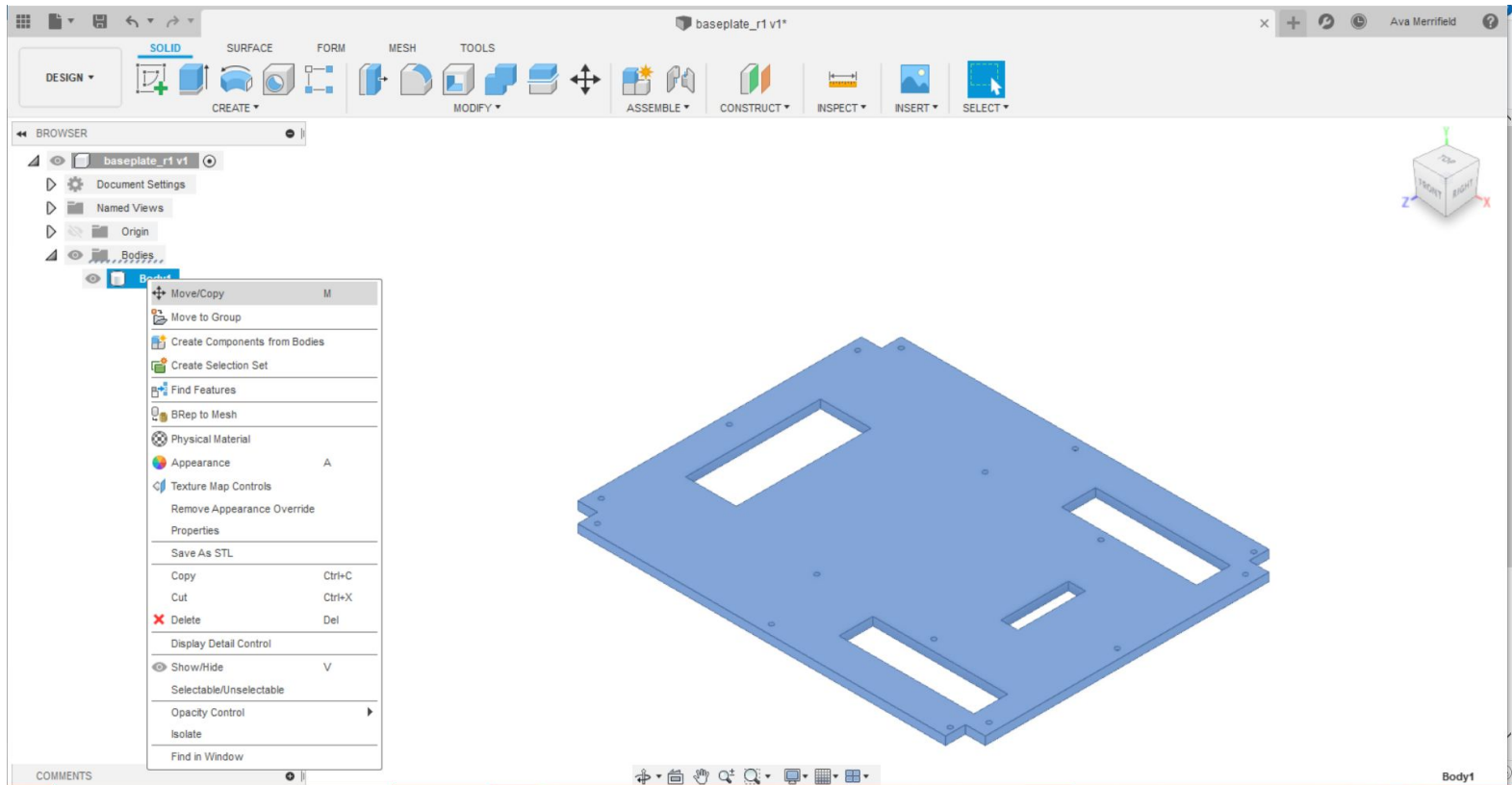
Email: [teamroboraiders@gmail.com](mailto:teamroboraiders@gmail.com)

Instagram: [@ftc7129](https://www.instagram.com/ftc7129)

