Enlarge these holes.

Make a round sandpaper file.

Today's Lesson is Sponsored by Raytheon

Contents

# Changing the View of a Design

- if you don't see a grid in the Fusion 360 window, as shown below, click on **Grid and Snaps** and check **Layout Grid**. Displaying the *Layout Grid* is a matter of preference. When designing for 3D printing, it can be used to represent the *build plate*.

- click on the **Orbit** tool and click somewhere on the **Grid** to practice rotating and changing the angle of the view.
- click on the **Pan** tool and then on the **Grid** to practice moving the view laterally.
- after using the *Orbit* or *Pan* tool one must press the **Esc key** to exit that mode.
- use the **Mouse Wheel** to practice Zooming in and out.
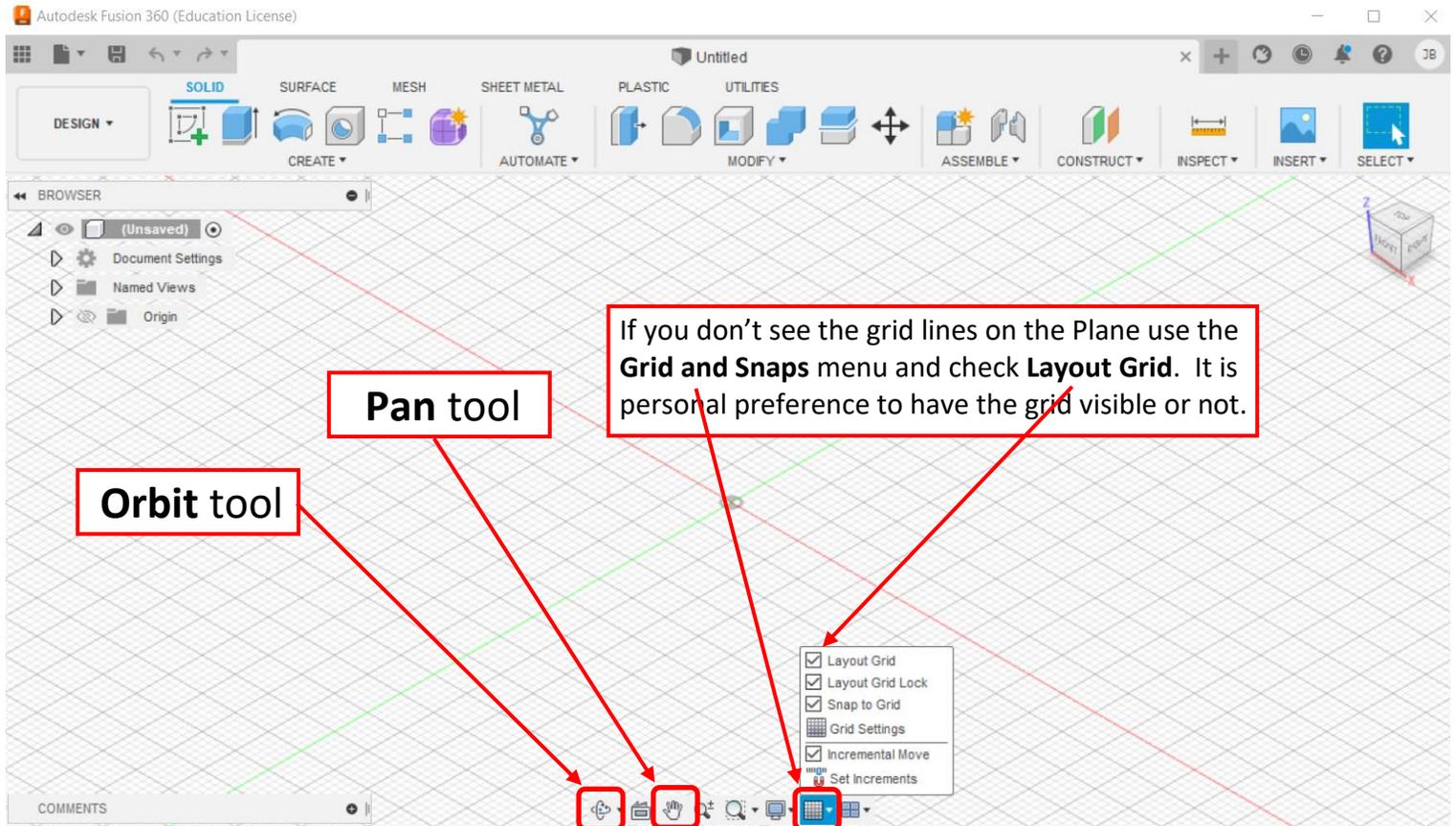
Here is a close-up of the View Cube at the top right of the window.

