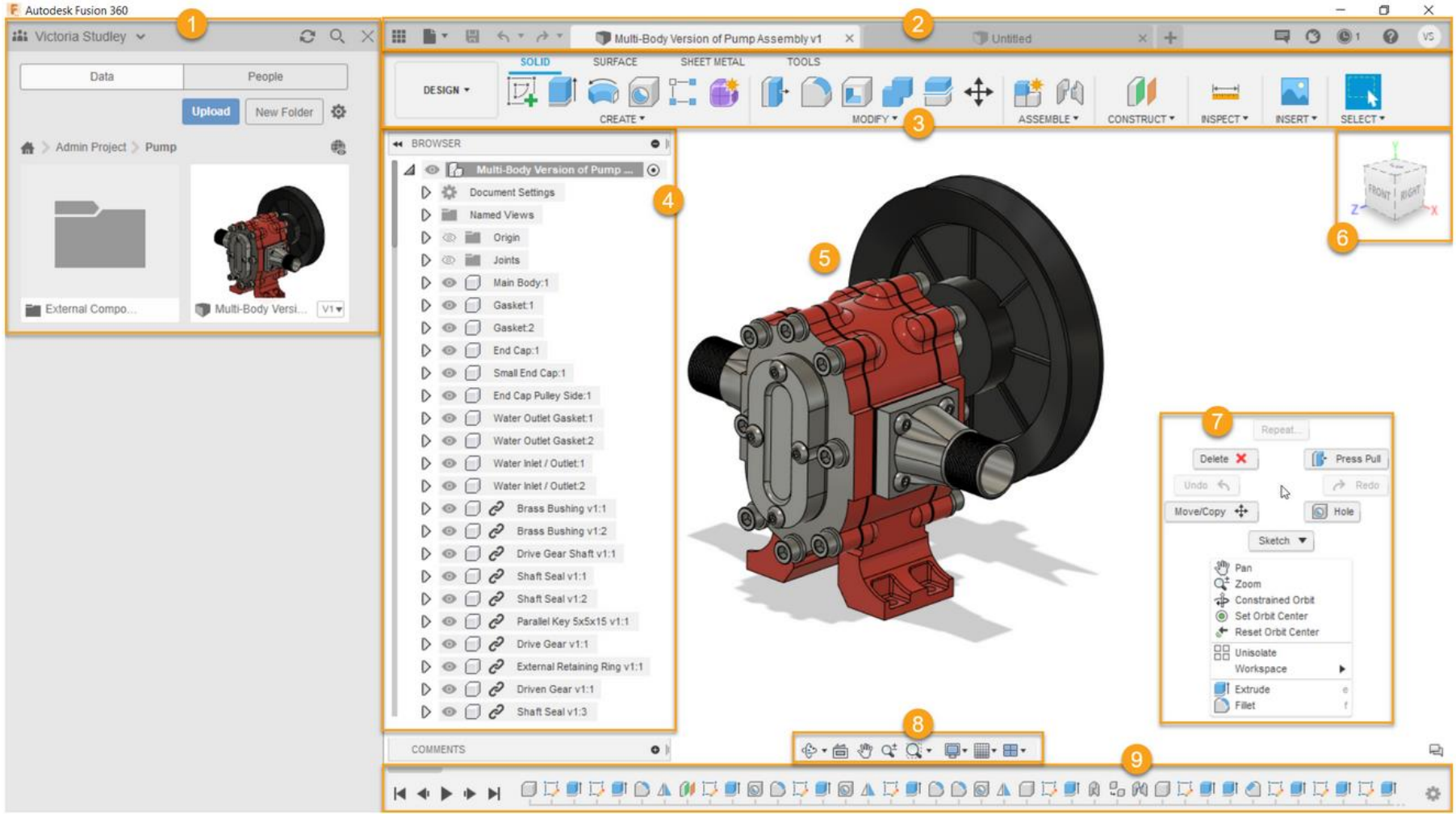
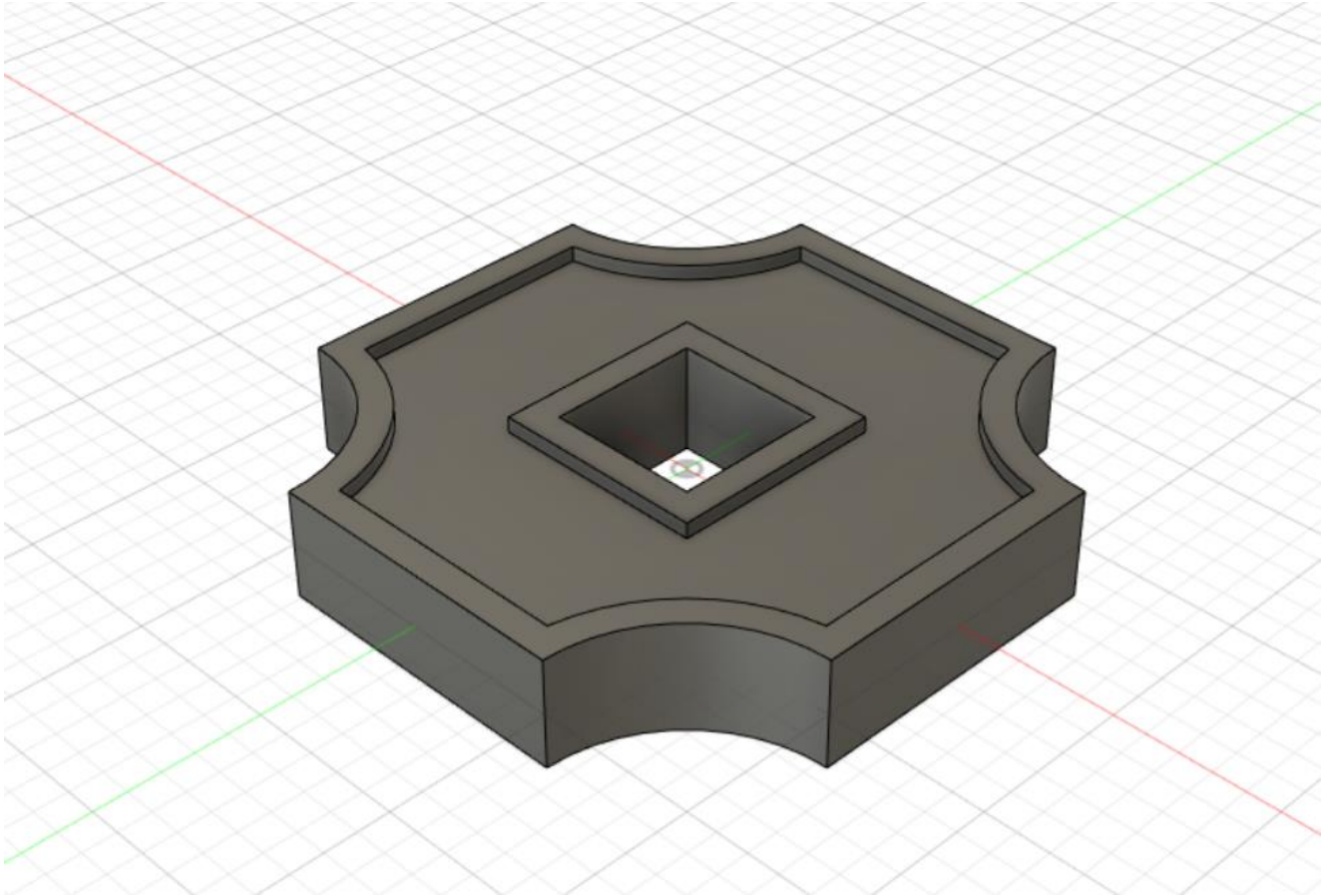
AUTODESK®
FUSION 360™


Autodesk Fusion 360
1
Victoria Studley
Data
People
Upload
New Folder
Admin Project
Pump
External Compo...
Multi-Body Versi...
V1
2
Multi-Body Version of Pump Assembly v1
Untitled
SOLID
SURFACE
SHEET METAL
TOOLS
DESIGN
CREATE
MODIFY
ASSEMBLE
CONSTRUCT
INSPECT
INSERT
SELECT
3
BROWSER
4
Multi-Body Version of Pump ...
Document Settings
Named Views
Origin
Joints
Main Body:1
Gasket:1
Gasket:2
End Cap:1
Small End Cap:1
End Cap Pulley Side:1
Water Outlet Gasket:1
Water Outlet Gasket:2
Water Inlet / Outlet:1
Water Inlet / Outlet:2
Brass Bushing v1:1
Brass Bushing v1:2
Drive Gear Shaft v1:1
Shaft Seal v1:1
Shaft Seal v1:2
Parallel Key 5x5x15 v1:1
Drive Gear v1:1
External Retaining Ring v1:1
Driven Gear v1:1
Shaft Seal v1:3
COMMENTS
5
6
FRONT
RIGHT
7
Repeat...
Delete
Press Pull
Undo
Redo
Move/Copy
Hole
Sketch
Pan
Zoom
Constrained Orbit
Set Orbit Center
Reset Orbit Center
Unisolate
Workspace
Extrude
Fillet
8
9