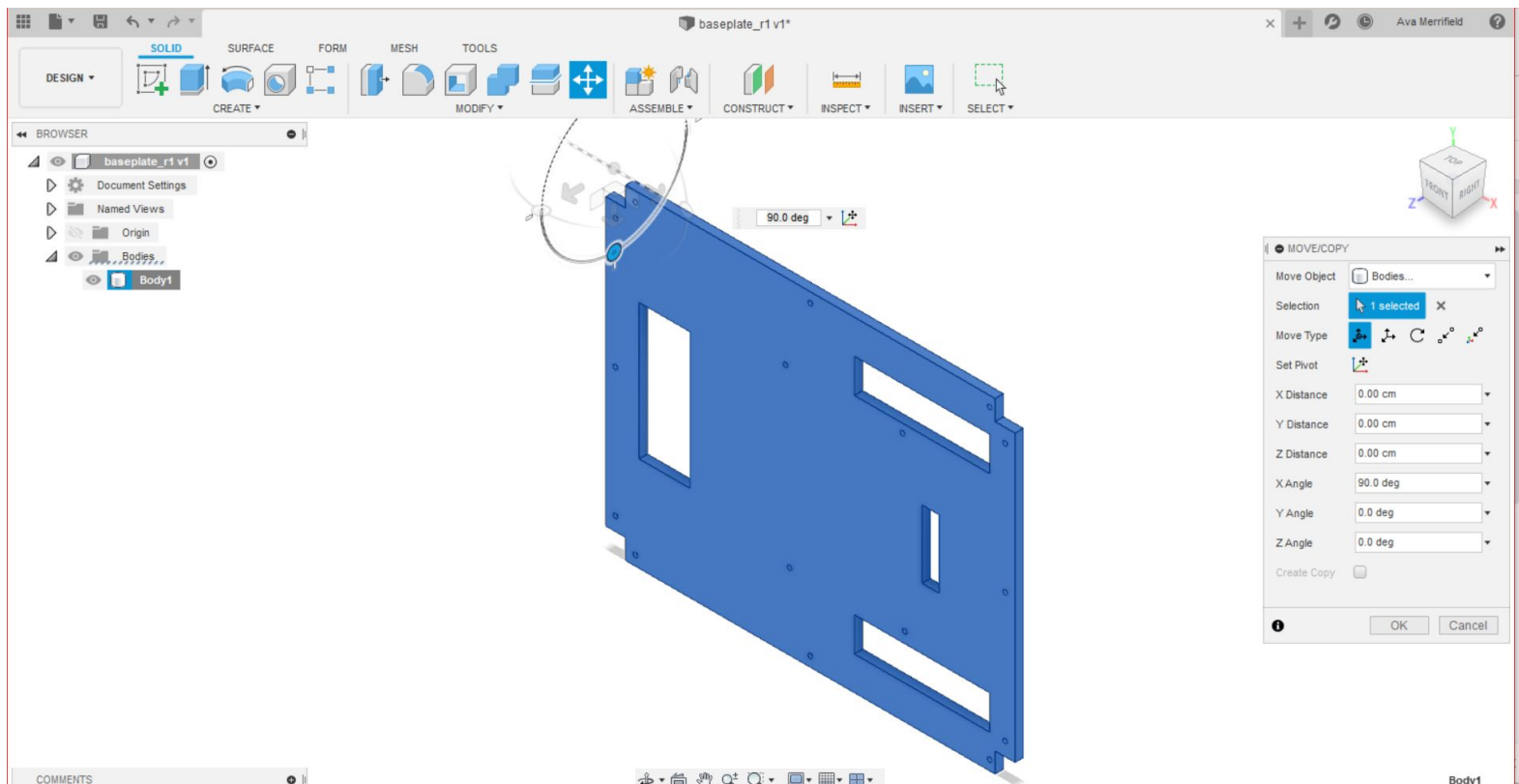
Facebook: Robo Raiders FTC 7129

# Initial Part Set-Up

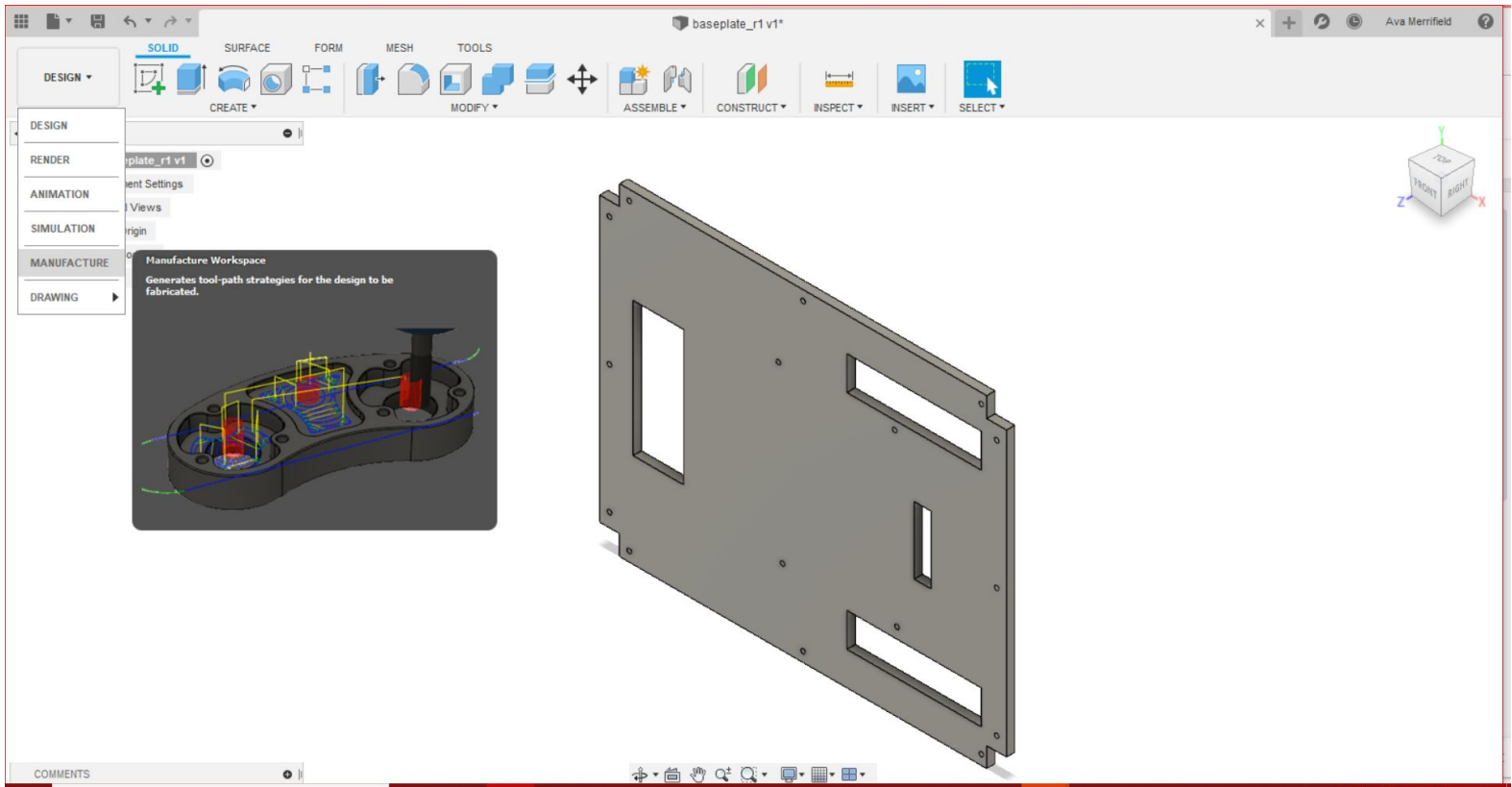
1. If your file is already in your Fusion project, then start there. If not, download your file as an SAT file.
2. Open your file in Fusion and click on “Body”. Then, right-click on the body you want to cut. Select “Move/Copy”.



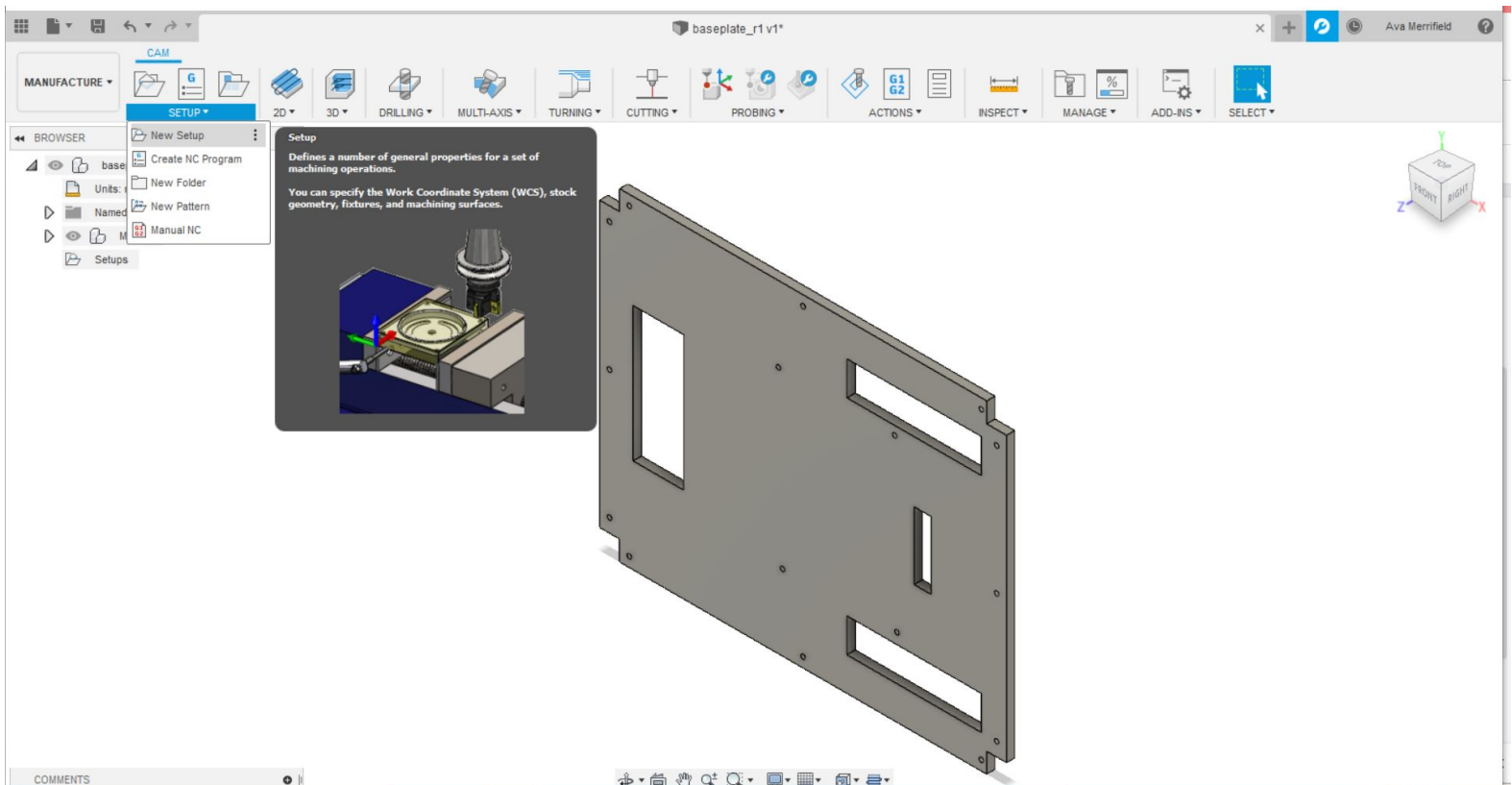
3. Grab the front arrow and rotate the part so that it is down 90 degrees. The top face should be perpendicular to your base grid. Select “Okay”.



