Fusion 360 to Shapeoko2 at PS1

This document is a step by step set of instructions to:

1) draw a 2D design in Fusion 360
2) specify the tool and toolpath in the CAM process in Fusion 360
3) post process the design to gcode in Fusion 360
4) run a 3rd-party software program to send the gcode to the Shapeoko2..

This is not a complete course in any tool in the tool path but is designed as a walk through of the process from start to finish, with an example. Detailed help and Youtube information is widely available independently for each application in this toolpath.

Fusion 360 is an amazing product, the example in this document can be done more efficiently with other products but I believe there is value in stepping all the way through a project before tackling more complex projects. That is the purpose of this tutorial.

Download Fusion 360

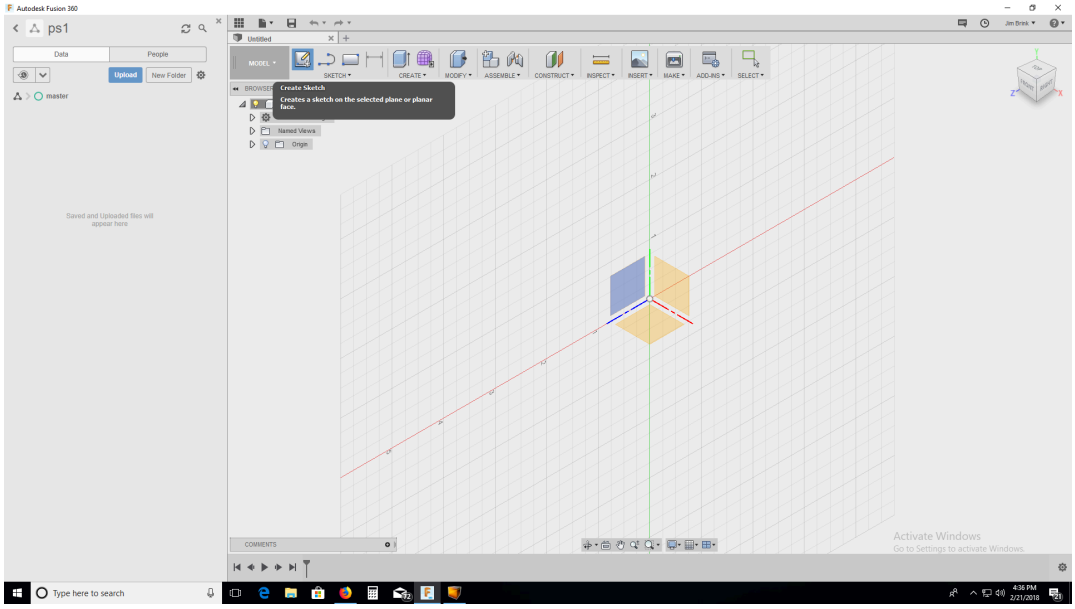
Fusion 360 is available for free download with an Autodesk educational, or “enthusiast” license. Instructions here and here. Because it’s essentially cloud software, you will be able to download and run this software on a PS:One workstation without an administrative login.

Model - Design 2d object

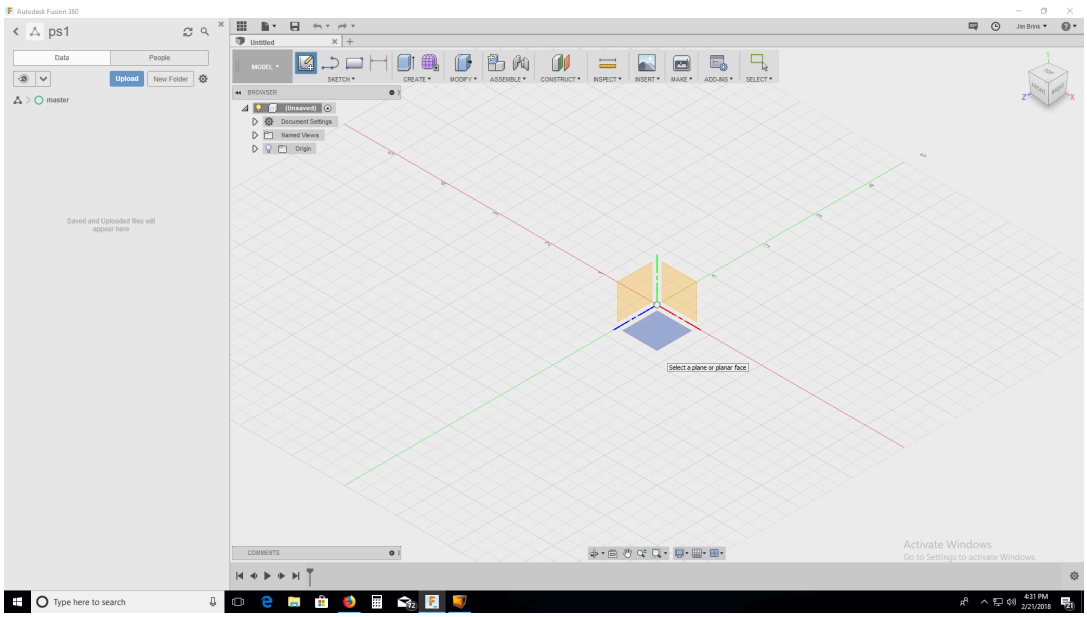
NOTE: If you already know Fusion 360, creating an indented torus (create >torus) will give you enough of a test design to skip to step 12 on p.7

1. Open Fusion 360, when it opens you should be in the ‘Model’ function.

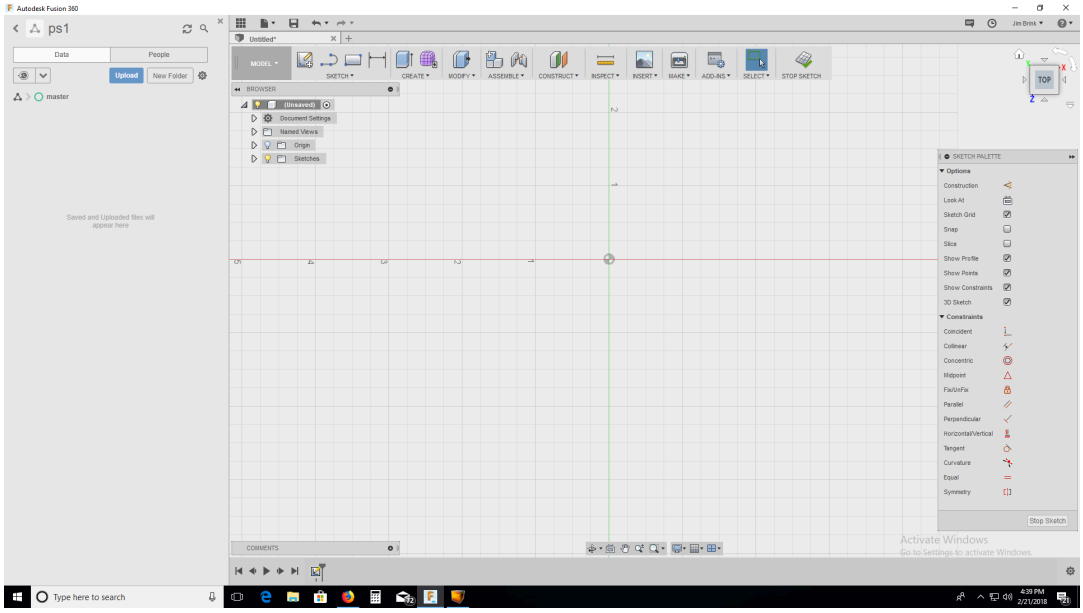
2. Click on ‘Create Sketch’ - it is the first item in the tool bar



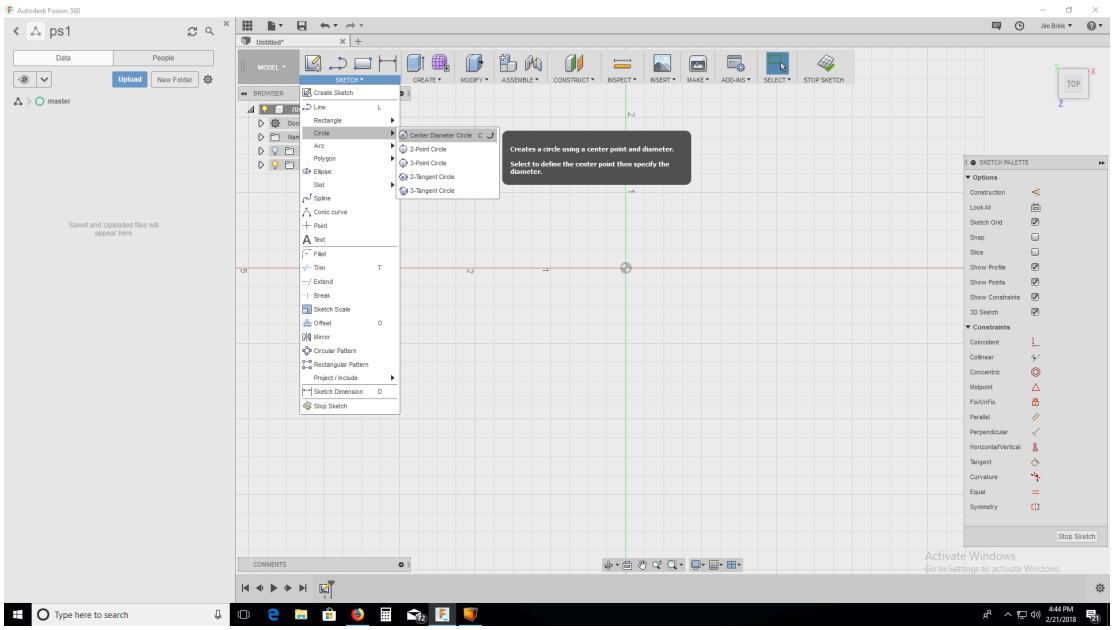
3. Select the plane - for example select ‘top’ as highlighted