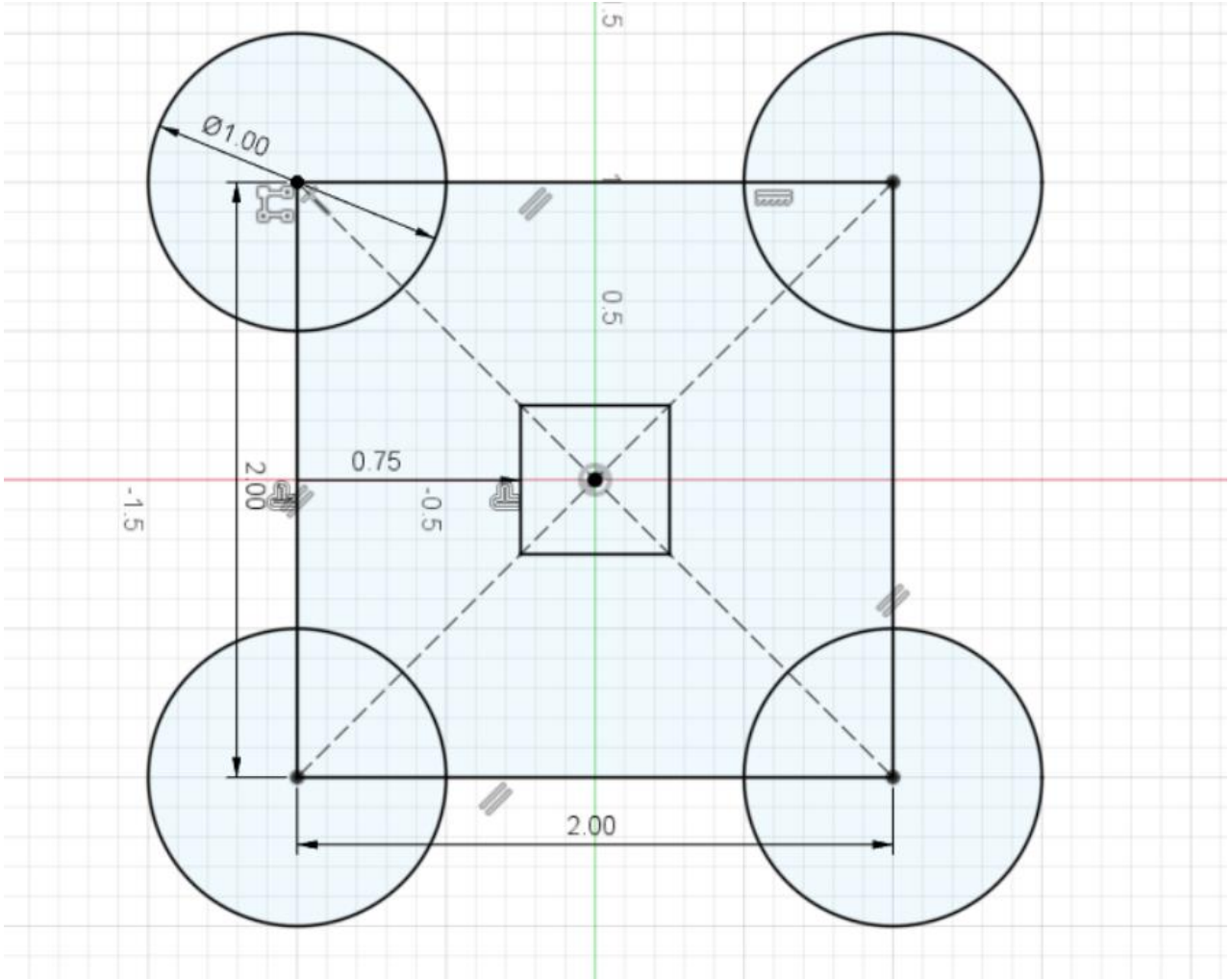
1. **Data Panel**: The Data Panel allows you to access teams, projects, and designs, manage design data, and collaborate
2. **Application Bar**: Contains most of the basic functions as other softwares, such as, undo, redo, Save, creating new designs
3. **Toolbar**: contains most of the basic tools used creating components & sketches
4. **Browser**: Lists objects in your assembly (components, bodies, sketches, origins, joints, construction geometry, etc.) and lets you control visibility of objects.
5. **Workspace/Canvas**: The area in which you will interact with the components you create
6. **ViewCube**: Allows you to orbit your design or view it from different positions.
7. **Marking Menu**: Right-click to access the Marking Menu, which contains frequently used commands in the wheel and additional commands in the overflow menu
8. **Navigation Bar**: Contains commands used to zoom, pan, and orbit your design, as well as display settings to control the appearance of the interface and how design assemblies are displayed in the canvas.
9. **Timeline**: Lists operations performed in your design. Right-click operations in the timeline to make changes. Drag operations to change the order in which they are calculated. (Parametric Modeling mode only)

Coin one

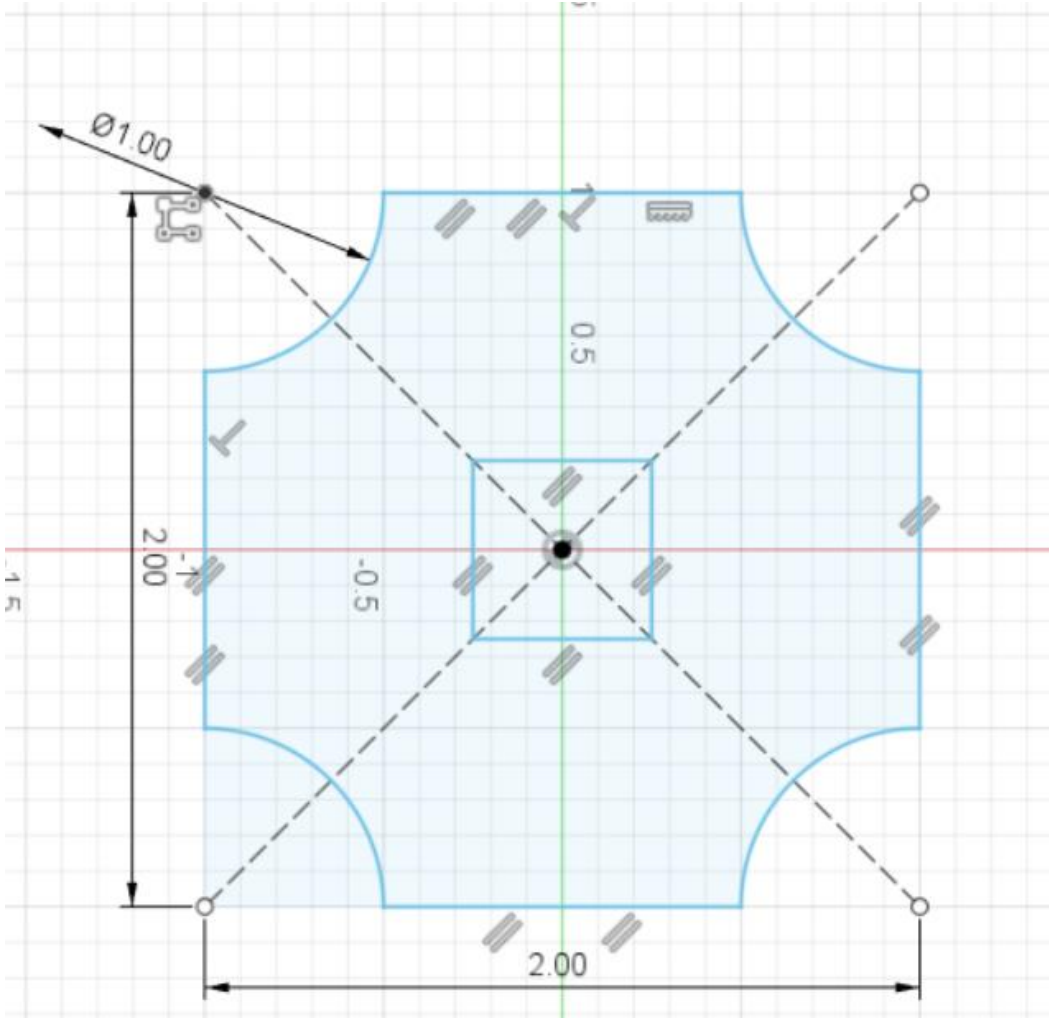


Make sure to start your sketch on the top plane

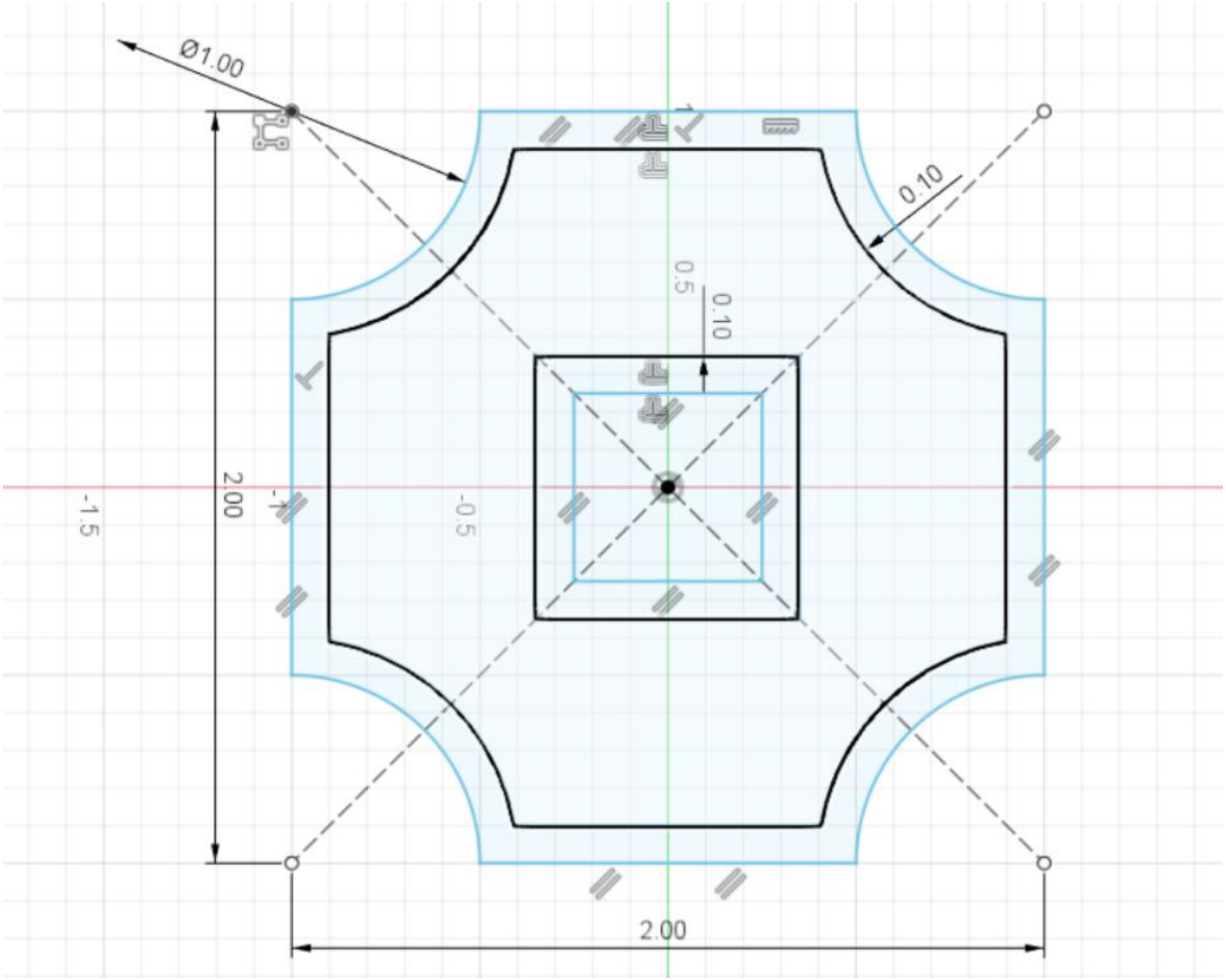
You may use whatever method you prefer to make the following sketch but i would recommend Using a mix of offsets, patterns and the circumscribed polygon tool to create the sketch