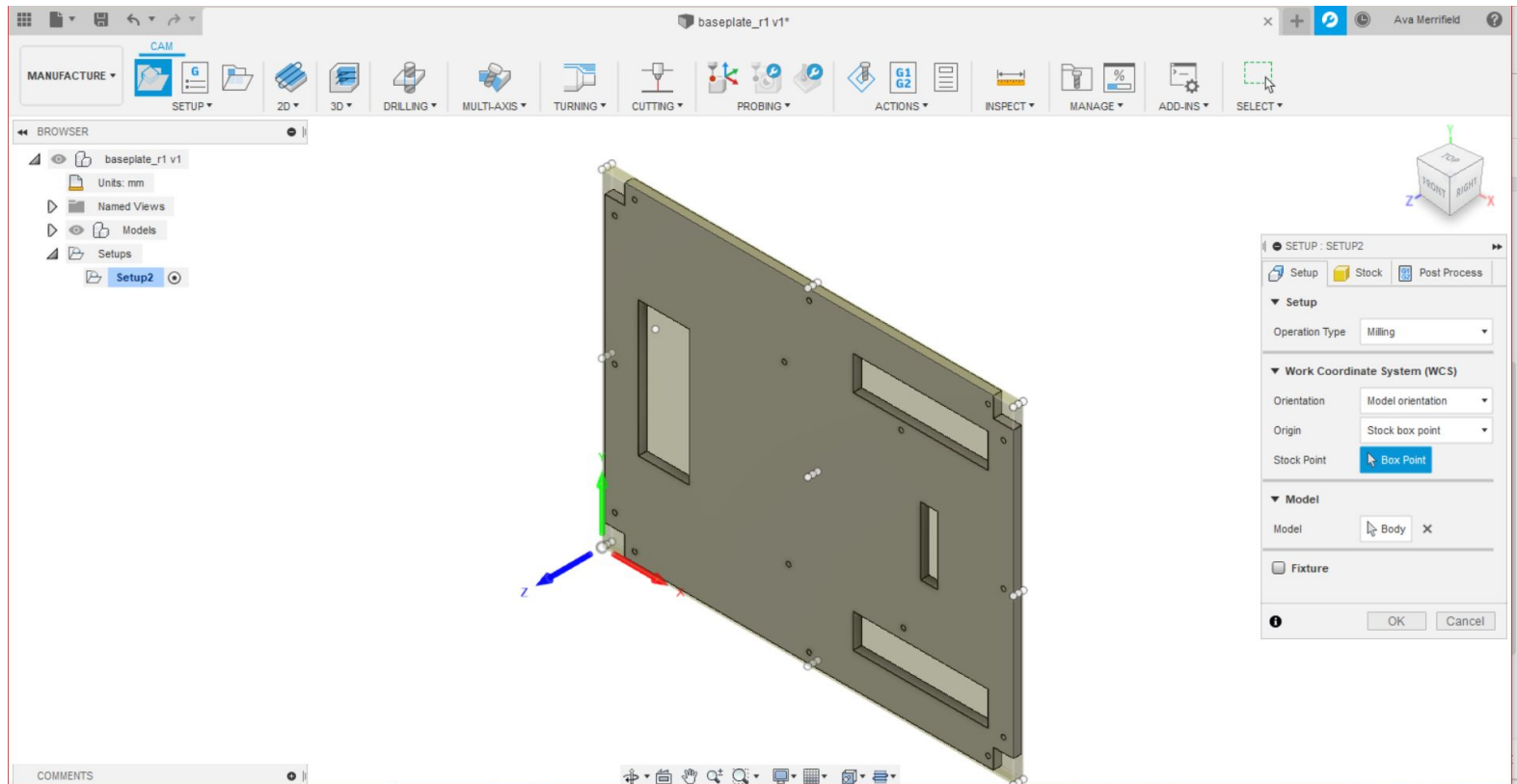
4. Go to the top left of your screen where it says “Design” and select “Manufacture”.



5. Go to the top left of your screen where it says “Setup” and select “New Setup”.

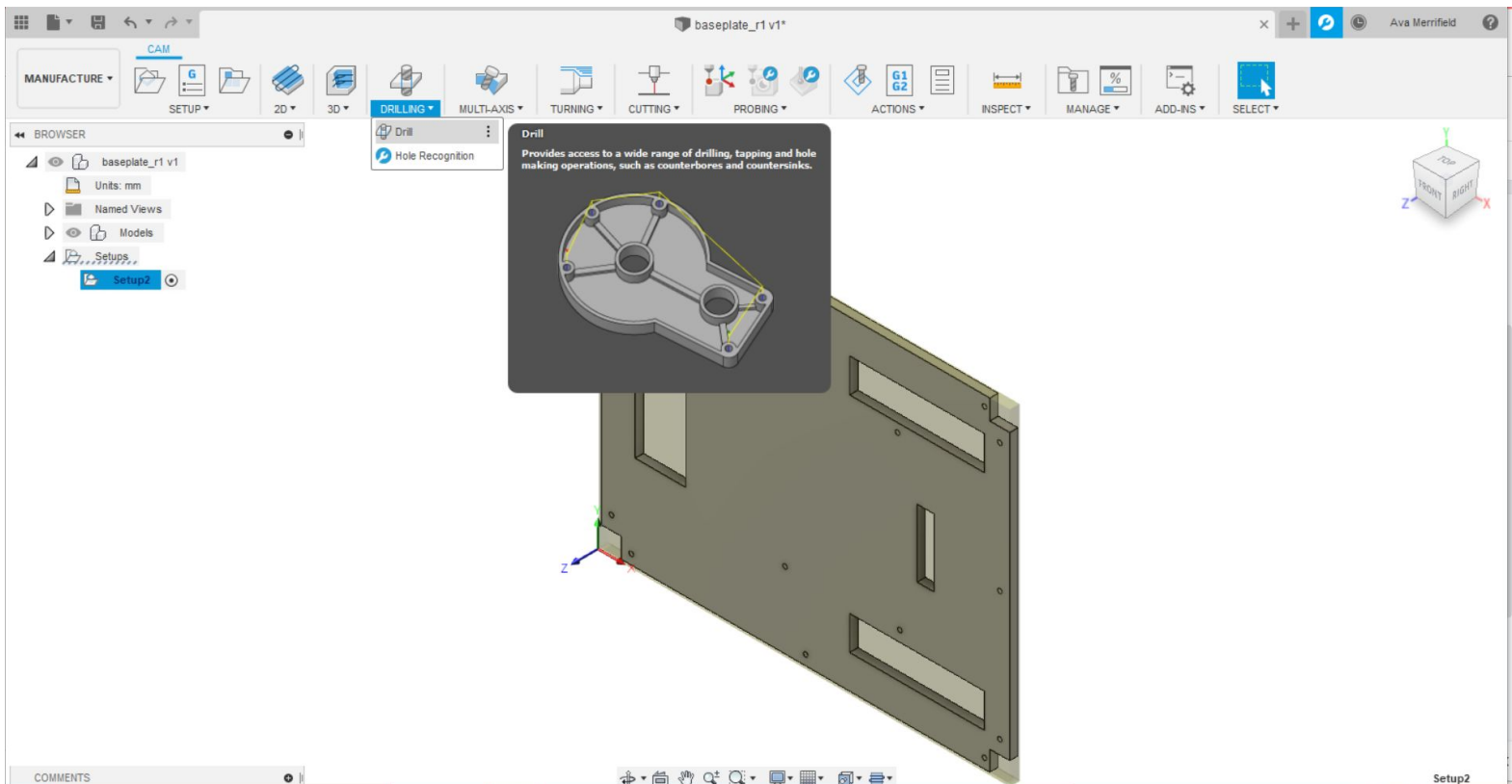


6. Click on the bottom left point of your part. That will move the bit origin to there, rather than having it in the middle.