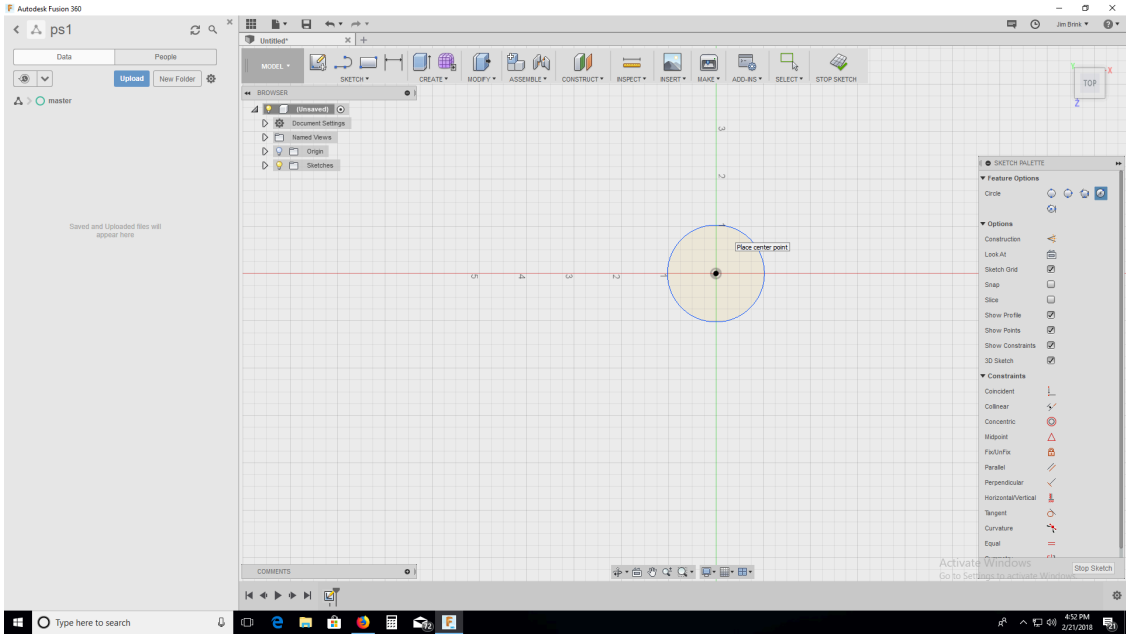
4. The view will change to plane selected, in this case ‘top’ , notice in upper right it indicates ‘top’. Buttons in the bottom of the screen control the view. If needed change the grid to inches, 1 in per major line.



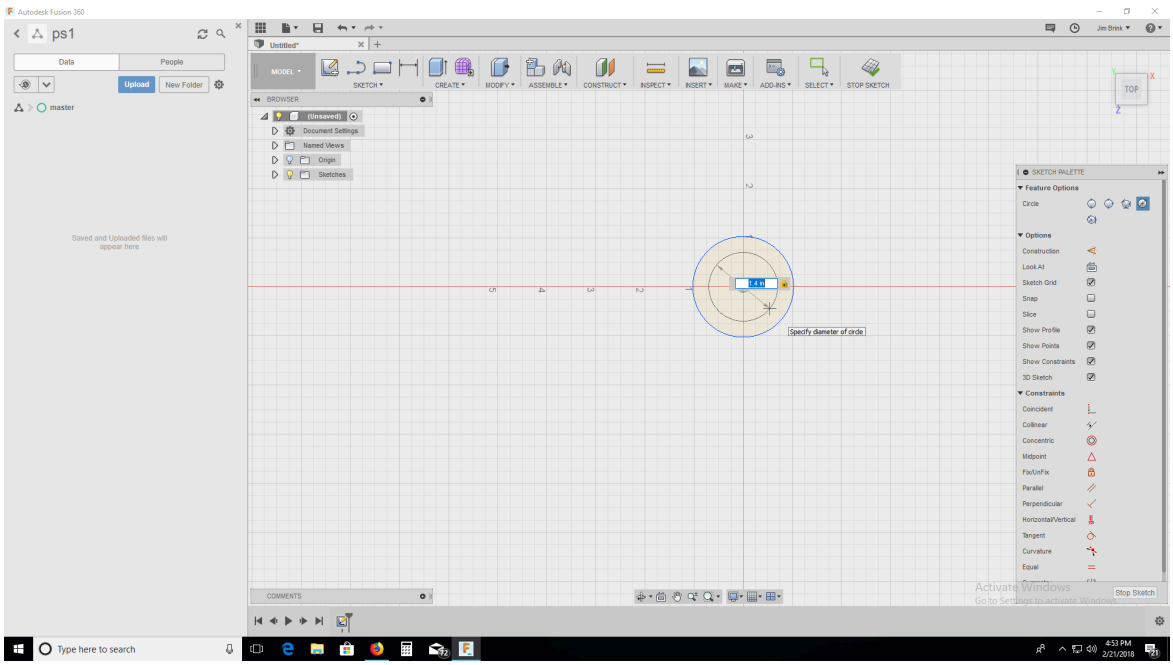
5. Next step is to draw the sketch on the plane. Select ‘sketch’ on the tool bar, then ‘circle’ then ‘center diameter circle’



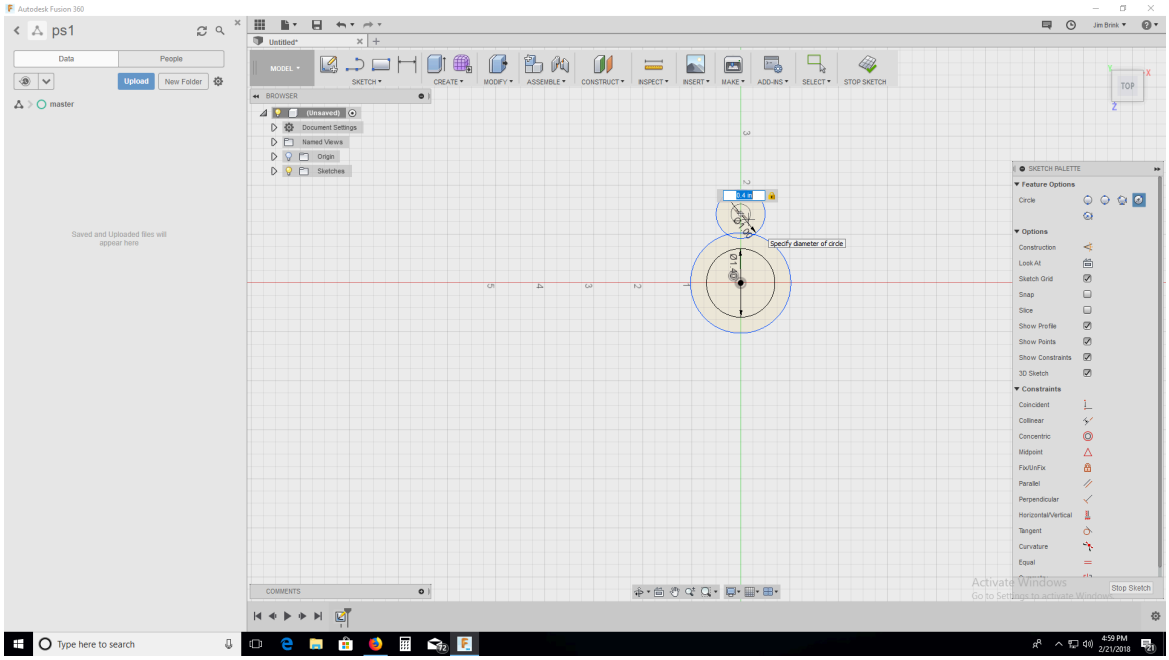
6. Move pointer to the spot in the graph where you want the center of the circle and click. Then move pointer to the diameter of the circle and click. In the example we drew a 2” circle.



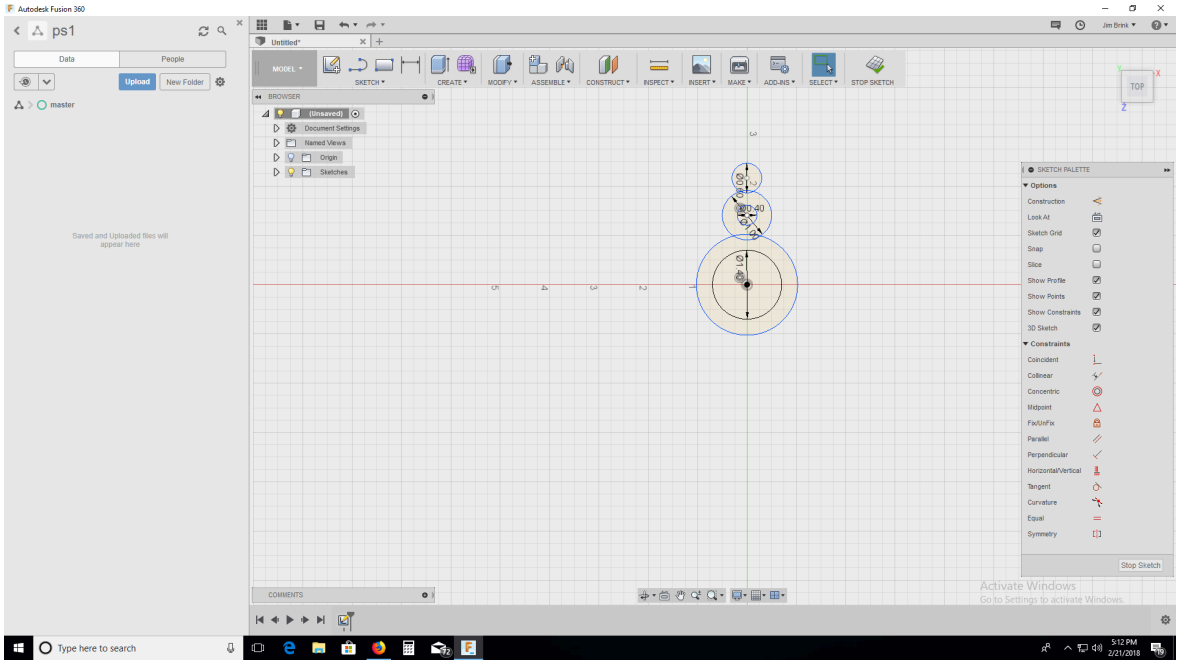
7. Draw a second circle by clicking on the center then draw a 1.4” circle.