Use the trim tool to remove excess lines



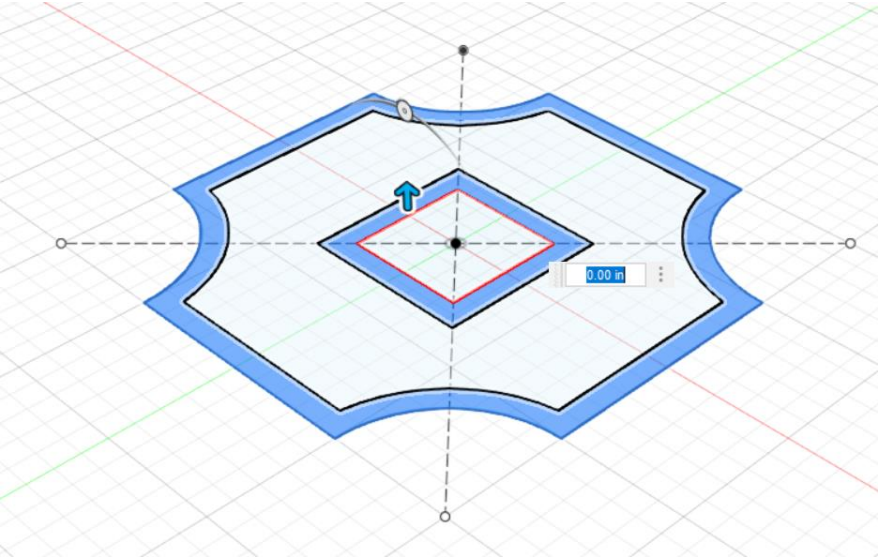
Create a .1 inch offset (the hotkey is o) of the outer perimeter of the coin as well as the square



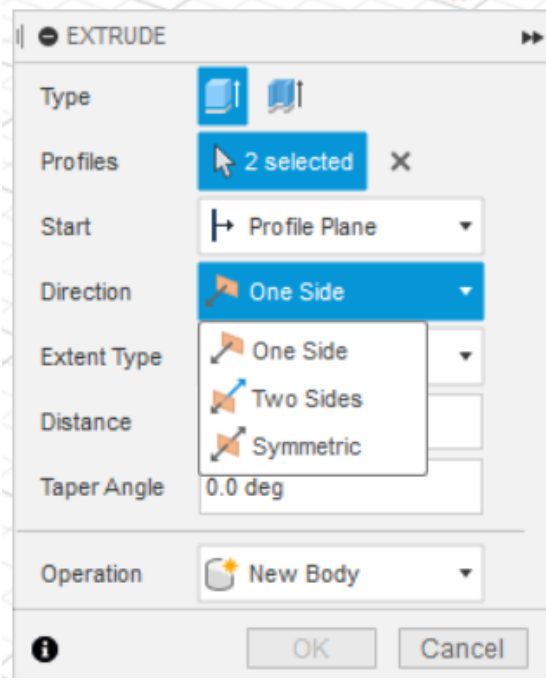
Select the extrude tool from the Toolbar and select the profiles shown in the picture A.

A box will appear on screen names “EXTRUDE”, this box contains different settings that can be used with the extrude tool

Go to the “Direction” setting and select symmetric. This will create an extrude going in both directions from the extrude profile.



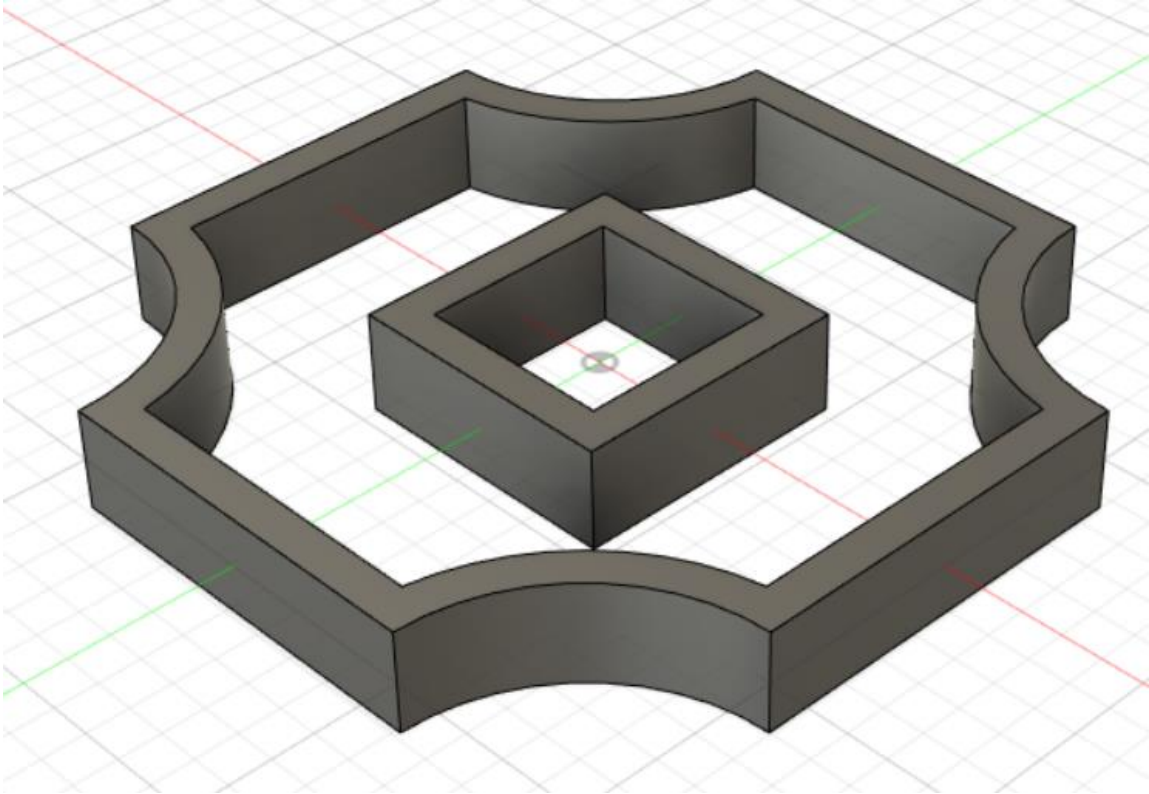
Picture A



We want the coins to be around .25 inches tall, so in order to get the correct height we will extrude half that, .125. (This is because both sides of the sketch profile are being extruded in opposite directions)

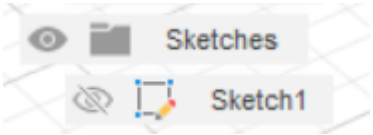
You should end up with the two objects shown here

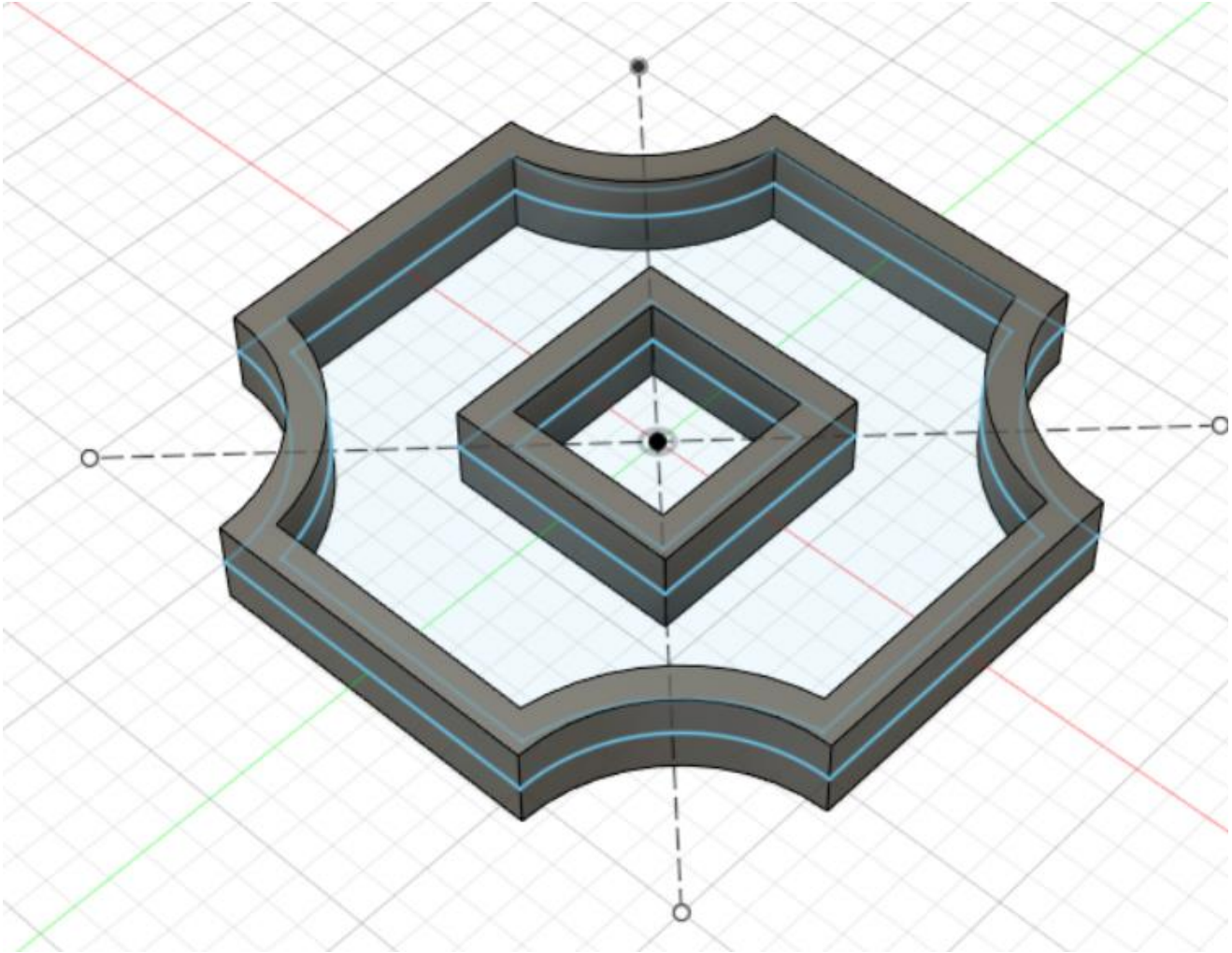
We’re going to do another extrude in the next slide



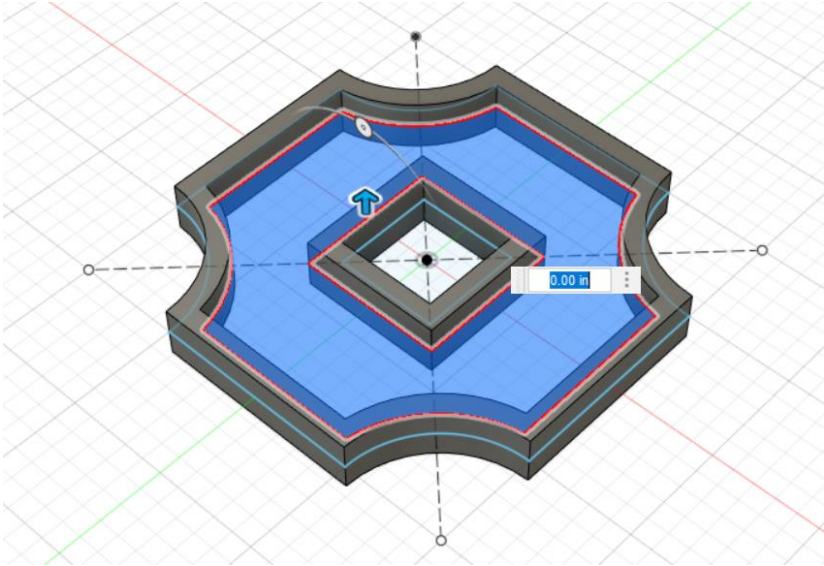
You may have noticed that the sketch you made is now hidden. This is because once a sketch is used by a tool, such as Extrude, is will automatically hide itself to avoid clutter.