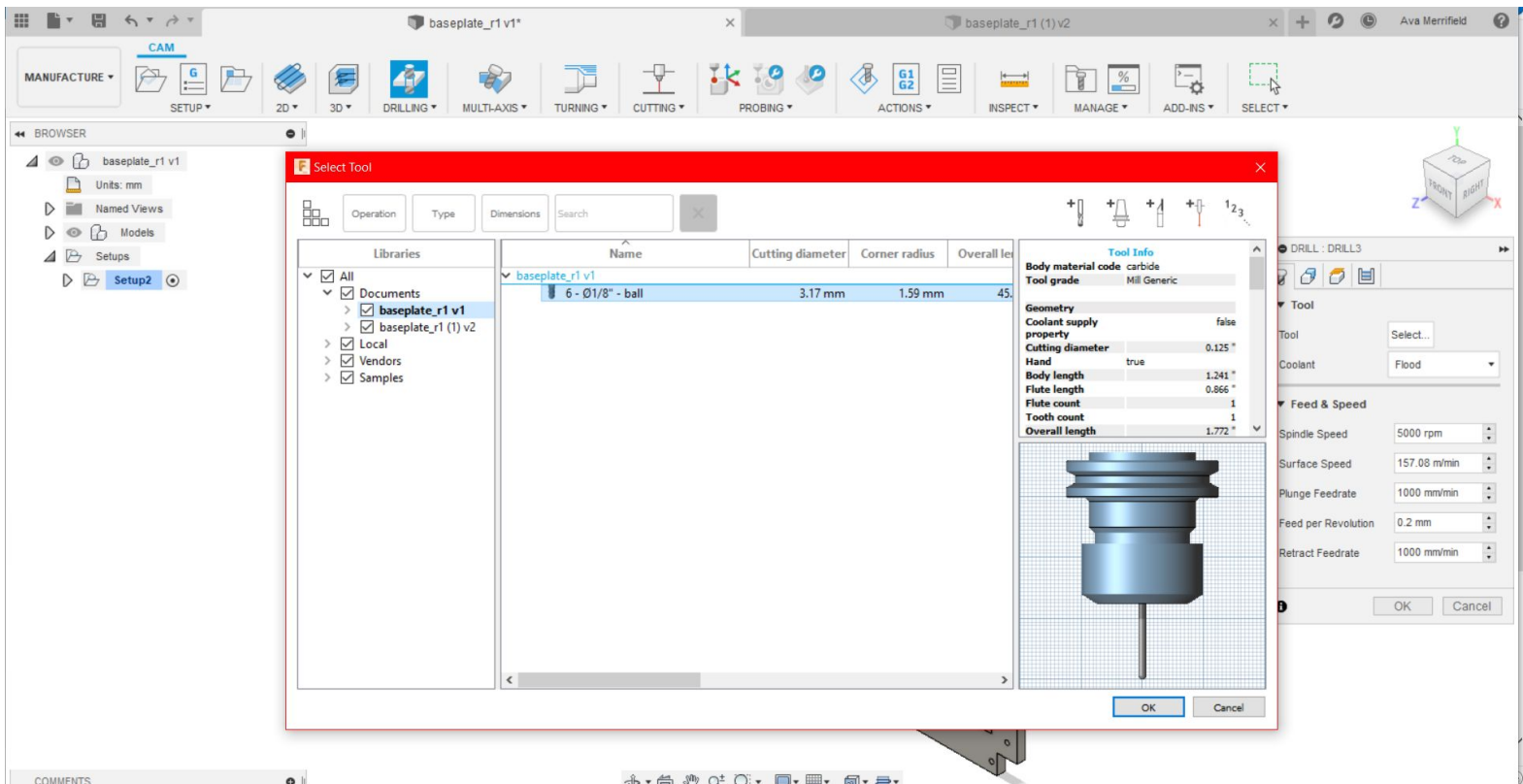


# Drilling Set-Up

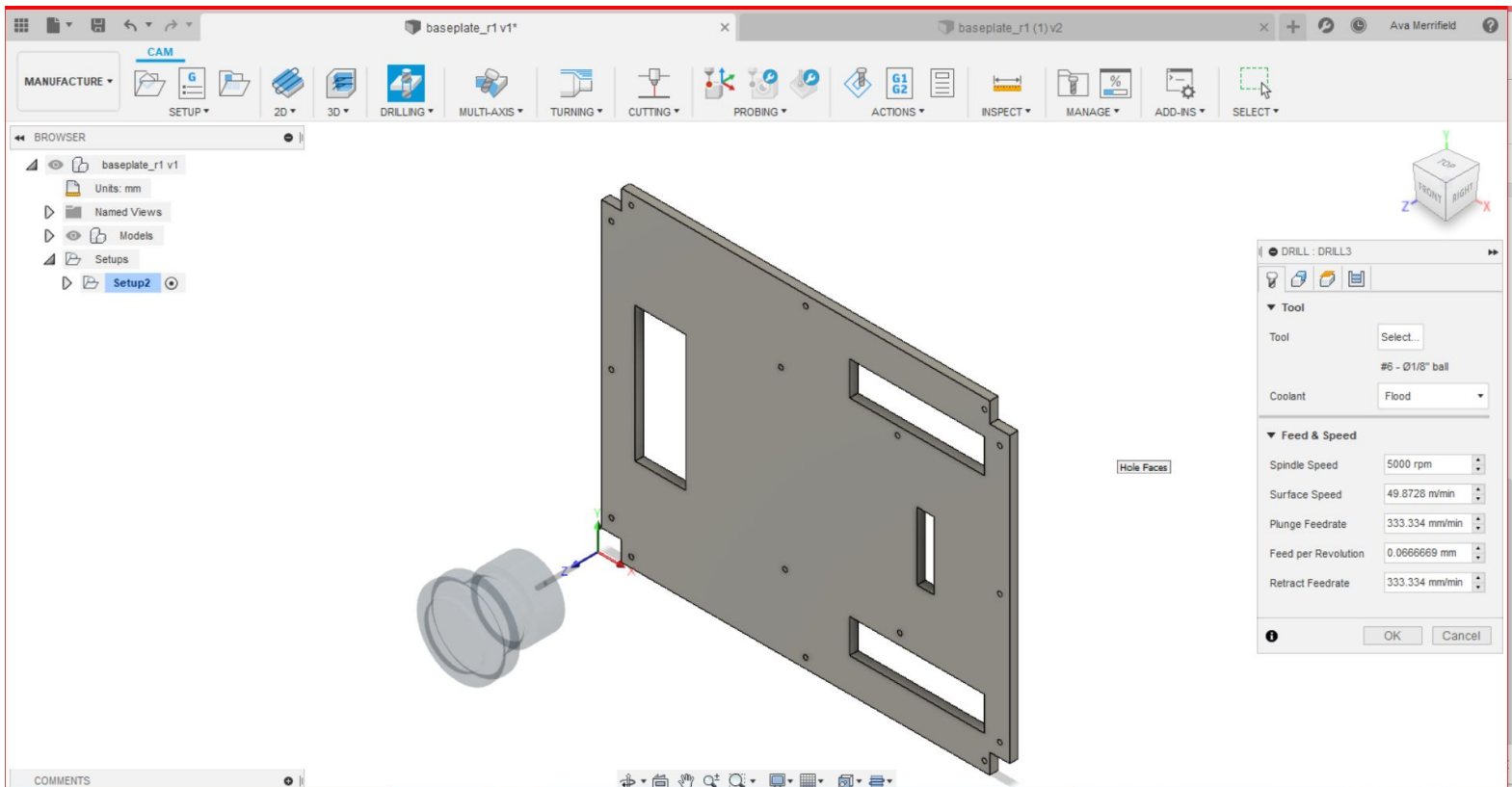
1. Go to the options bar where it says “Drilling” and click on that.