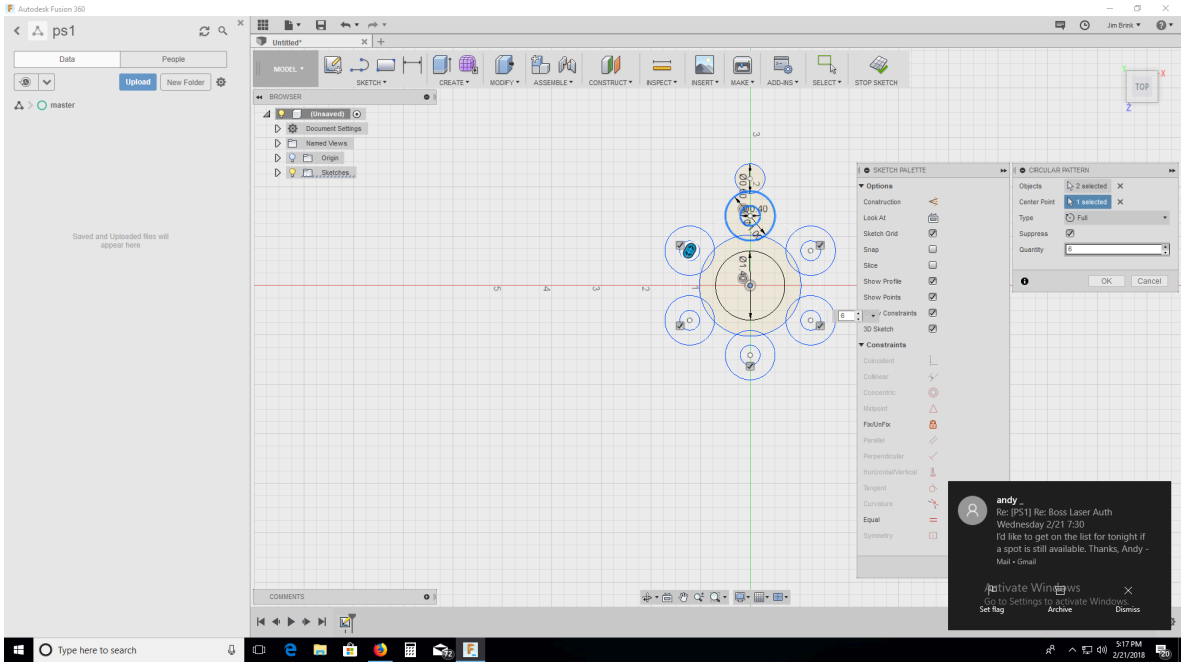
8. Draw a 1” circle with a .4” circle above the first circle.



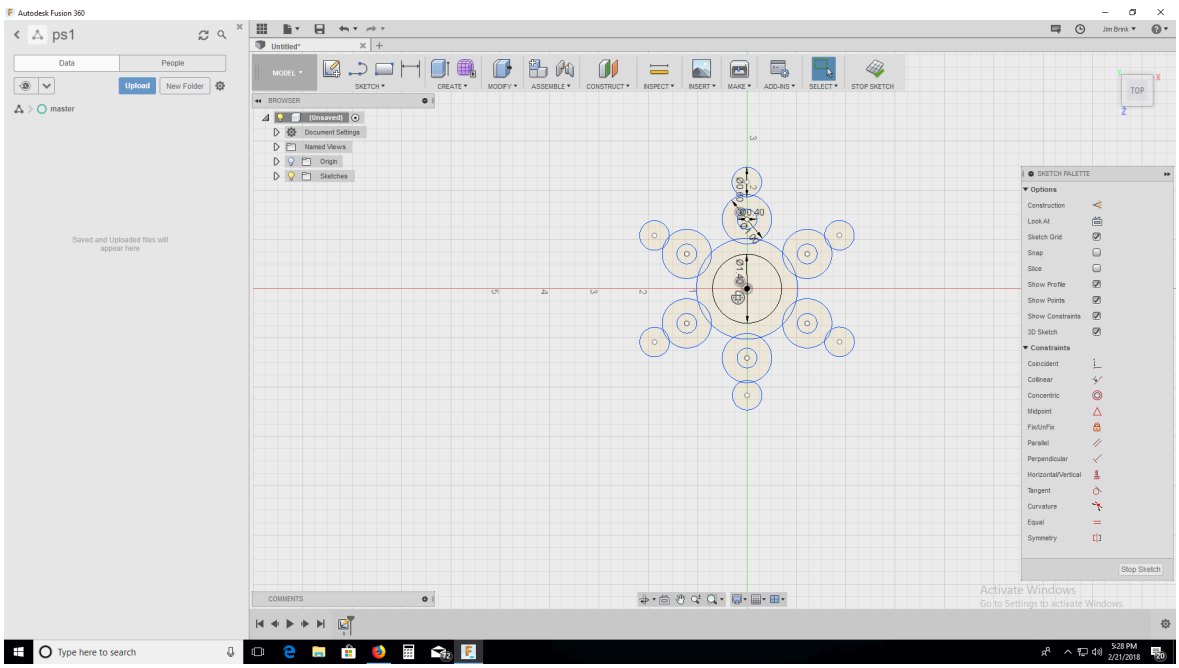
9. Draw a .6” circle above the 1” circle. Press escape to exit the sketch circle function.



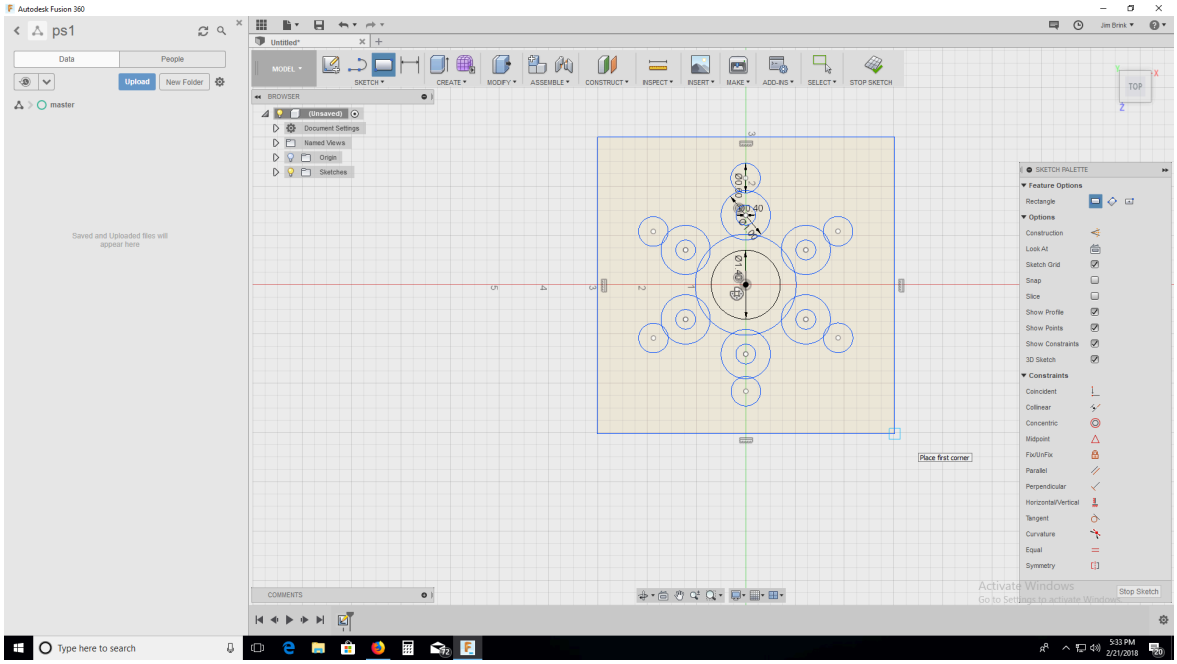
10. Under sketch, select ‘circular pattern’, a menu will appear on the right. Click the outer ring on the 1” circle and the ring on the circle within the 1” circle. In the ‘circular pattern’ box after ‘objects it should indicate ‘2 selected’. Click in ‘center point’ and click on the dot in the 2” circle. Three circles should appear, change the quantity to 6. Click OK to save.



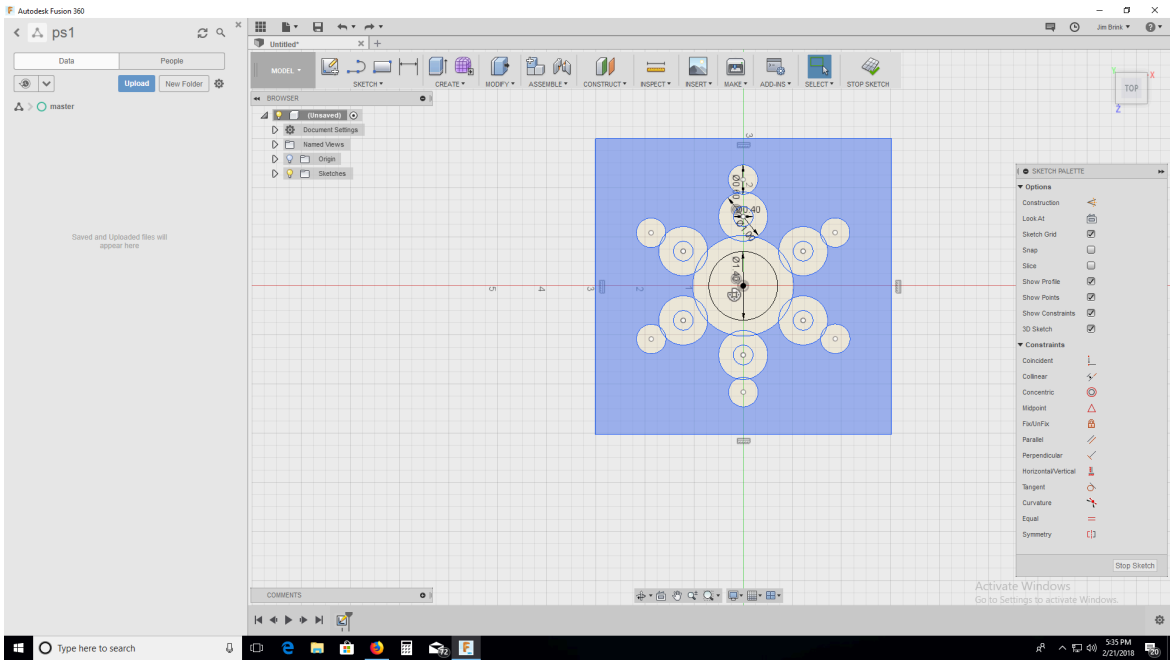
11. Follow the same circle pattern process for the .6” circle.



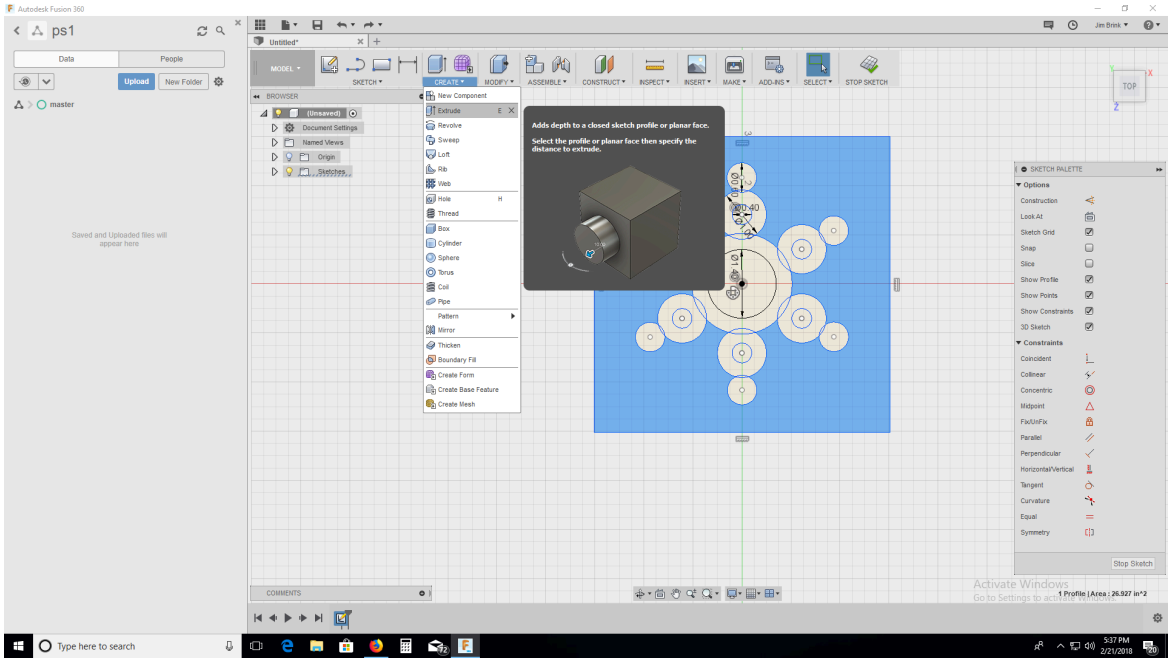
12. Under sketch select ‘rectangle’ then ‘2 point rectangle’ . Click 3” up from the base line and 3” left of the center. The pull the rectangle to 3” below the base and 3” right and click. This will be the base material we will cut the logo in.