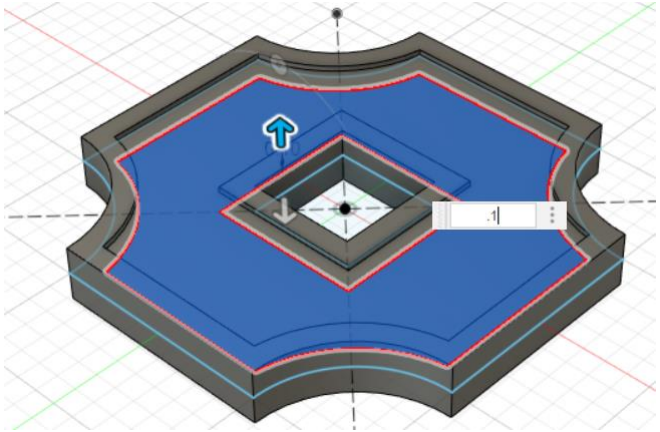
To once again see the sketch simply click on the crossed out eye next to “sketch one” as shown below





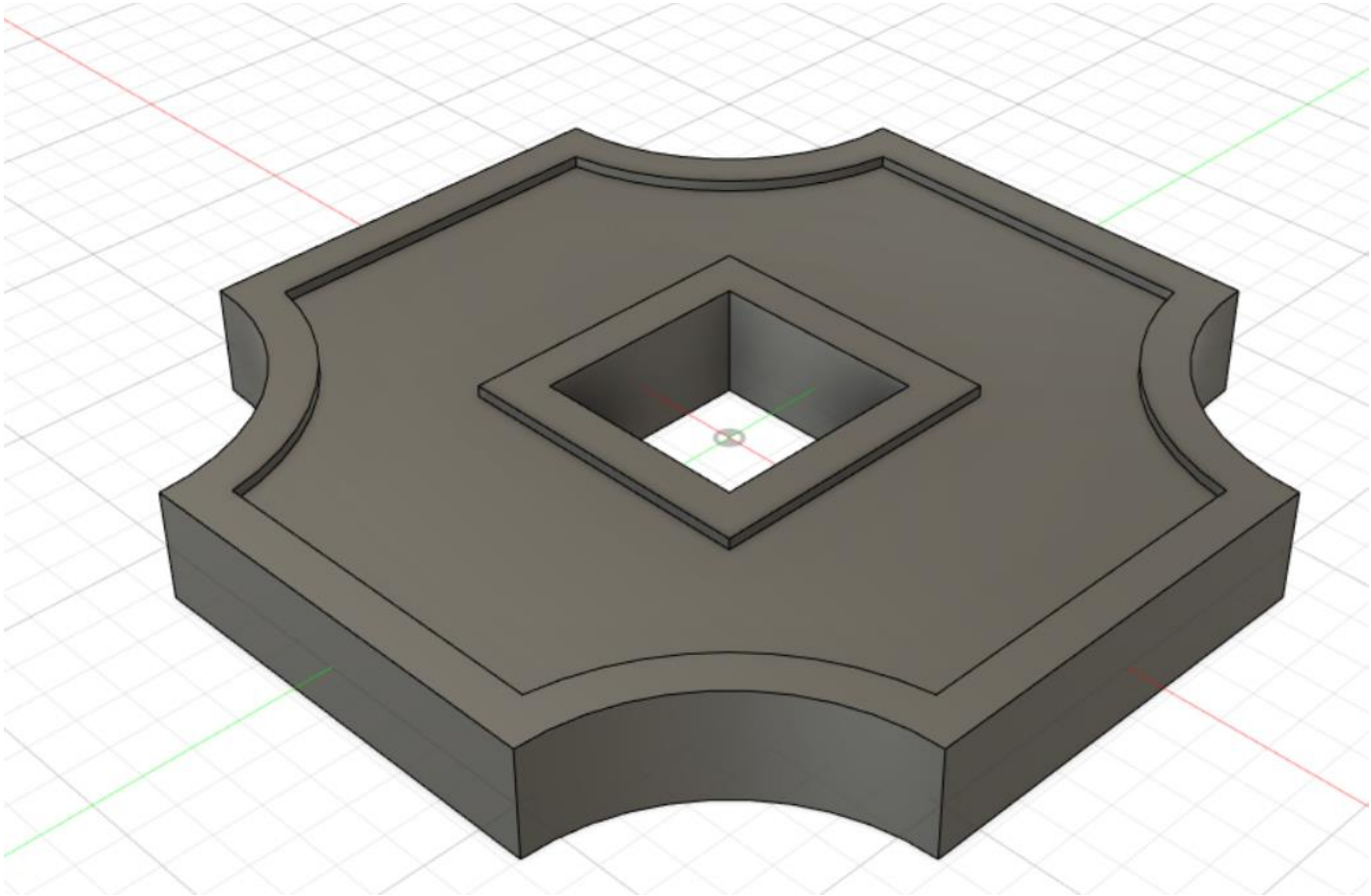
Were going to follow the same steps we just did but instead were going to select the inner portion of the sketch profile (as seen in the picture to the right)



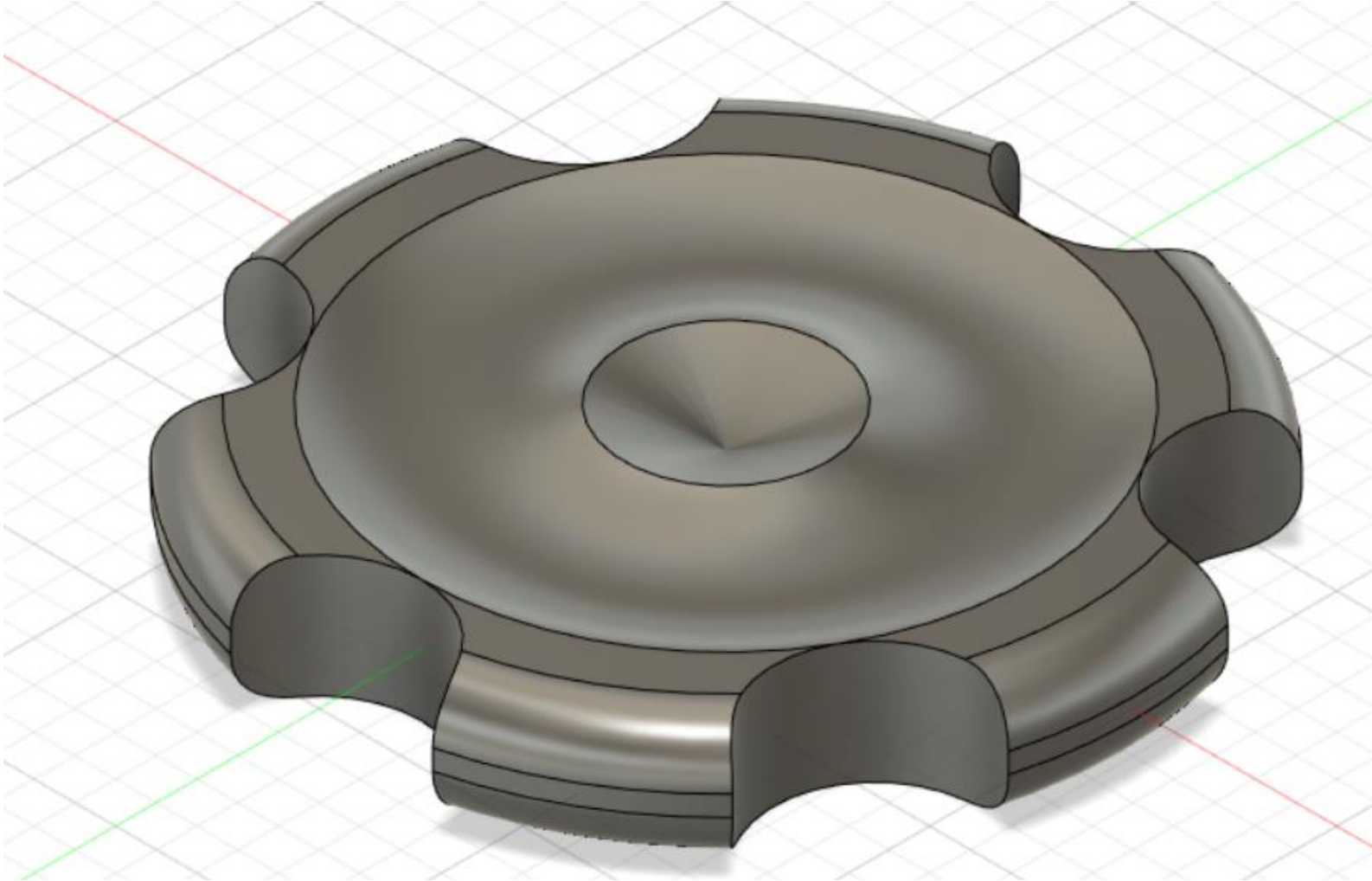


Once selected we will extrude this profile by .1 inches (make sure to have selected the “symmetrical” setting in the extrude box)

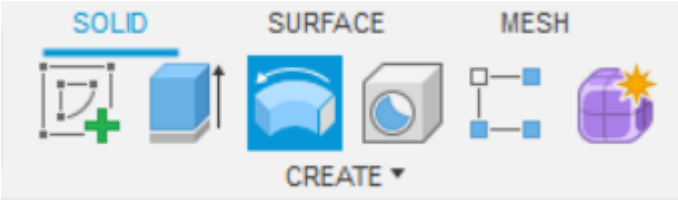
Congratulations on completing your first coin



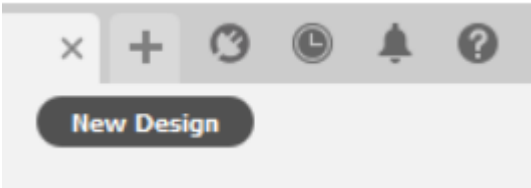
Coin 2



Coin 2 will mainly taking advantage of the “Revolve” tool in order to create the main body of the coin.