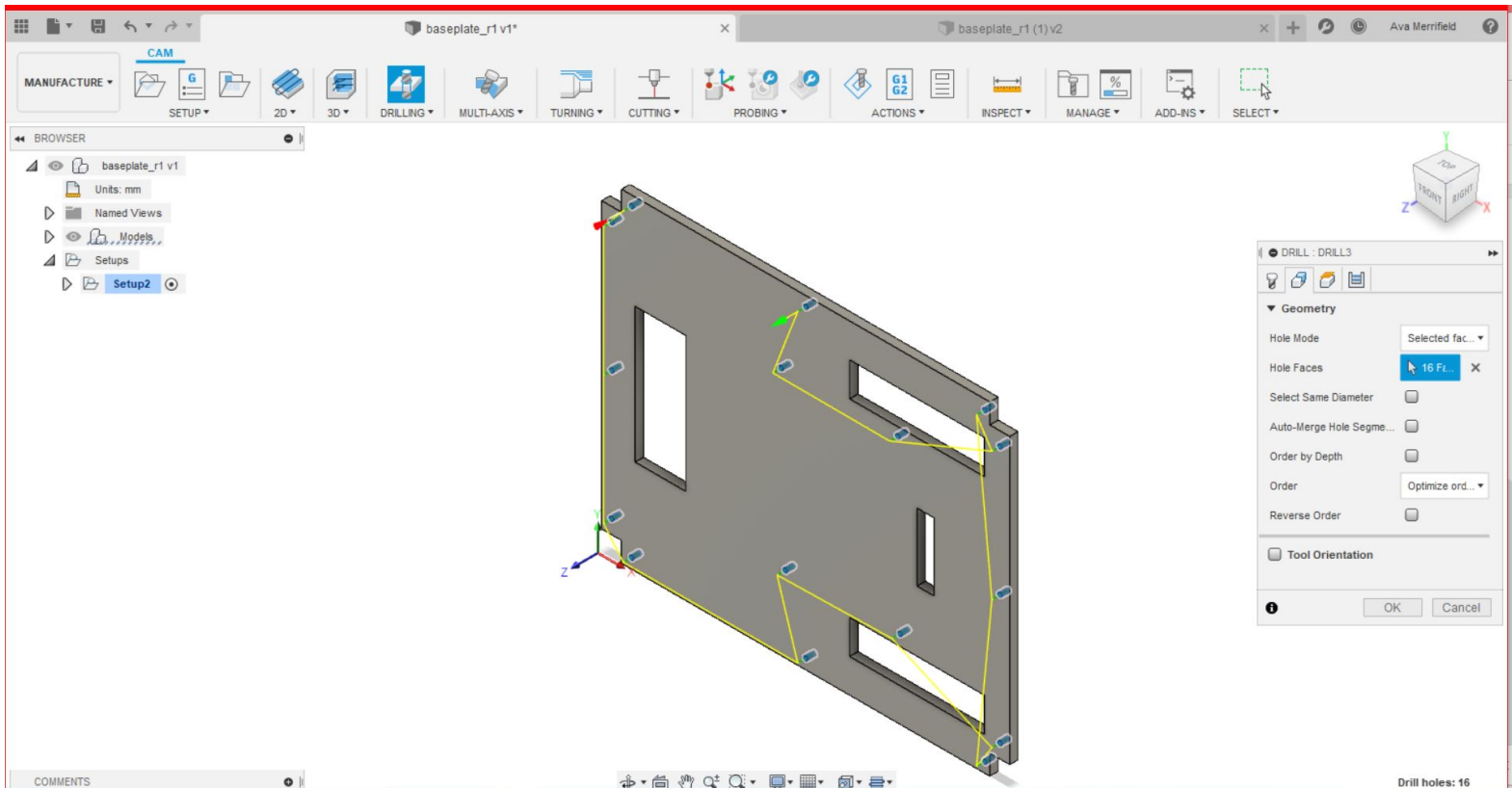
2. Go to your drilling menu on the right side of your screen, and press “Select” next to where it says “Select Tool...”. Select the tool you want to use (My team usually selects the tool from another part we previously CNC'd).



3. Select “Okay.” Your tool should be in the bottom left of your part.



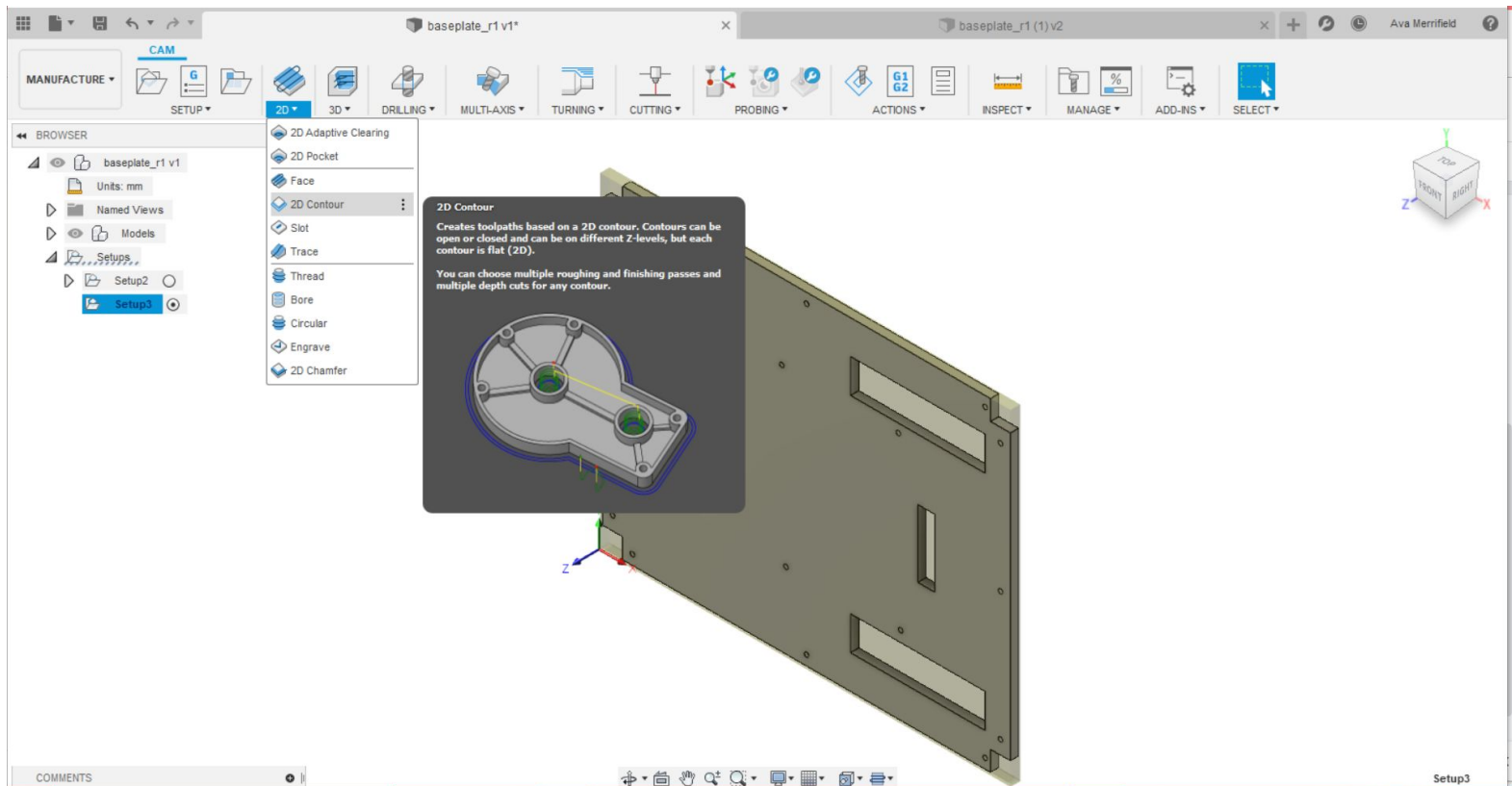
4. Click all of the holes on your part. A toolpath should auto-generate. Once you’ve selected all the holes, click “Okay” in the drilling menu.