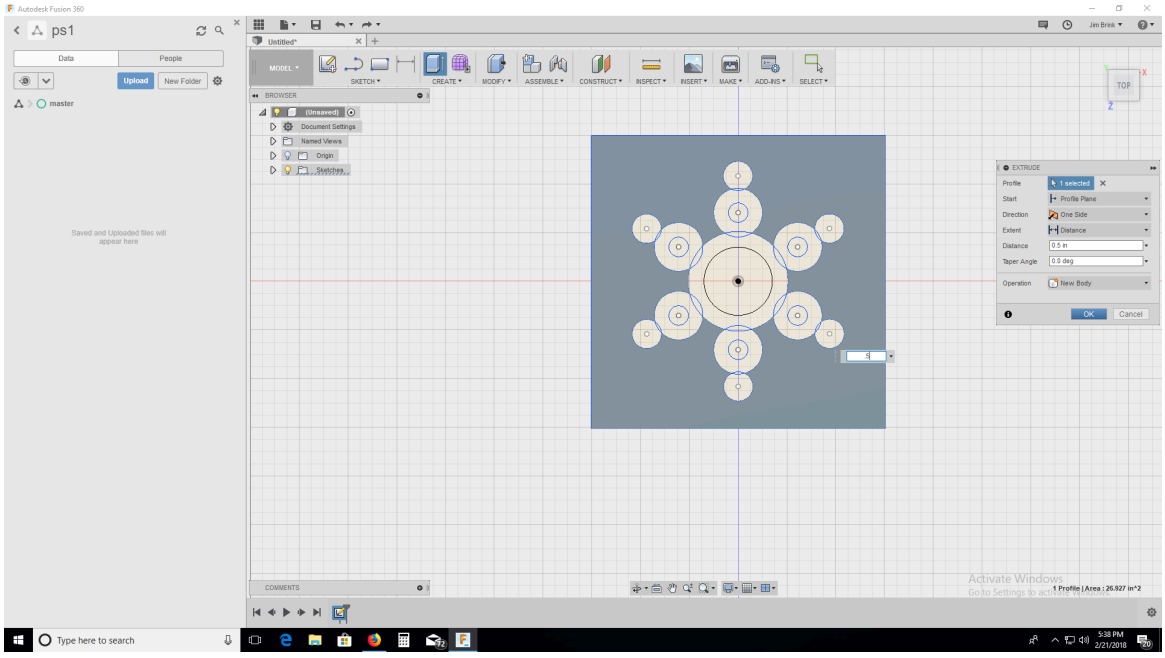
13. Move the arrow in the base area until the base turns blue and click.



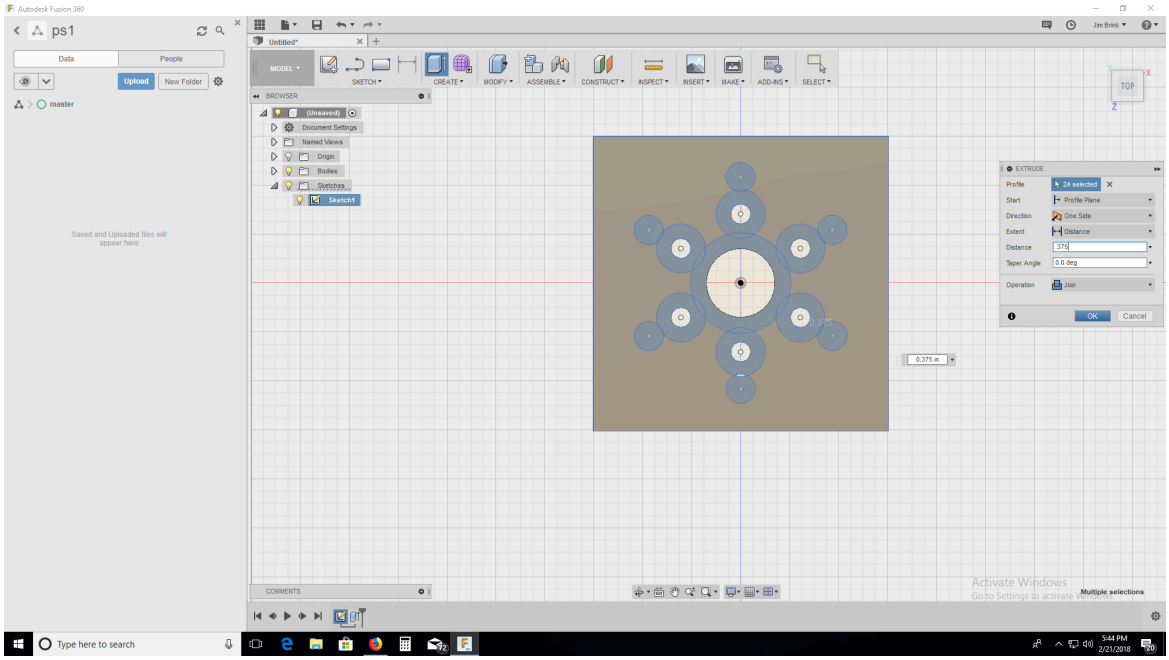
14. Select ‘create’ then ‘extrude’



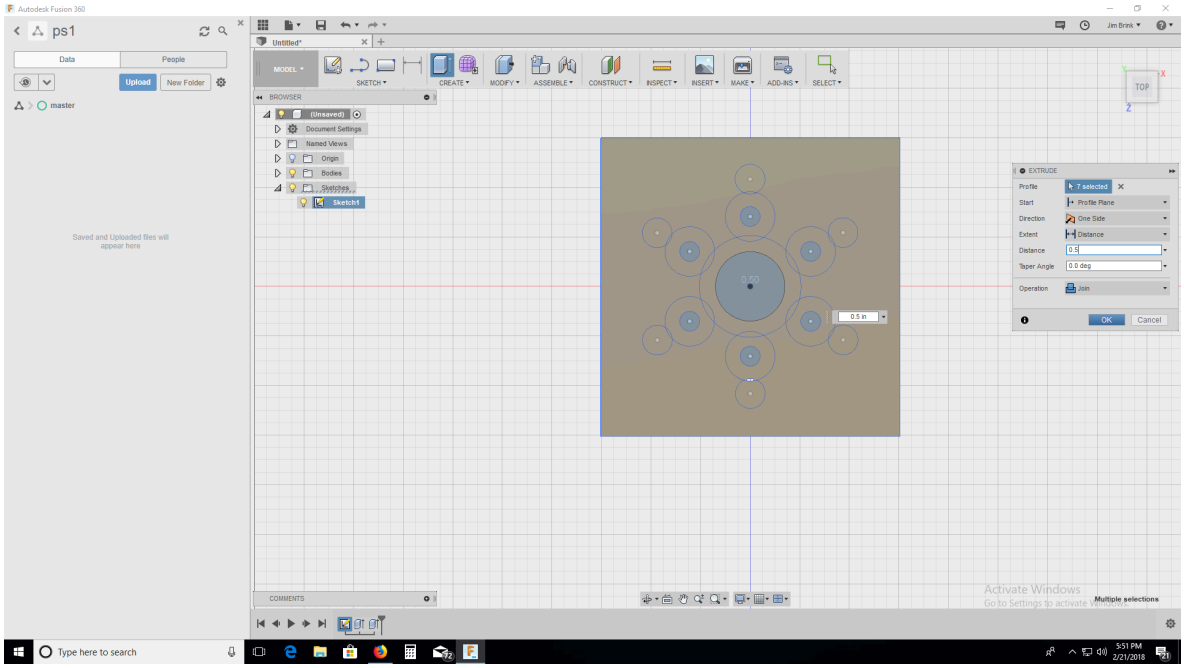
15. Click on the blue base and a dialogue box extrude will appear on the right. It should say ‘1 selected’, which is the background. Enter .5 in ‘distance’ and click ok. The base is now .5” thick.



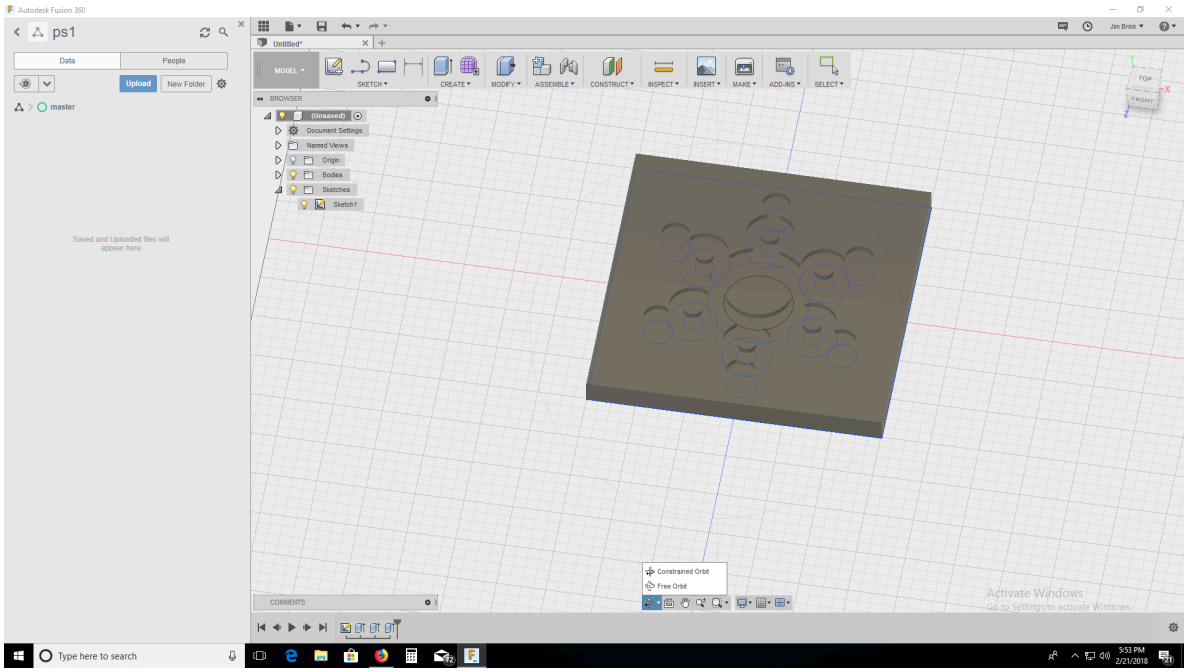
16. Click on ‘sketches’ and make sure ‘sketch 1’ on in the drop downs on the left . Then select ‘create’ and ‘extrude’ and select each of the circles indicated in the following screen print. Enter a distance of .375 and click OK.