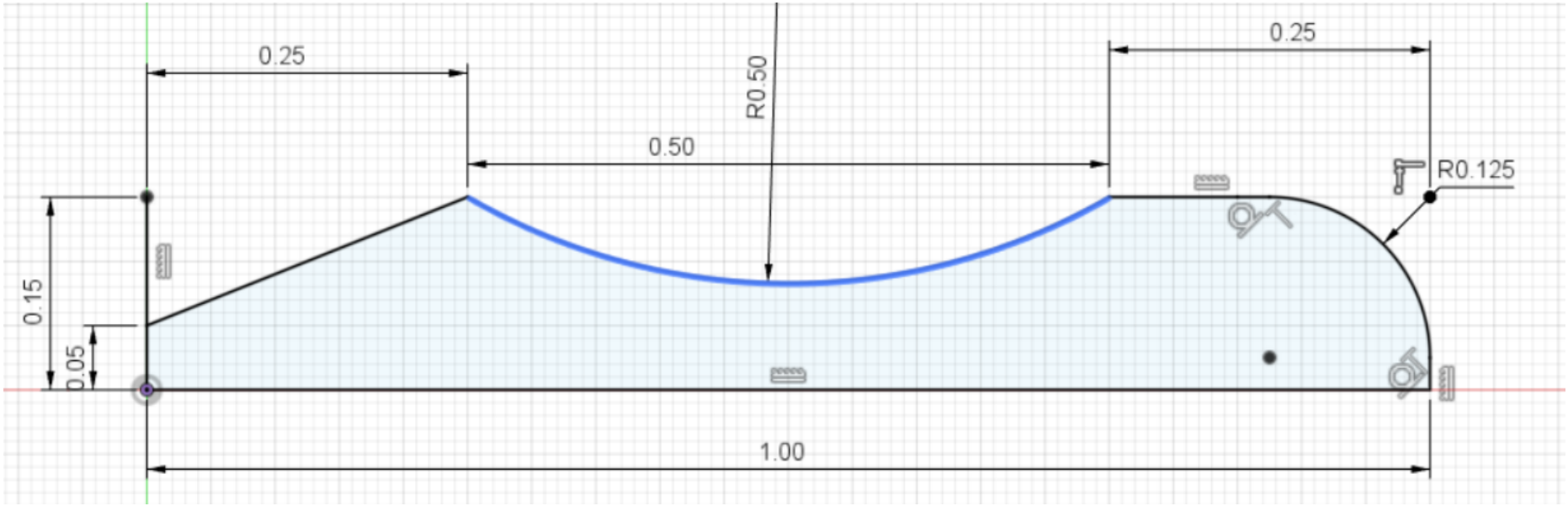


Make sure that you start a new design to create this next coin



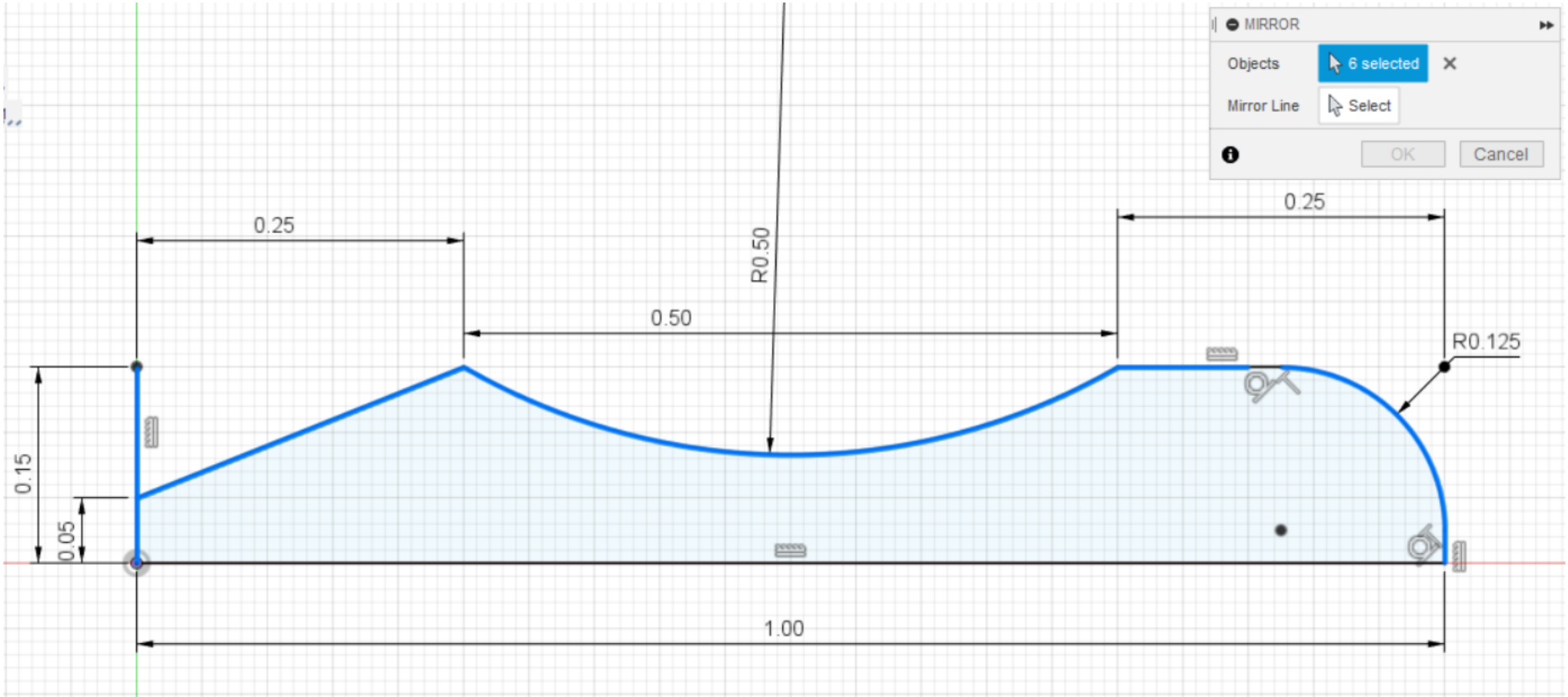
For this sketch we will be drawing it on the front plane

To sketch this i used a mix of arcs, fillits, and trims.
I started with a rectangle for the basic shape



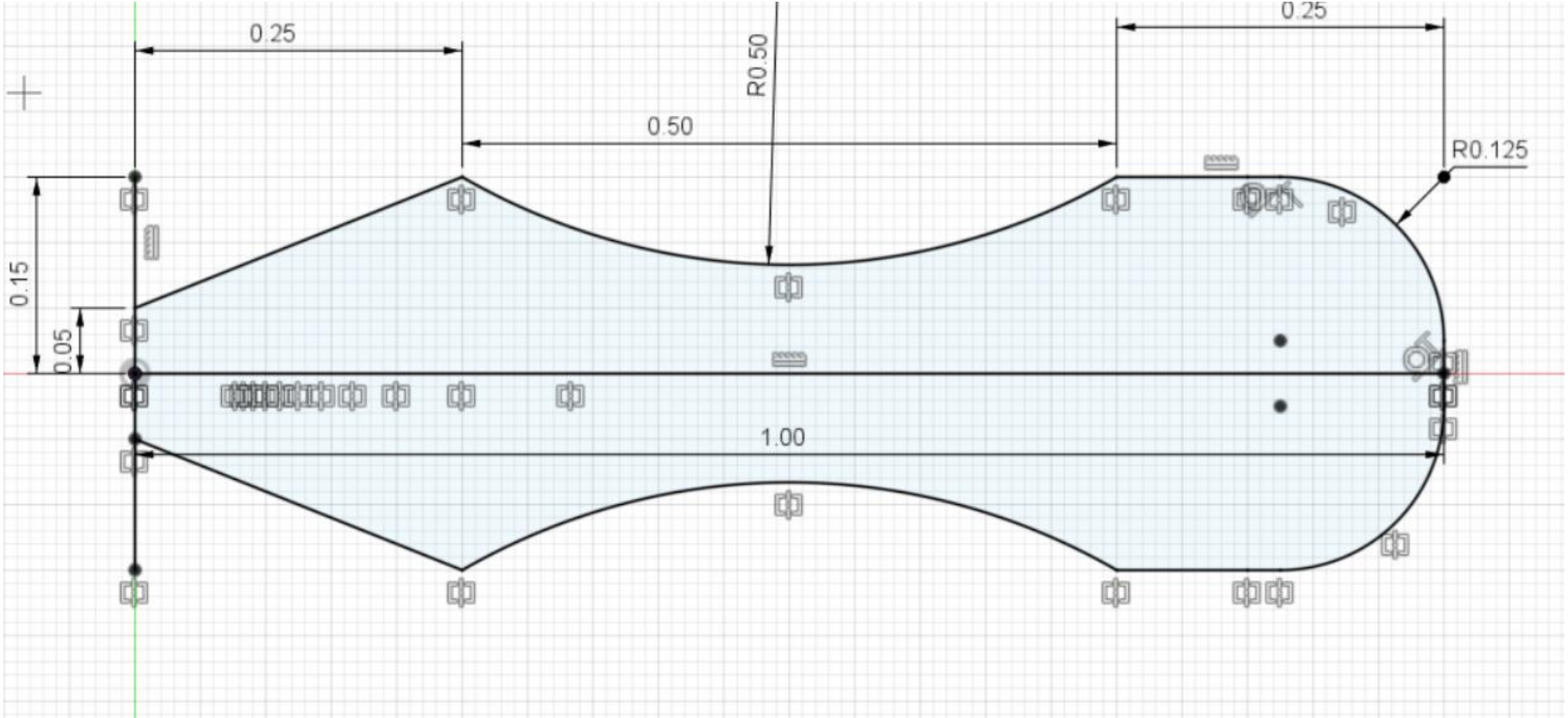
From here we will mirror the geometry across the bottom line of the sketch.