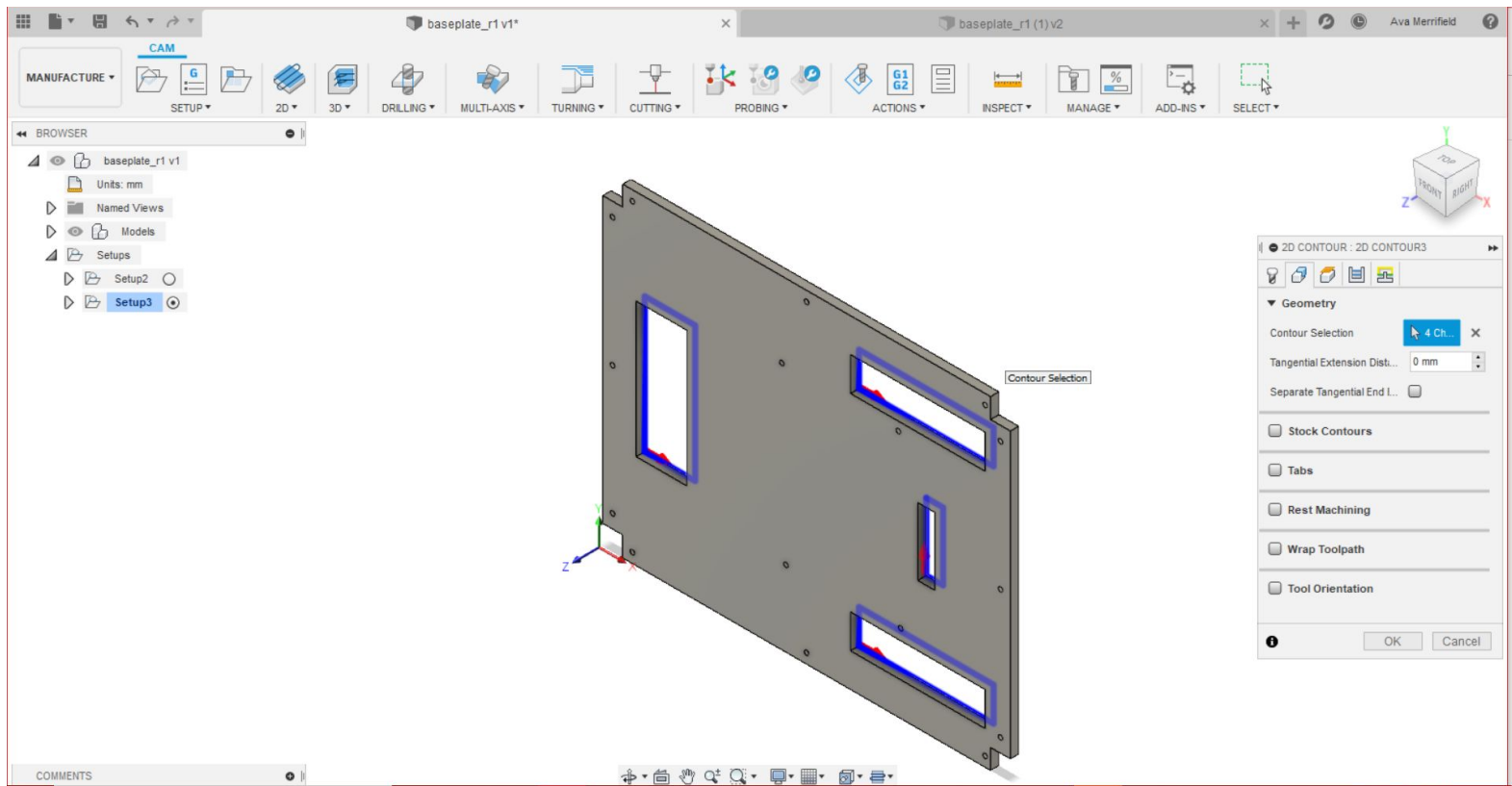


# Cutting Set-Up

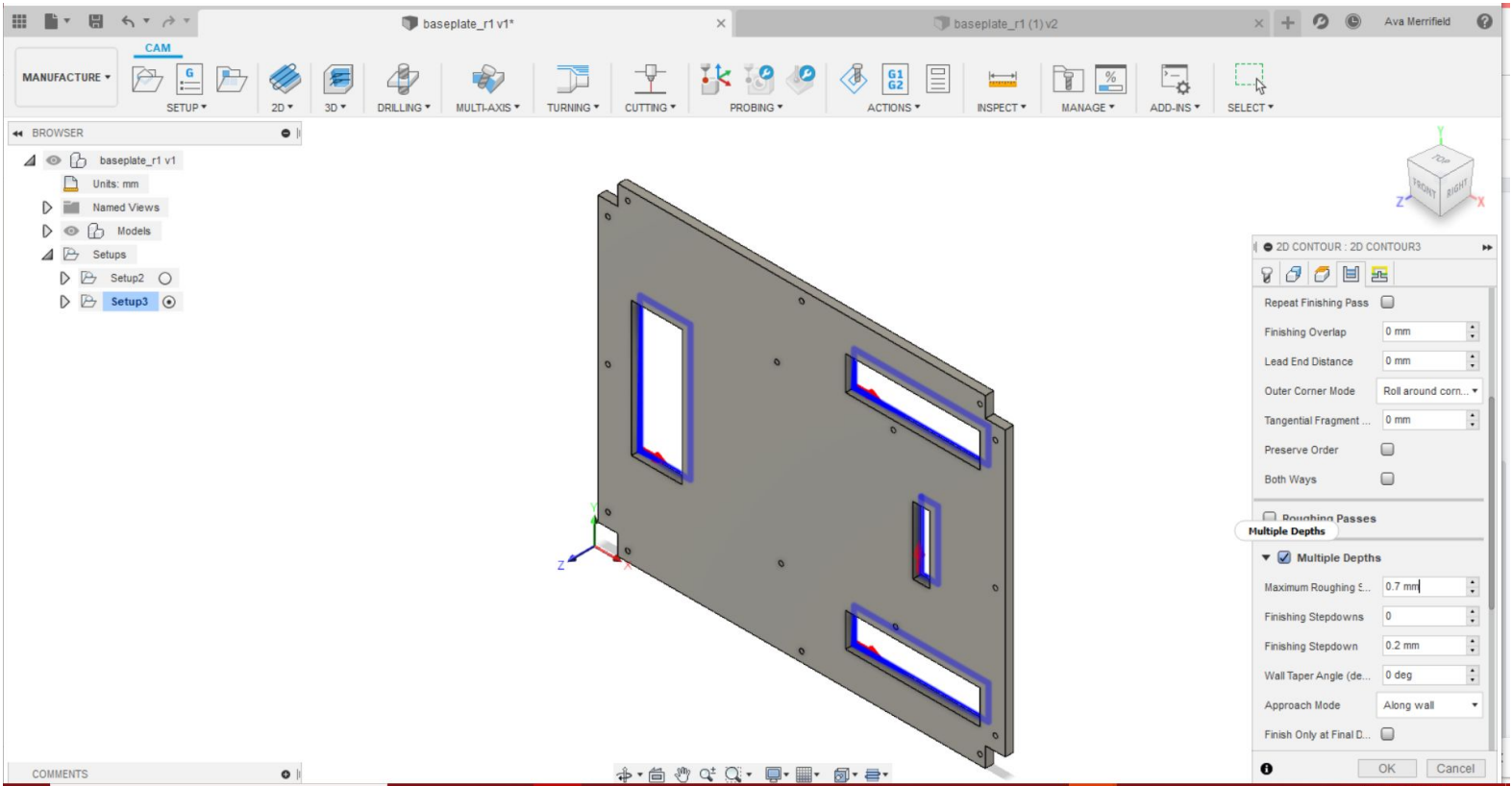
1. Go to the top right of your screen where it says “Contour” and select “2D Contour”