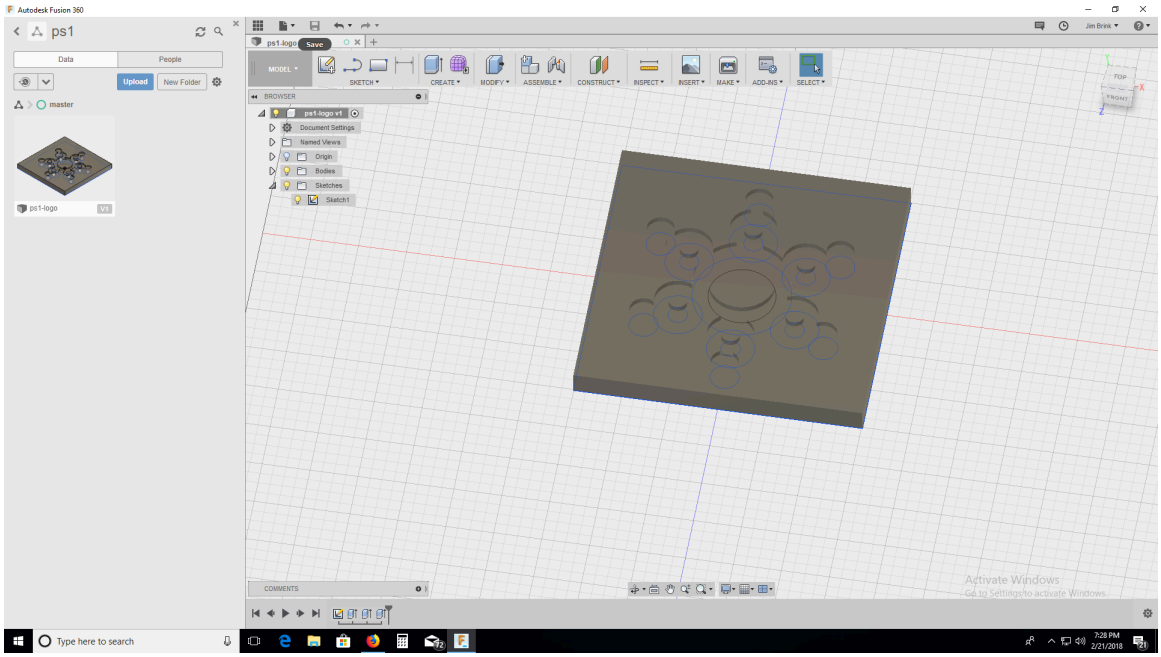
17. Repeat the extrude commands for the remaining seven objects but enter .5 for the distance. Be sure to click ok.



18. Click the ‘orbit’ button in the center on the bottom, select the base and orbit the object to view the design. You can also orbit by clicking and dragging the cube in the upper right.