When the mirror toolbox appears, select the lines you want to mirror (seen in blue)
Once selected click the “mirror line ” setting and select the bottom most line (seen in black)



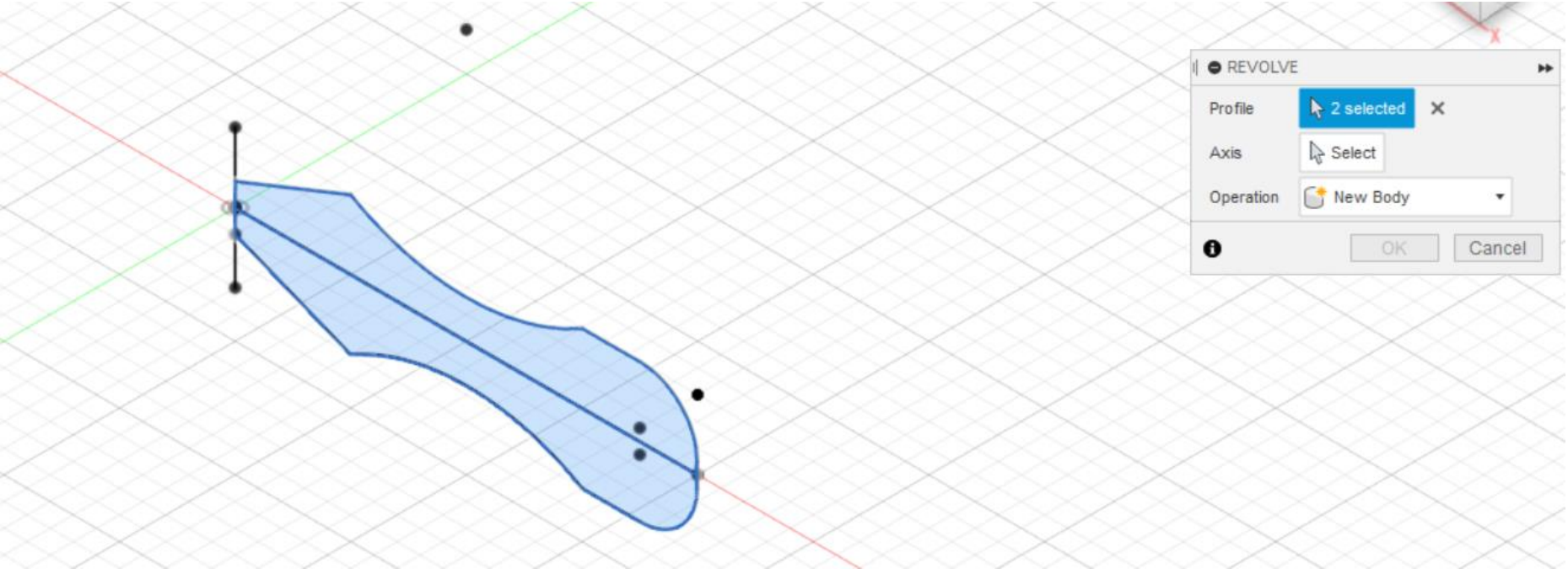
Click “OK” and your sketch should end up like this

From here you are free to finish the sketch

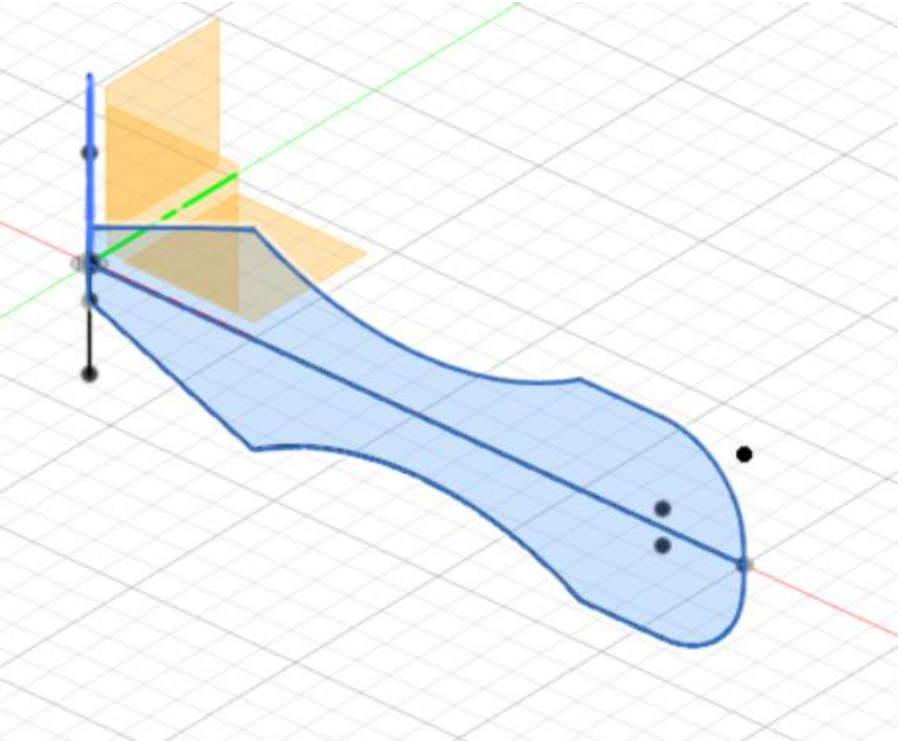


Once your sketch is finished, choose the “Revolve” tool from the Toolbar.

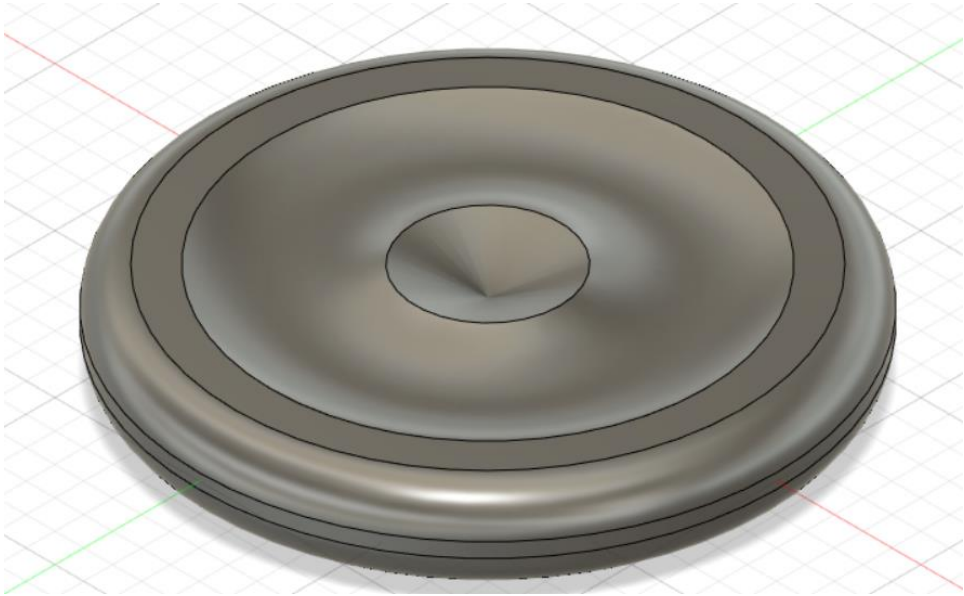
Select the 2 sketch profiles shown



Select the “Axis” setting and the origin will appear. Click on the blue vertical line and then click OK