2. Click on the bottom edges of where you want to cut.



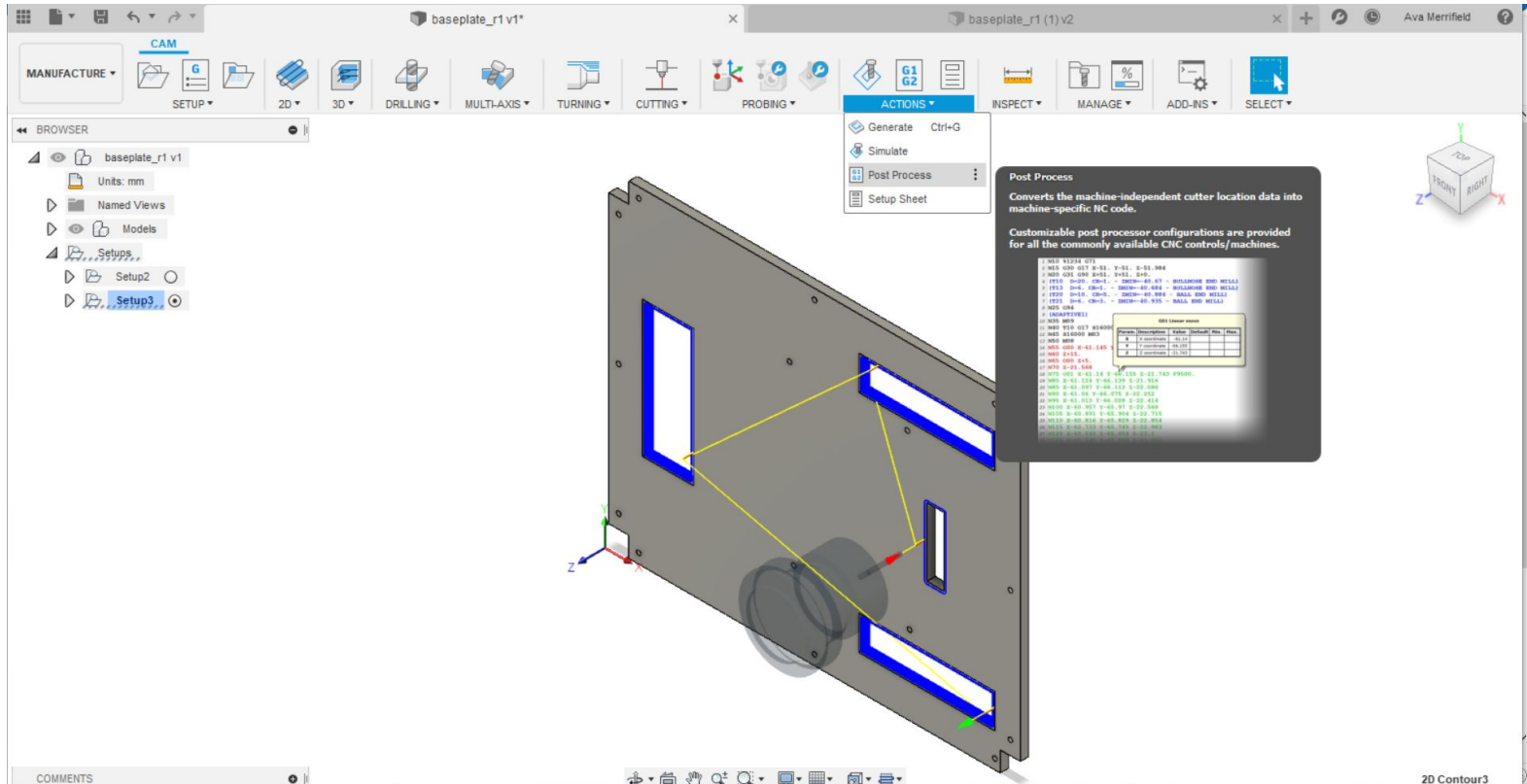
2. In the contour menu, go down to where it says “Multiple Depths” and select it. This means the CNC will make multiple passes as it cuts, rather than try and cut all the way through the material the first time. Select “Okay”.



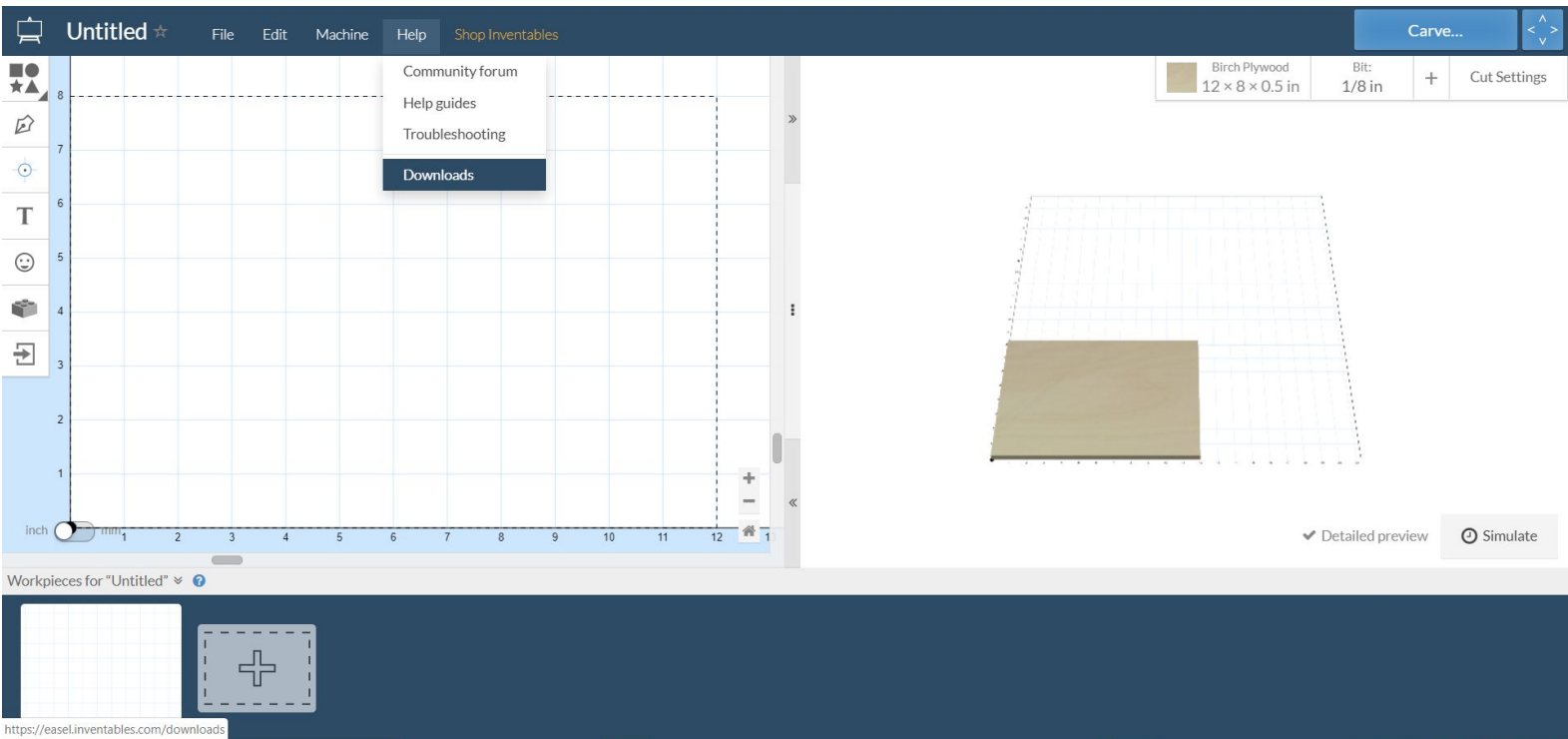


# Easel - Exporting and Cutting

1. In the menu bar, click on “Actions” and select “Post Process”



2. Log into Easel. Once logged in, go to “Help” and select “Downloads”. This will take you to the page where you can download the post processor.



3. Download the Fusion 360 post processor.

Downloads

Easel Driver

Easel Driver is a small software that communicates between Easel and X-Carve or Carvey.