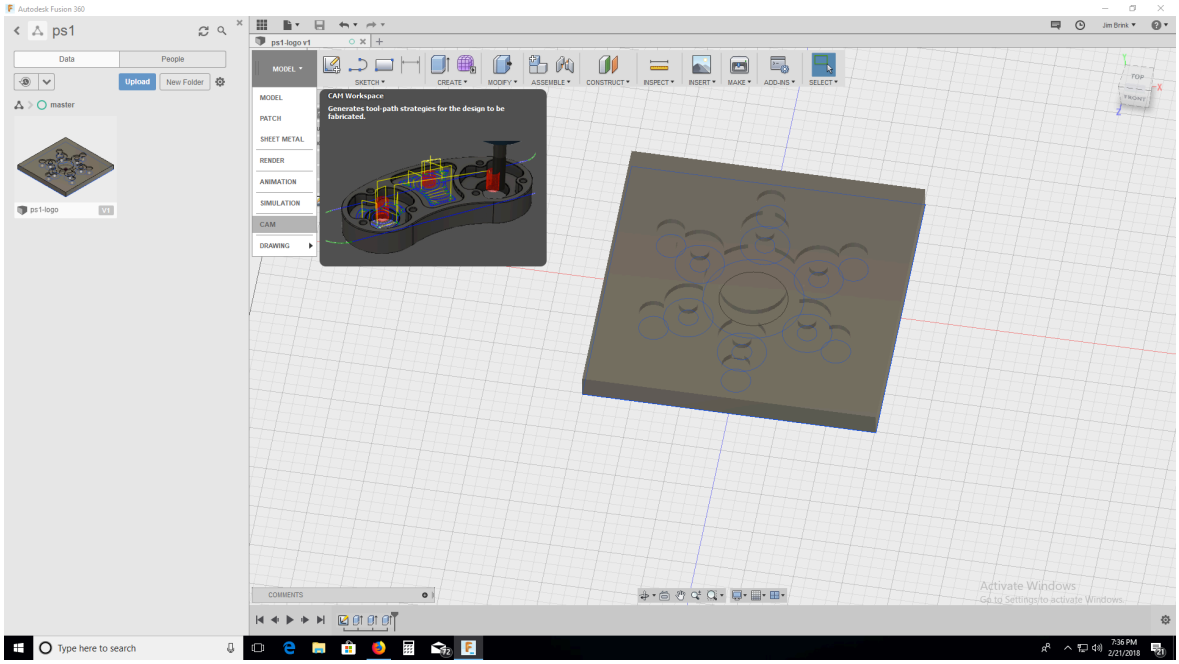


19. Save and name the design. It should show as a design in the left section.

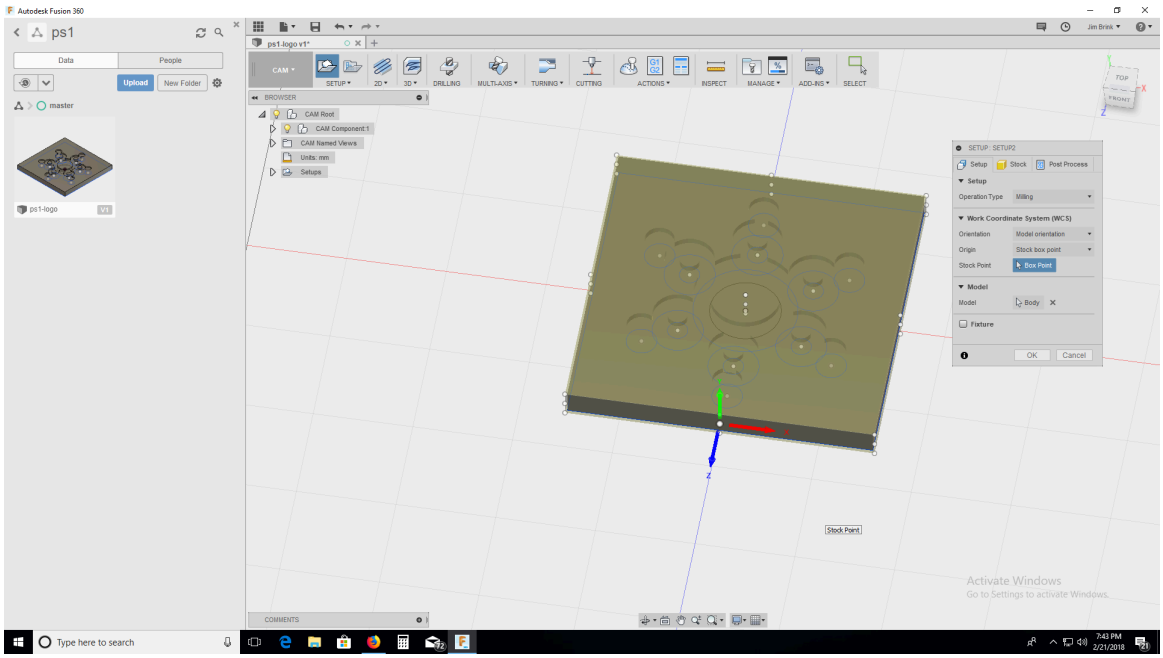


CAM Process - Tool definition and Toolpaths

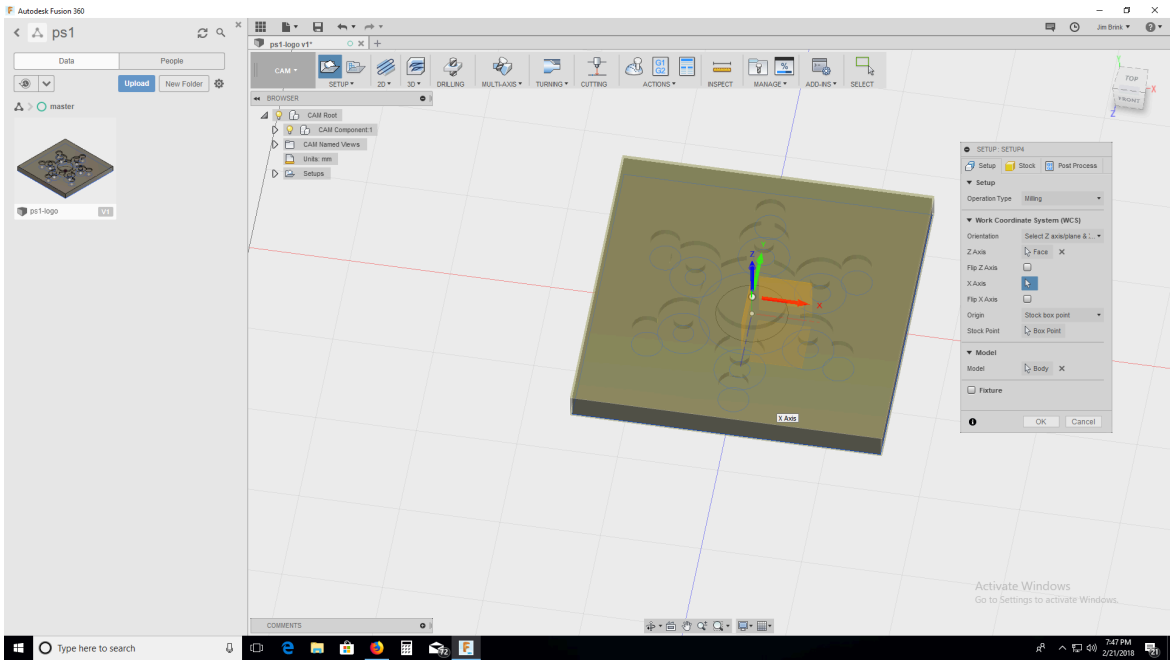
20. Select CAM



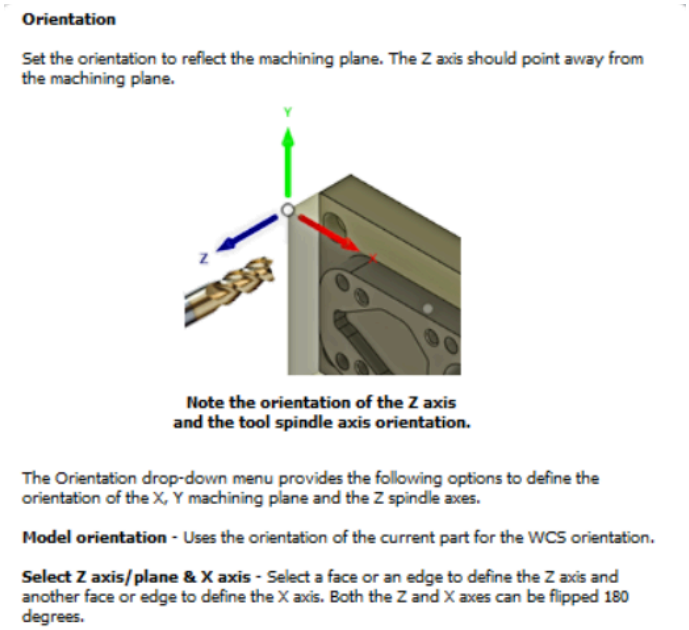
21. Select setup, new setup. You should see the following screen with the blue (z axis) pointing to you or in the Y axis.and x access'.



22. Click on ‘model orientation’ in ‘Orientation’ of the dialogue box. Select ‘z axis/ plane and y axis’ option. Click it, then move the arrow to the body of the object and click. The z axis (blue) should change to up. The y axis (green) points to the back and the x axis (red) across. Click ok to save this orientation. This matches the orientation on a mill, the z axis is the spindle, the x axis the table side to side and the y axis table front to back.



For more help orienting the Z axis to point upward, mouse over “Orientation” for the following direction pop-up. Click the 1st “Select Z axis/plane” option of the two.



Orientation

Set the orientation to reflect the machining plane. The Z axis should point away from the machining plane.

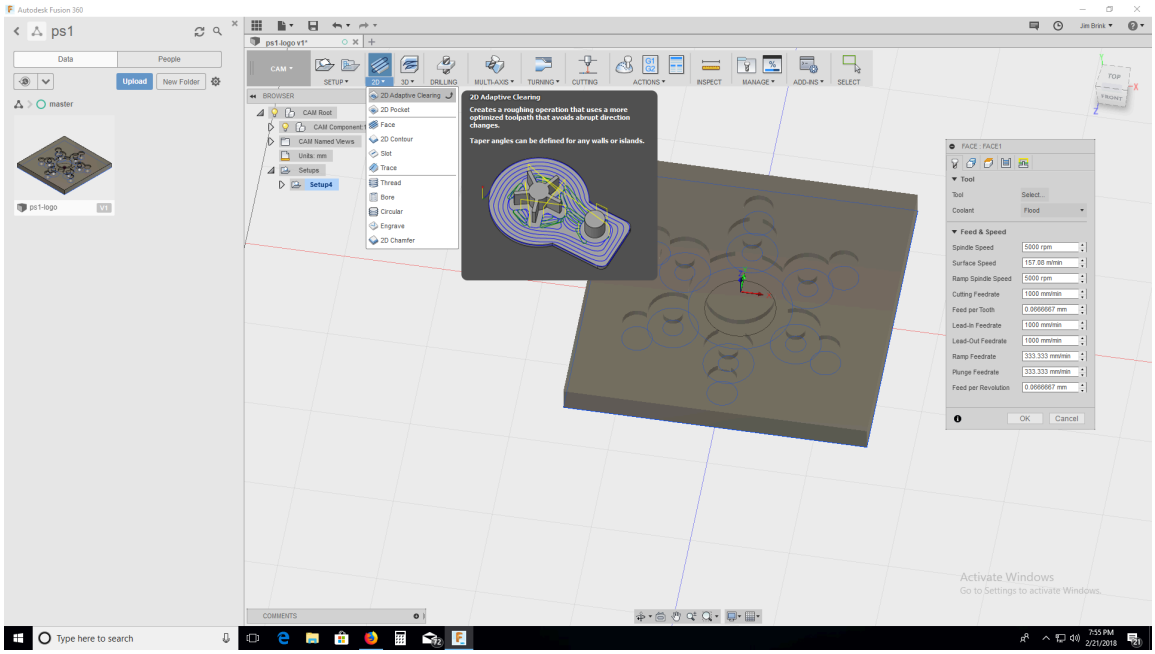
Note the orientation of the Z axis and the tool spindle axis orientation.

The Orientation drop-down menu provides the following options to define the orientation of the X, Y machining plane and the Z spindle axes.

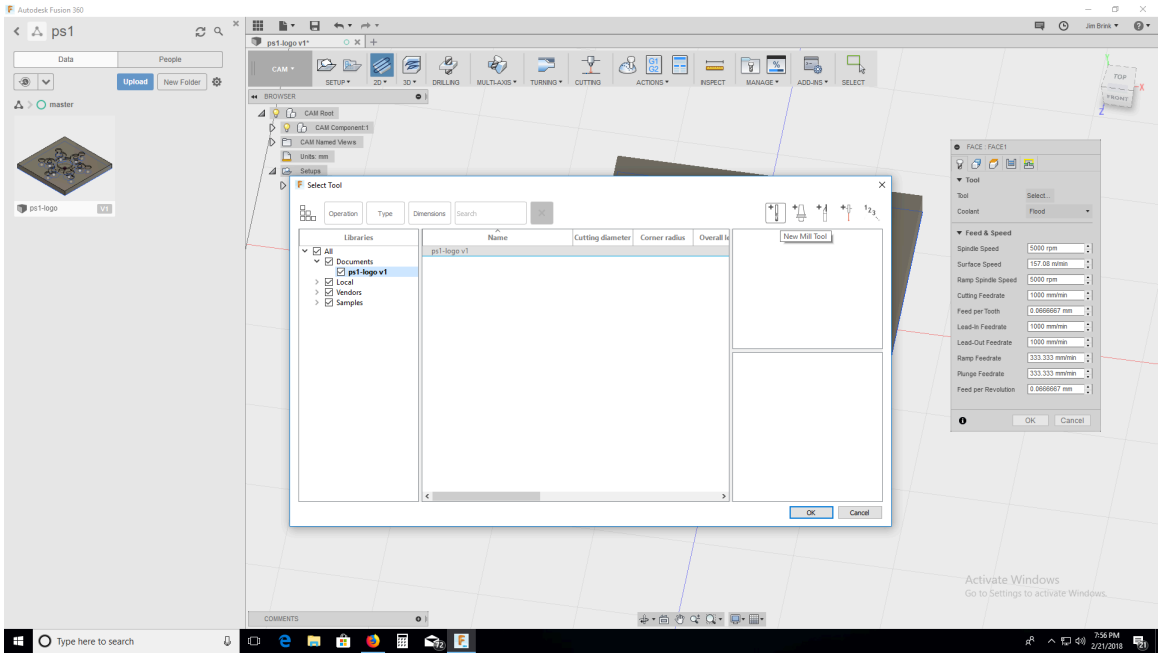
Model orientation - Uses the orientation of the current part for the WCS orientation.

Select Z axis/plane & X axis - Select a face or an edge to define the Z axis and another face or edge to define the X axis. Both the Z and X axes can be flipped 180 degrees.

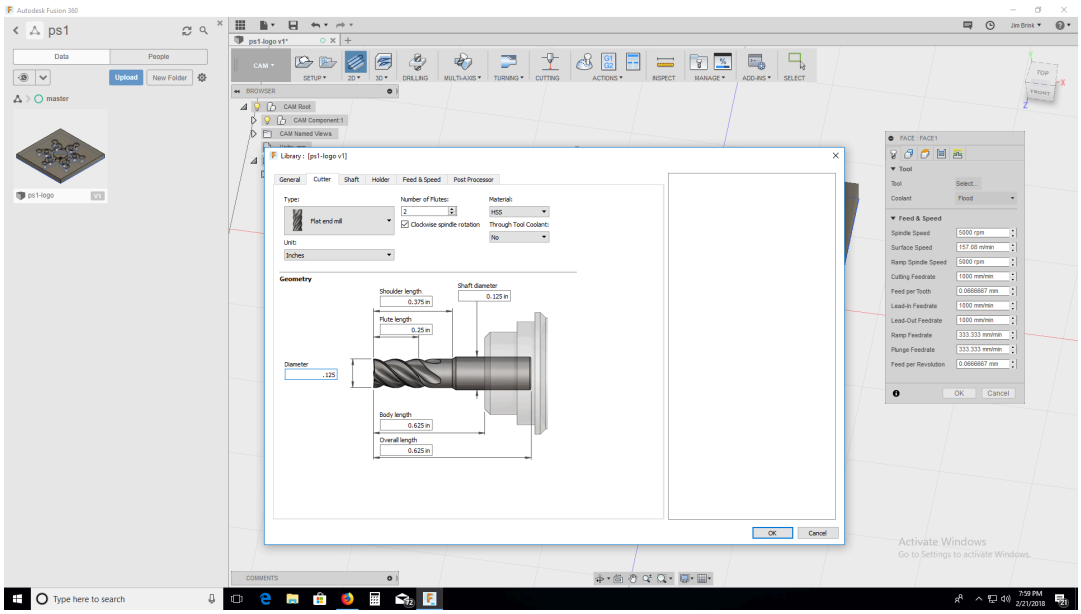
23. Under 2d select ‘adaptive clearing’