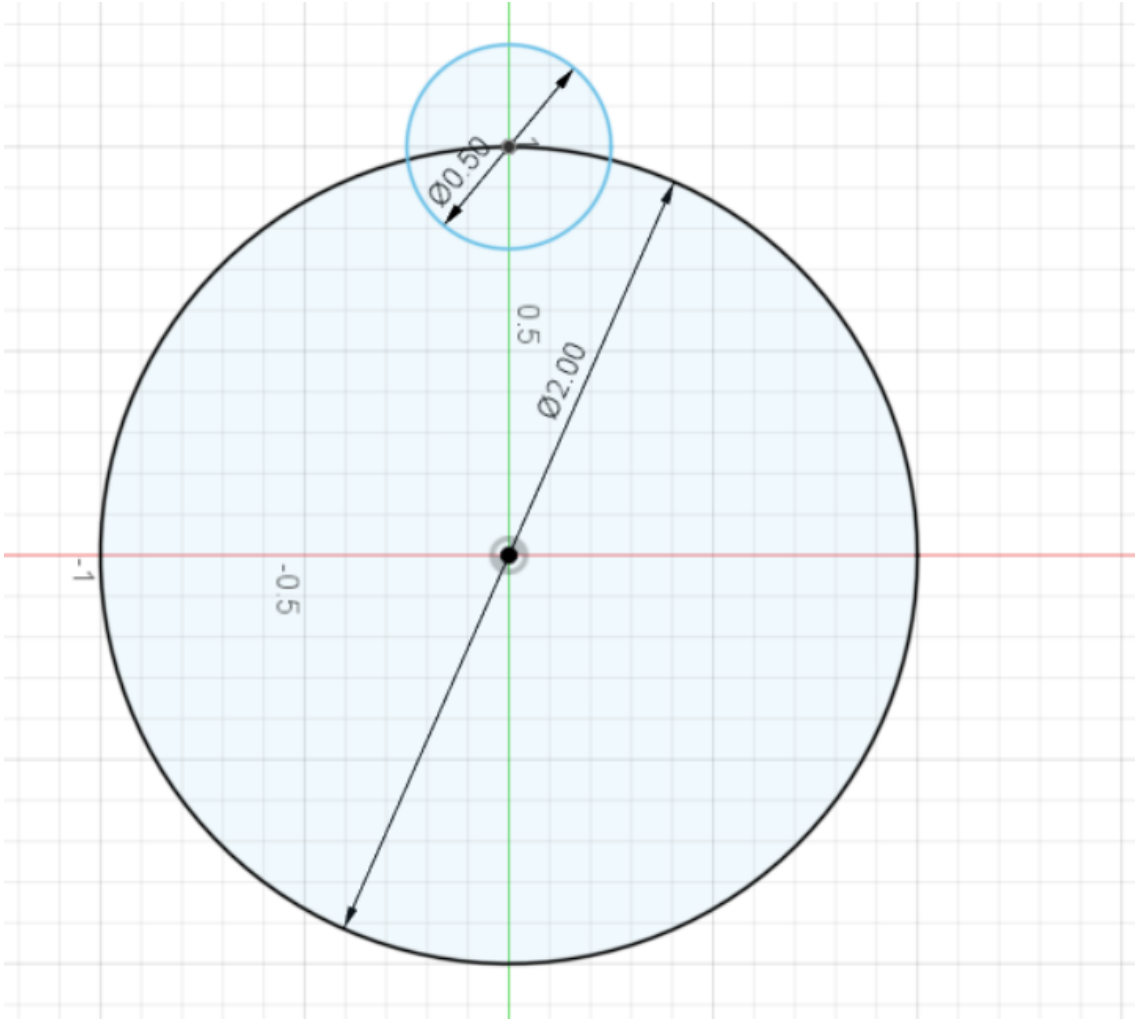
You should end up with this shape



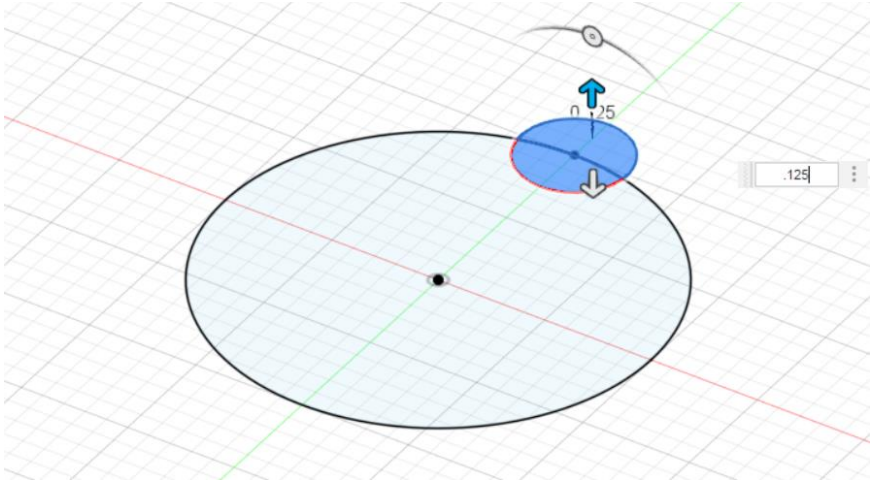
For now were going to hide the body and use it later

For now we will want to create a sketch on the TOP plane that looks like this

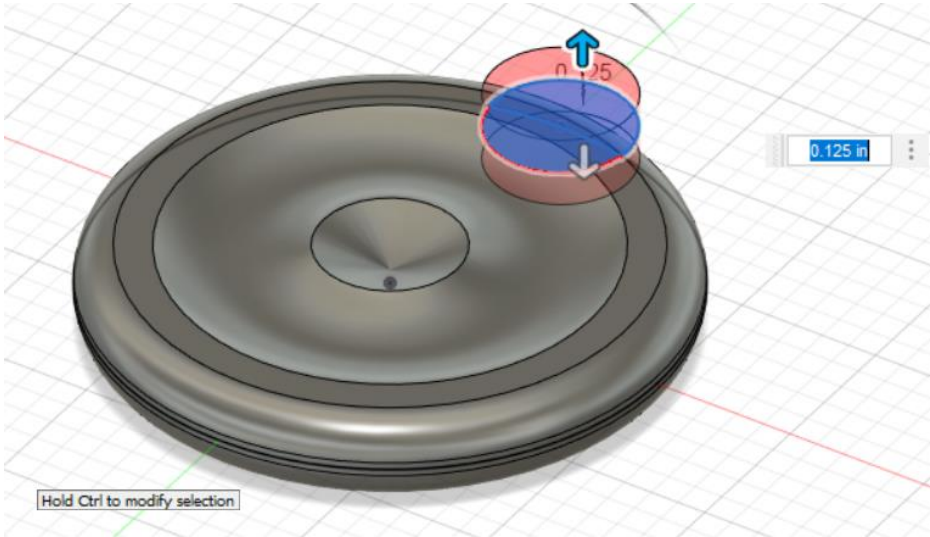
Finish the sketch and select the extrude tool. Select the topmost circle and extrude it (make sure to use the symmetrical setting) .25 inches. However we dont want to click OK just yet



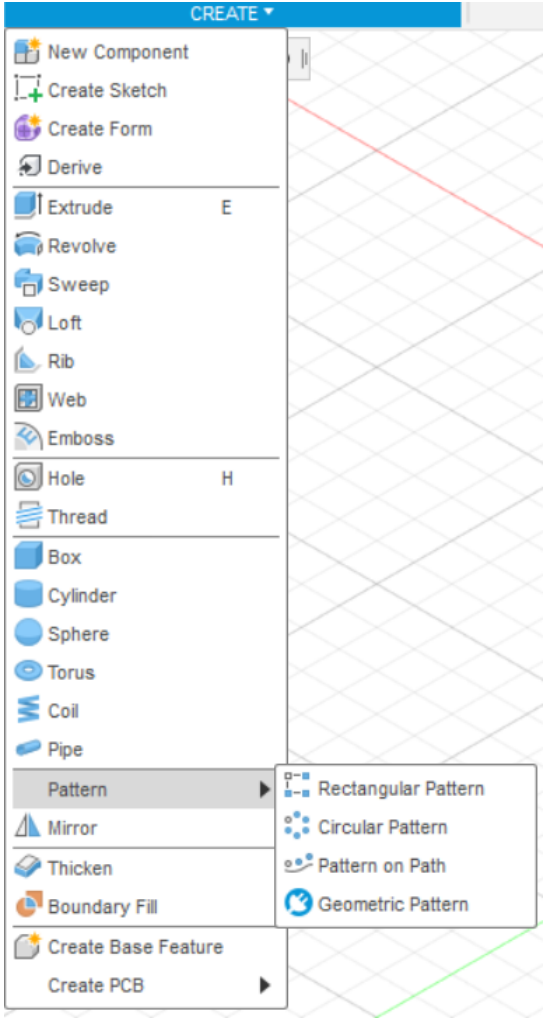
So far you should have something like this.



Reveal the main Body and you should have something like this. Once you're here you can finish the extrude

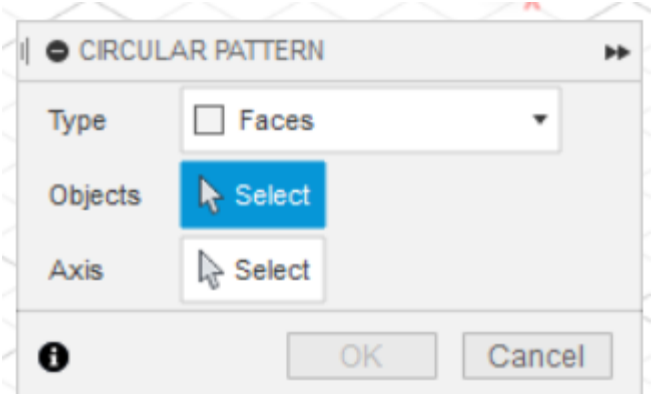


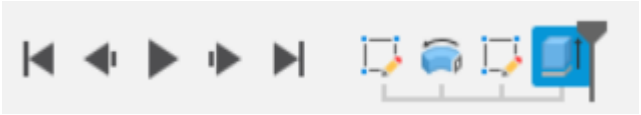
Next click the create dropdown button and navigate to “pattern”
From here select “circular” pattern



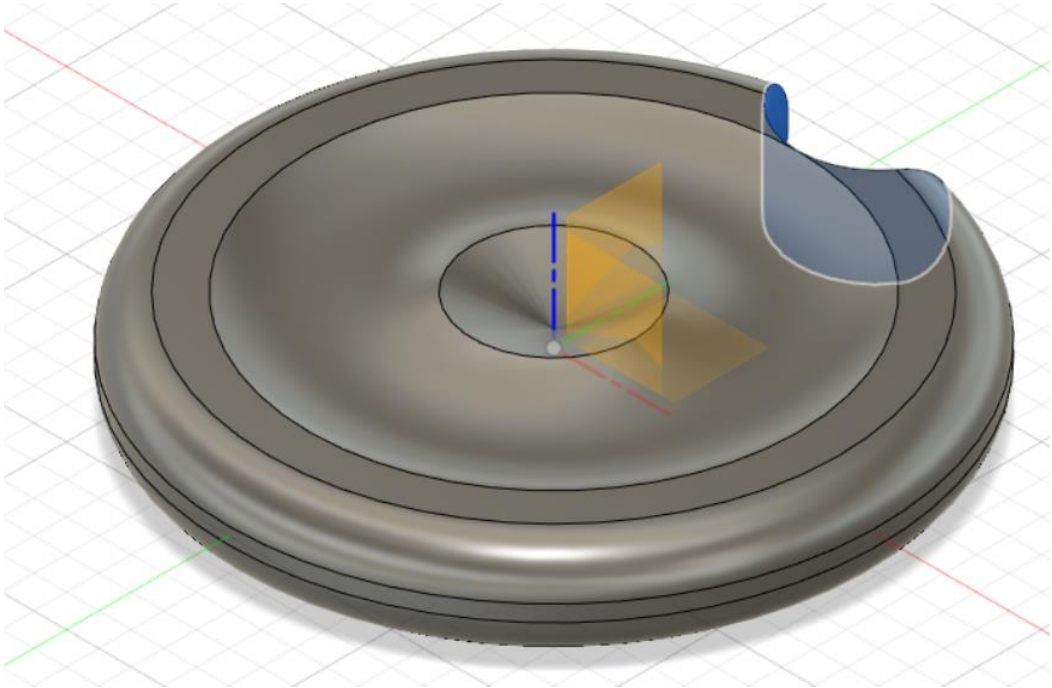
The object that you are going to want to use for this pattern is the last extrude we did to cut out a semicircle in our coin.