Download for Windows

Version 0.3.14

Other Downloads

Return to Easel

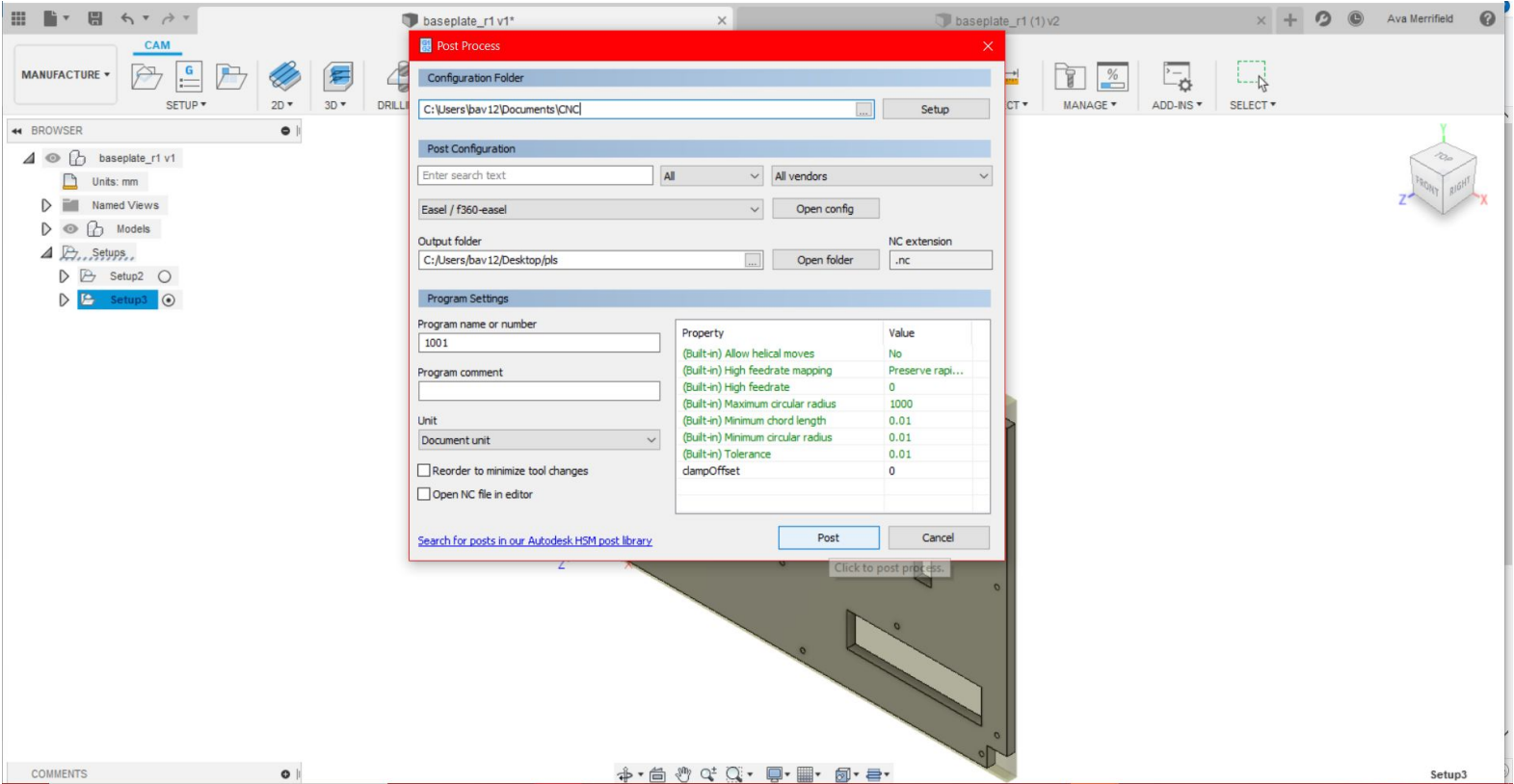
Post Processors

Download the post processor to make your software work with X-Carve.

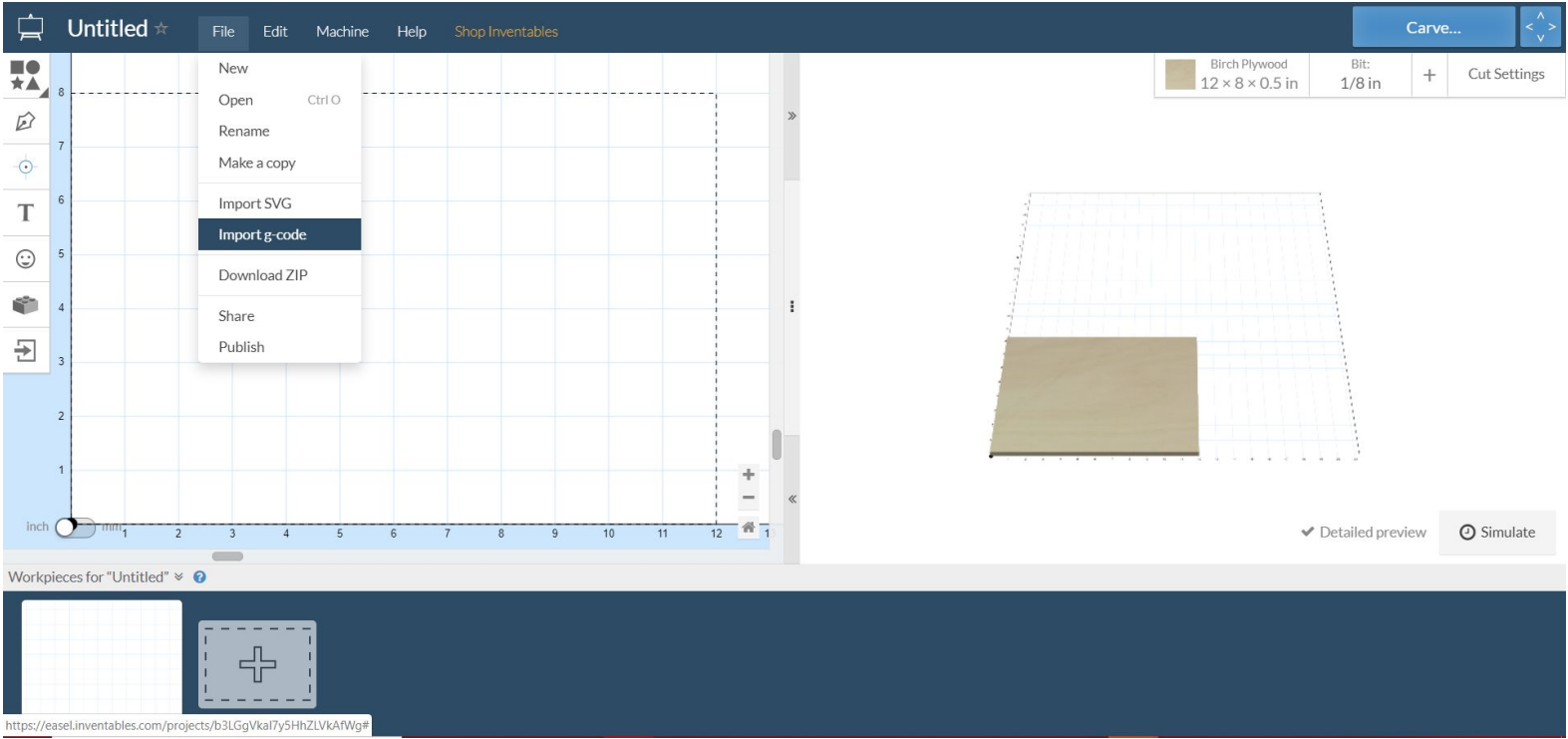
Fusion 360

Vectric Aspire

4. Where it says “Configuration Folder”, make sure you’ve selected the folder that contains your post processor. Your output folder should be a folder you have easy access to, maybe one on your desktop. Do not select “Open NC File in Editor”. Click “Post”.



5. In Easel, click on “File” and select “Import G-Code”. Click on the file in your output folder that you want to cut.



5. In Easel, click on “File” and select “Import G-Code”. Click on the file in your output folder that you want to cut. Your part should appear on the grid. Make sure it’s set up the same way you have it set up on the wastebord.