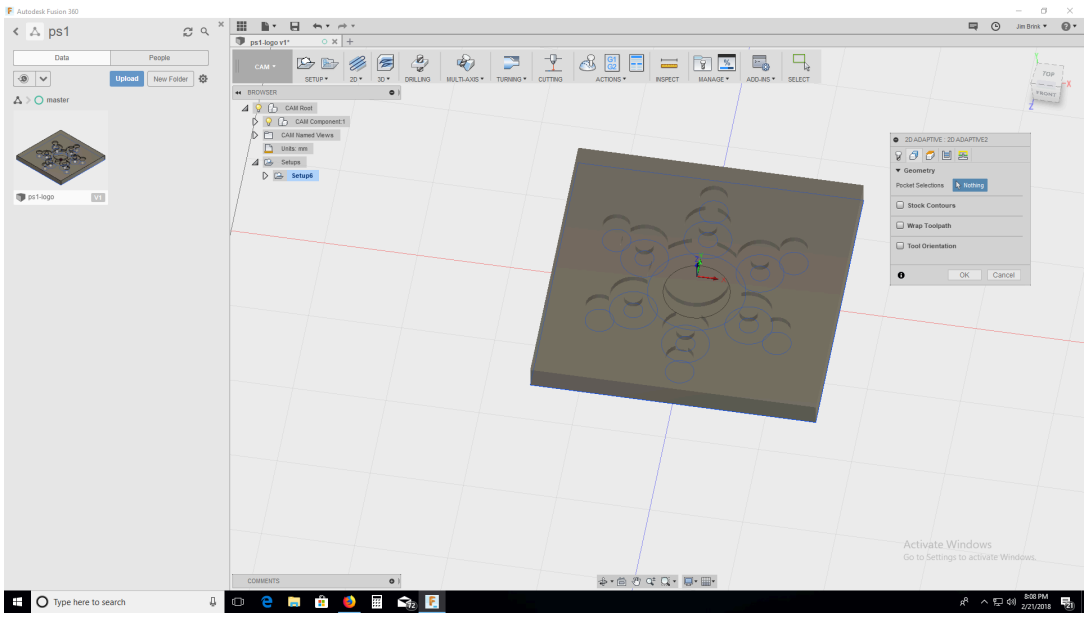
24. In the box that opens for ‘adaptive clearing’ click on ‘select’ at tool. You will get a window to define a tool.



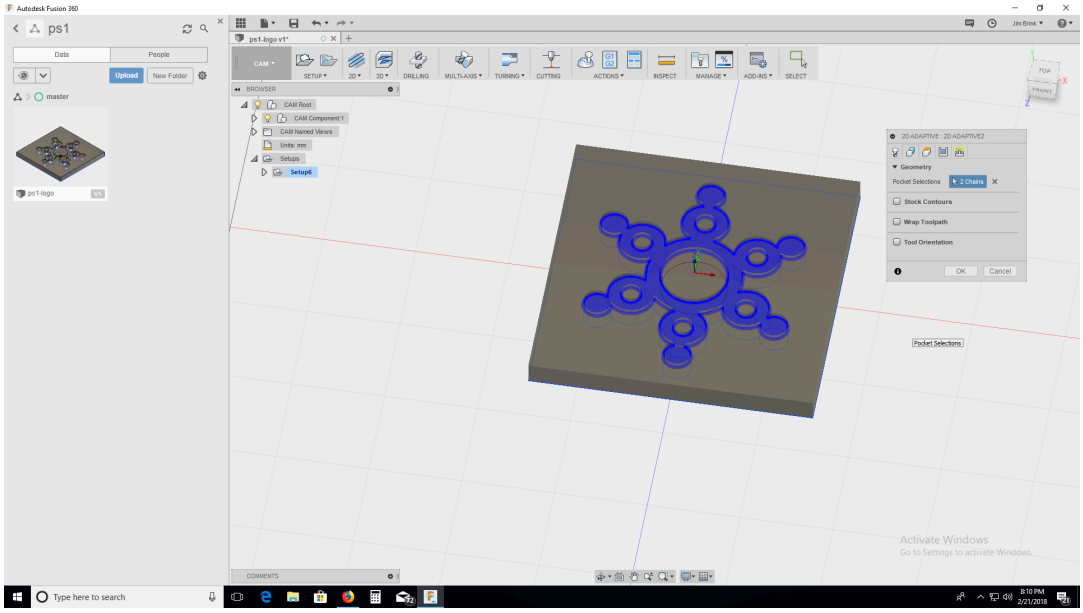
25 For a ⅛ sq end mill select the ‘new mill tool’ at the top. You will get a screen to define the mill. I selected type ‘sq end mill’, changed the unit to in, number of fluites to 2 and changed the diameter to .125. Click ok to save the tool definition.



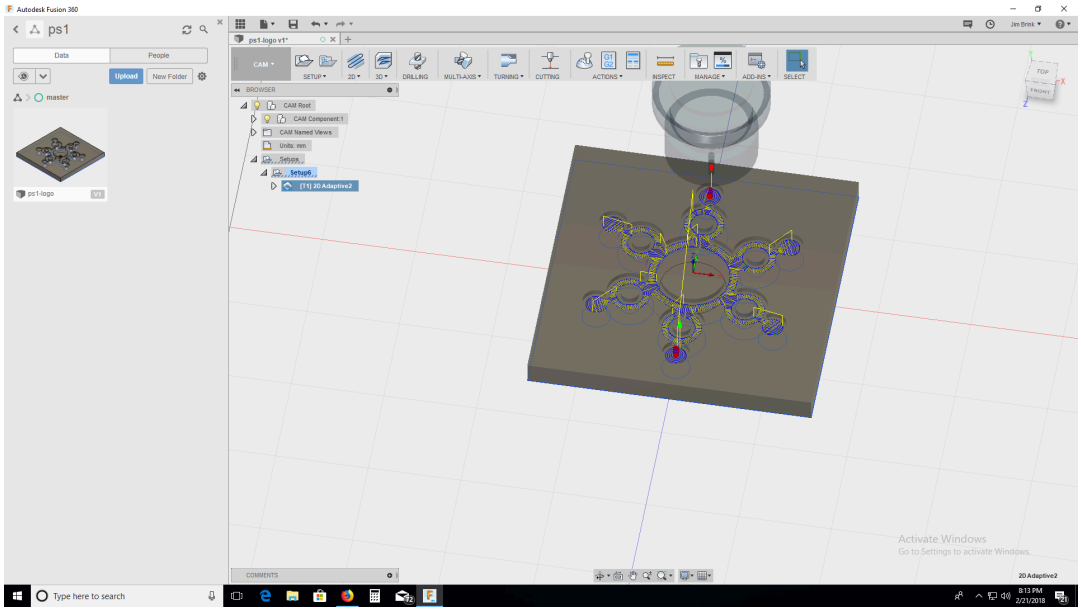
26 Select ‘geometry’ , the second button in the ‘adaptive clearing’ box.



27. Move the arrow over the design until the design lights. Click to choose the chain for pocket selection. If any part of the design did not light move the arrow to that part and click to select.



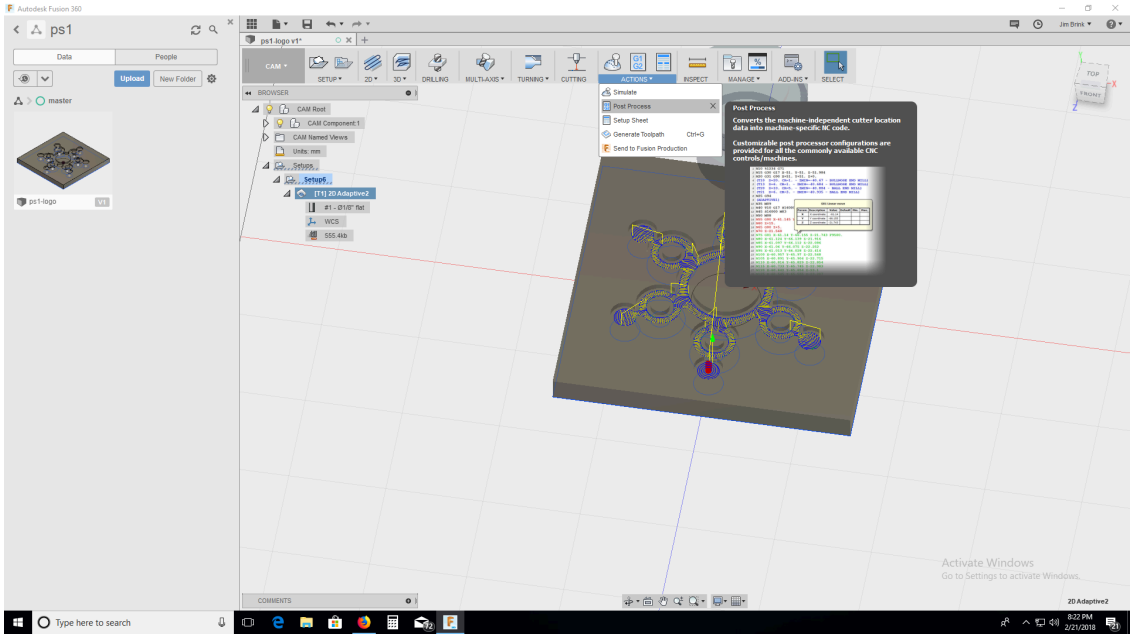
30. Click on ok in the box, you should get an additional line in the browser list that sez ‘2d adaptive 2’. It will take some processing time but the tool path will be displayed in blue, yellow and red lines.



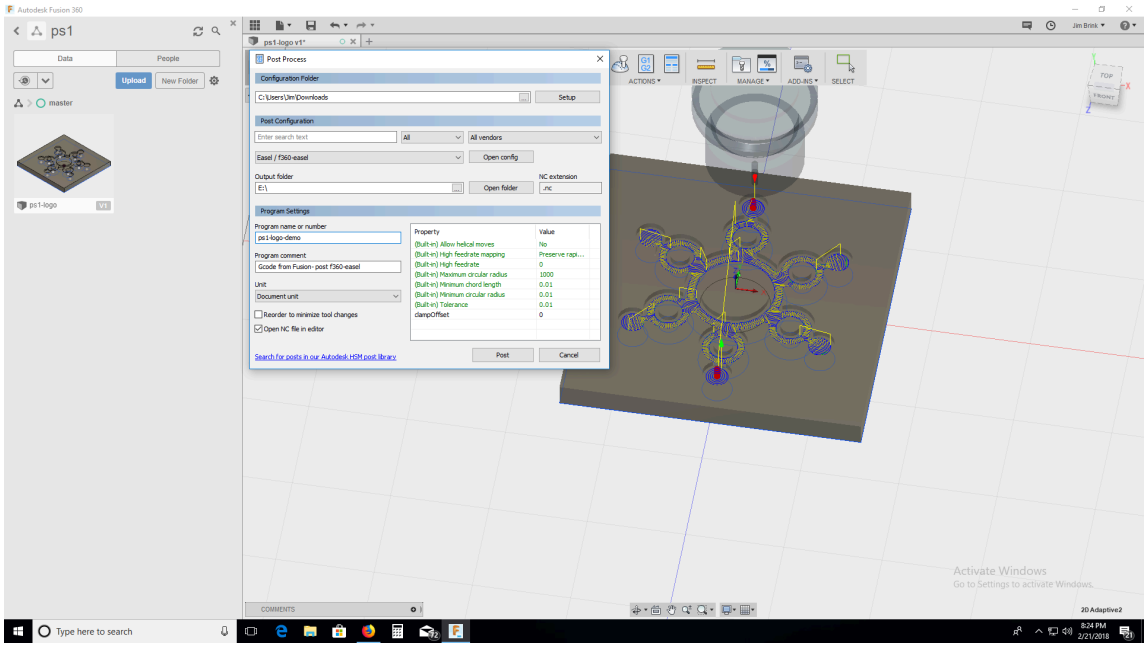
Post Process- build Gcode

Once your model is complete, the post-process steps will prepare your model for milling by converting it to Gcode, which can be read by 3rd party software products that connect to the Shapeoko.