You can find this in the Timeline at the bottom of the screen (it should be the last thing on your timeline)

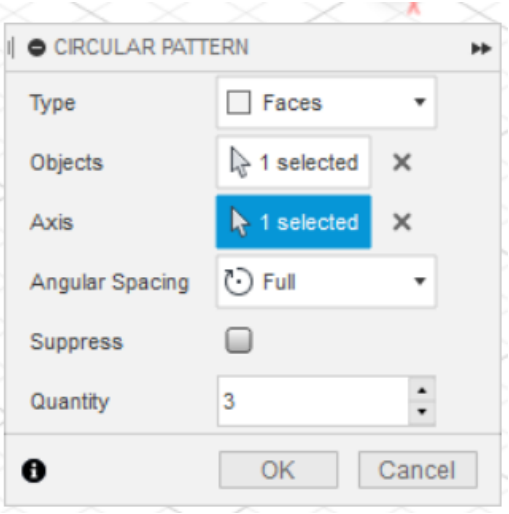


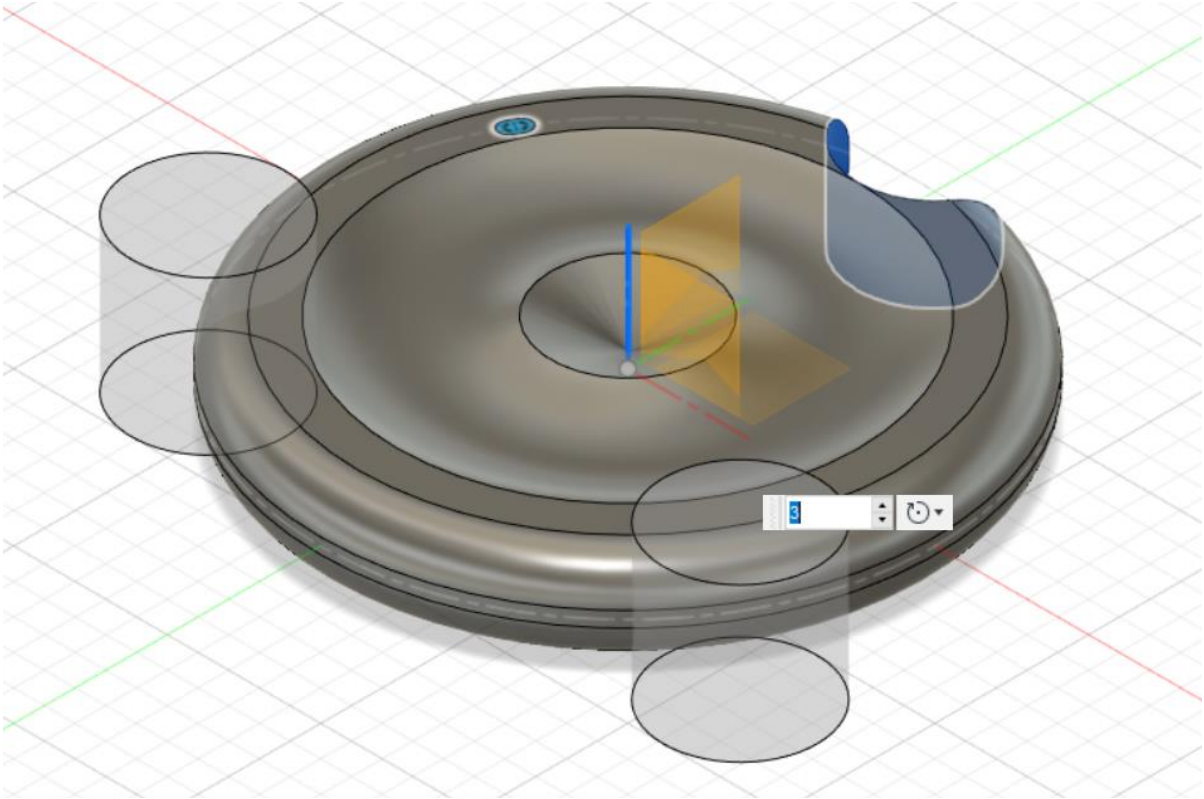


Similar to the “revolve” tool, once you select the “Axis” setting you will want to click on the blue vertical line

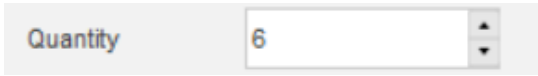


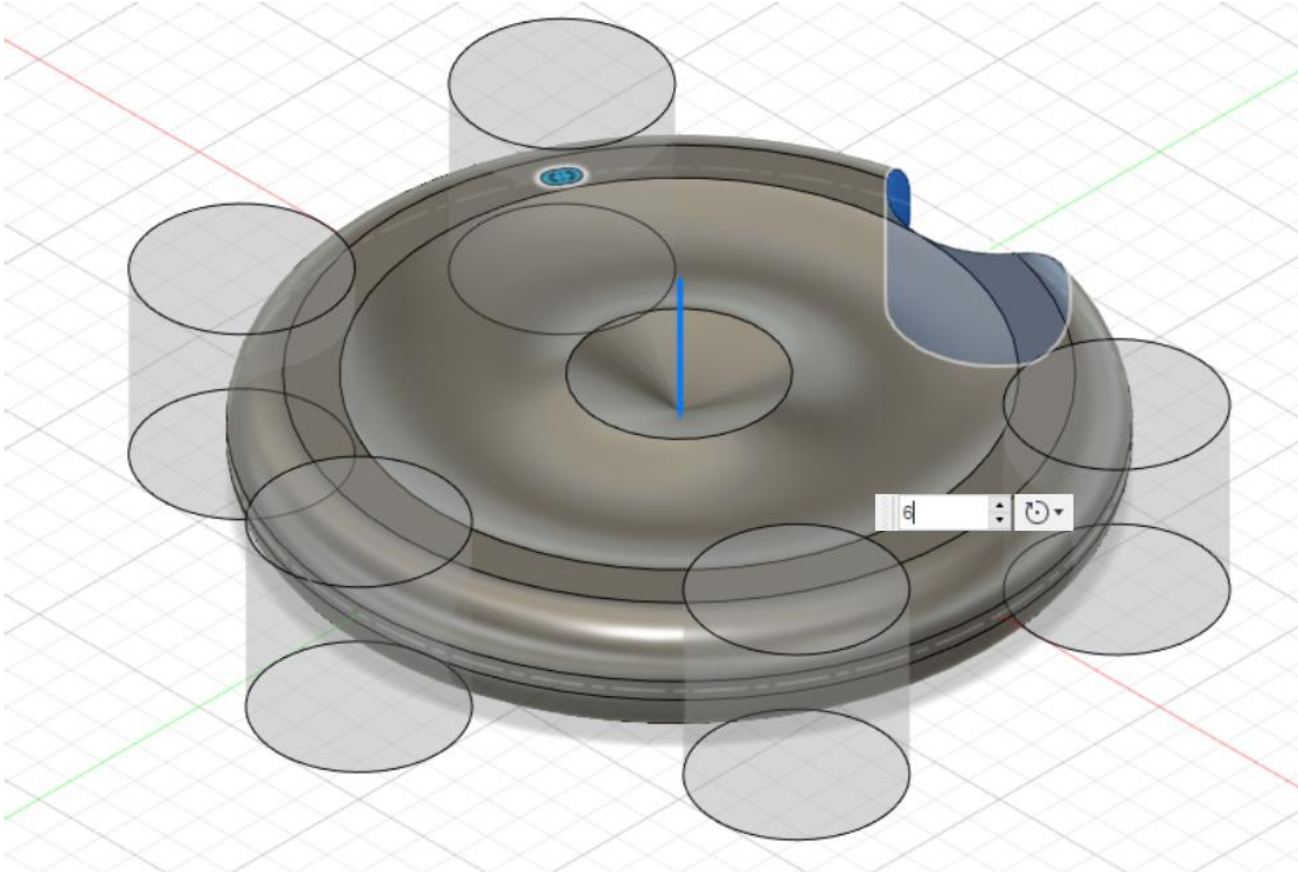
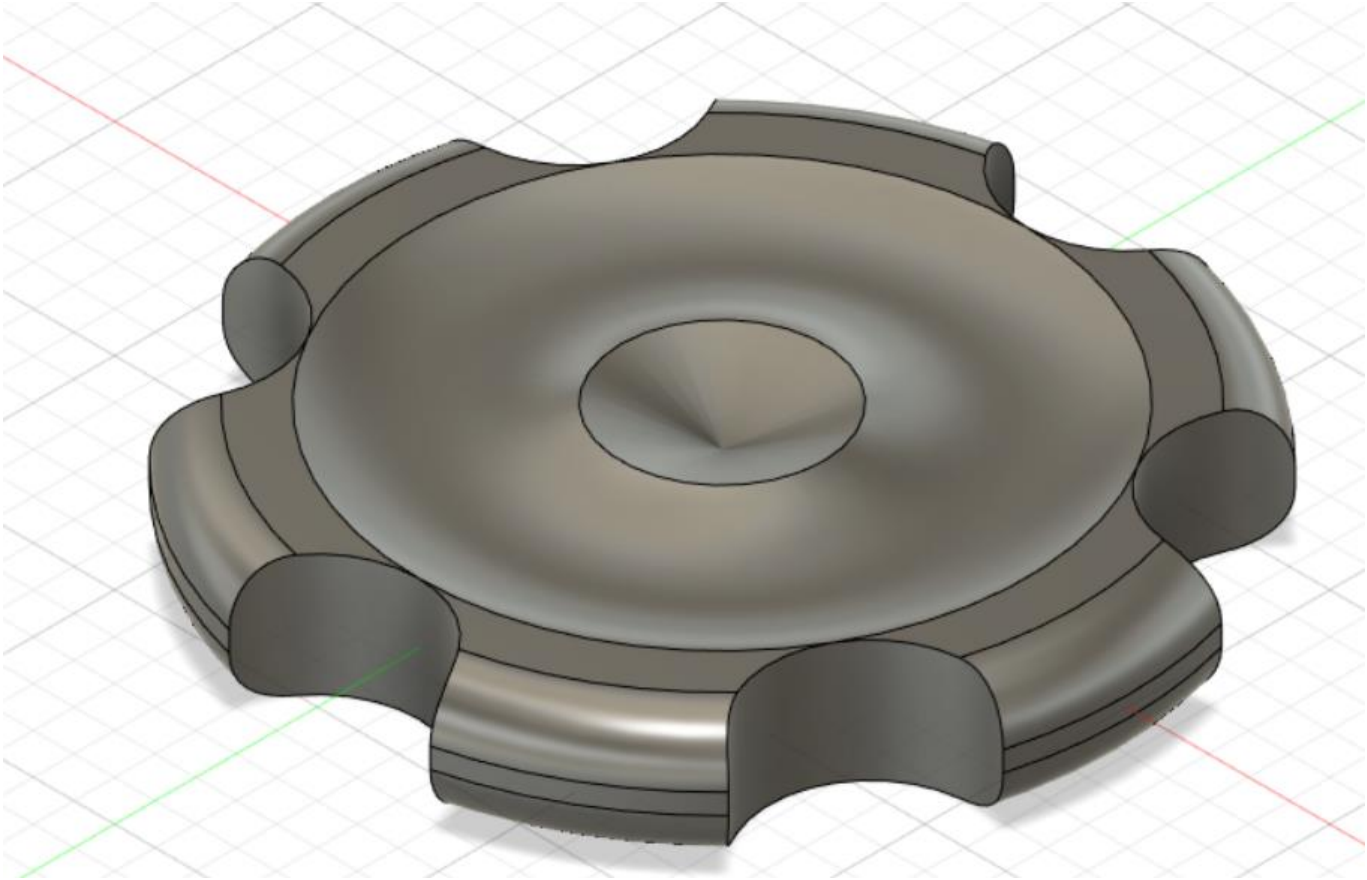
You should see 2 more cylinders appear around the coin, as well as the “Circular Pattern ” settings box





Change the number of cylinders from 3 to 6 and click OK.





Congratulations on finishing the second coin