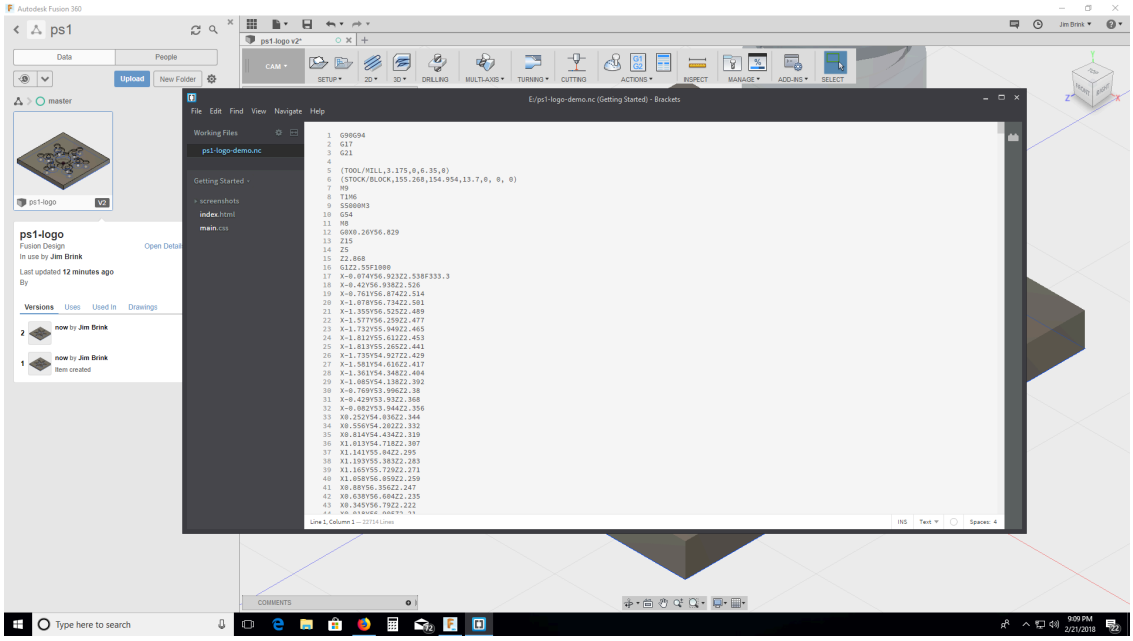
31. Under ‘actions’ select ‘post process’



32. I used the Easel post processor saved on my machine. Other processors may work but I have not found a loaded processor that works.



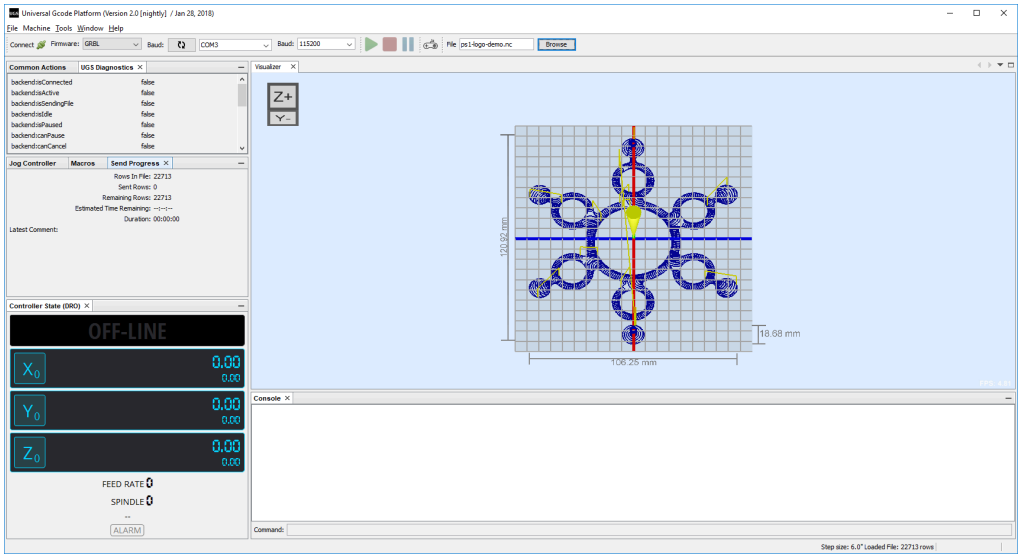
33. The gcode will be displayed. Close the screen to proceed. The gcode file will be found in the directory specified in the post process screen.



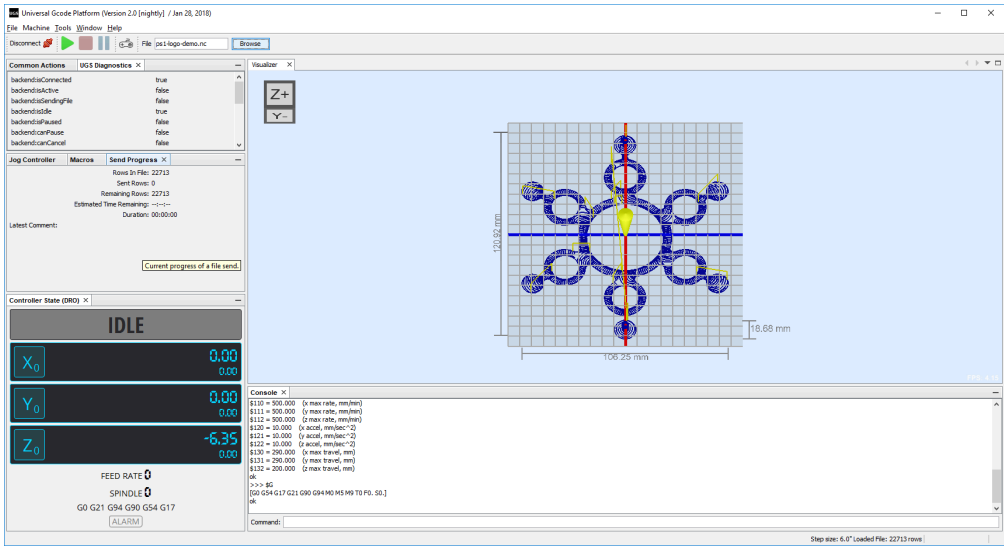
Execute Gcode - Universal Gcode Platform

34. Many Gcode processors (3rd party software products) may be used, Easel, the post processor from the makers of Shapeoko2 (Inventables Corporation), is recommended and available on the Inventables site. A 2nd option is to use Universal Gcode Platform https://winder.github.io/ugs_website/#downloads

Open the gcode processor (Universal Gcode shown below). Browse and open the Gcode file



35. Select the correct port and press the connect, it should change to ‘disconnect’ and several messages will be displayed in the console.



36. Secure the stock material in the Shapeoko2.

37. Put the ⅛ in bit in the spindle.

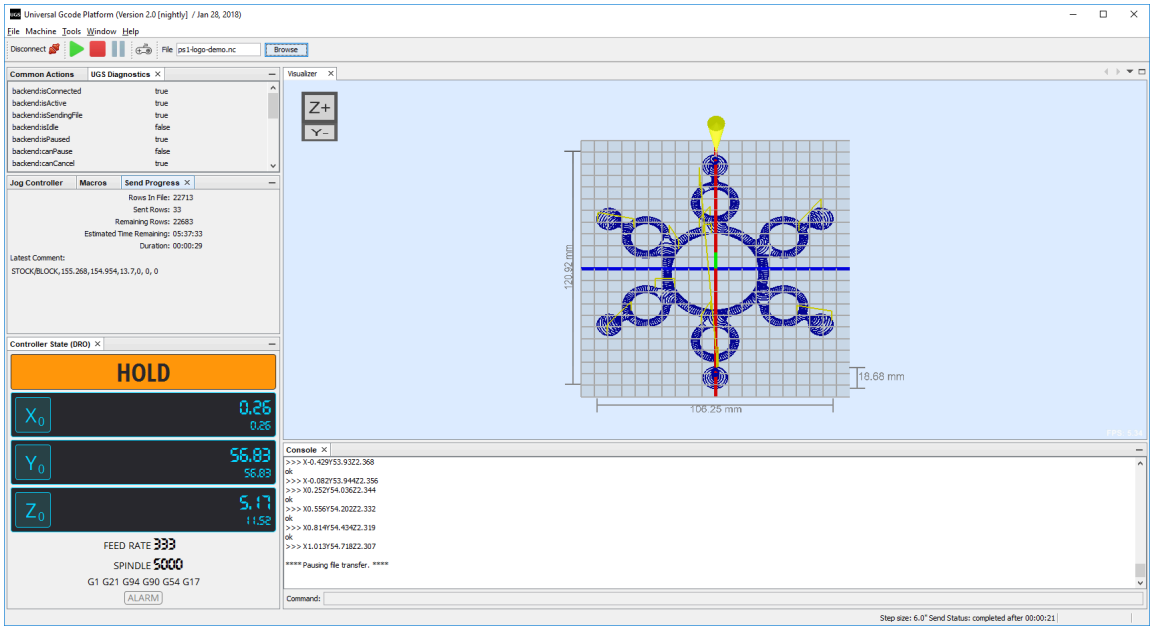
38. Position the spindle in the center of the stock, see the diagram in the gcode sender.

39. Make sure power is on for the Shapeoko2

40. Turn spidle to 2000 rpm.

41. Press the green arrow to run the gcode. The executing gcode will be displayed in the console and program time and progress will be displayed in the ‘Send Progress’ window.

Note: I get an error about a T1M6 command but hit enter to bypass. Also the program stops after moving to the start position, also hit the green arrow to proceed.



Alternative: Execute Gcode Using Easel

Run the cloud-based Easel software from the Inventables site. The software wizard will take you step by steps through the tool setup, the material and thickness, and homing the drill bit to the correct X, Y, and Z position.

Select the green “Carve” button in the upper right corner when the settings are entered, the material is securely fastened to the base, and the tool is positioned correctly above your material.