- click on the **View Cube** and move the cube while holding the mouse button down. This is another way to rotate the view.
- click on the Top of the View Cube and note how the view just jumped to a Top View.

The View Cube now resembles that on the right.

- click on the **Curved Arrows** at the upper right of the View Cube and practice Rotating the View.
- click on the **Arrows** at the sides of the View Cube to practice jumping to various Views.
- click on the **Home** icon to the upper left of the View Cube. This can always be used to reset the view to the Home View
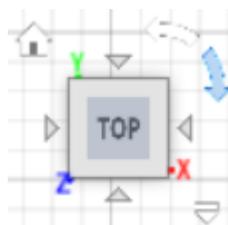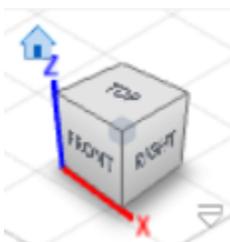
- open **Fusion**. If there is no icon on the Desktop, use the Windows search (magnifying glass icon) and type fusion
- from top **File** icon select **Save** and name the file.
   Use your name followed by **_RoundFile** e.g. **JoeBarbetta_RoundFile** (note the use of the underscore)

Note that by default Fusion saves your project to "the cloud", which are the servers managed by AutoDesk. When you log into Fusion on a different computer, your projects will be available.

As you work you may want to occasionally save your work in case Fusion crashes or we lose power.

- in the left **"BROWSER"** click the **arrow next to Document Settings**
- click on the **edit icon** that appears to the right when you hover over **Units**
- ensure **Active Units** are set to **Units: in, ouncemass** and click **OK**. You can also enable **Set as Default** if it is not grayed out.
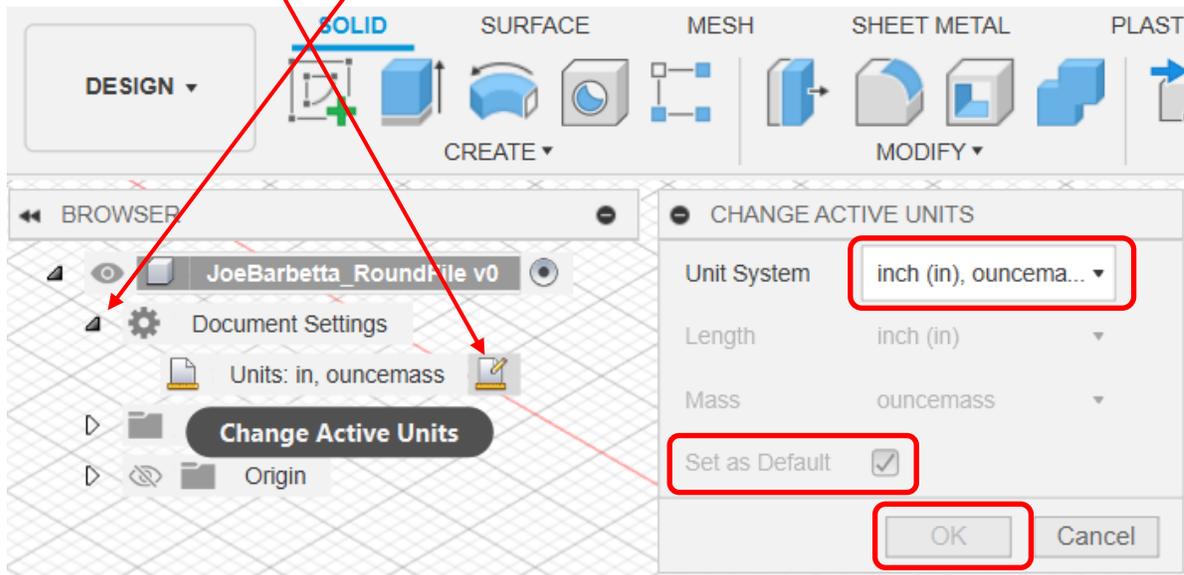
Note that the default units are in mm, which we just changed to inches.

Did you know that the default units have changed over the years? The earliest version used cubits as the default unit.

Creating the First Sketch

Note that a Fusion expert may tell you to create a Component first. Just say **"Dude. I'm just making a file."**

- select the top **Create Sketch** tool and click on the **front rhombus** to select the X-Y Plane.
If a tool can't be found, one can always look in the **CREATE** and **MODIFY** menus for it.
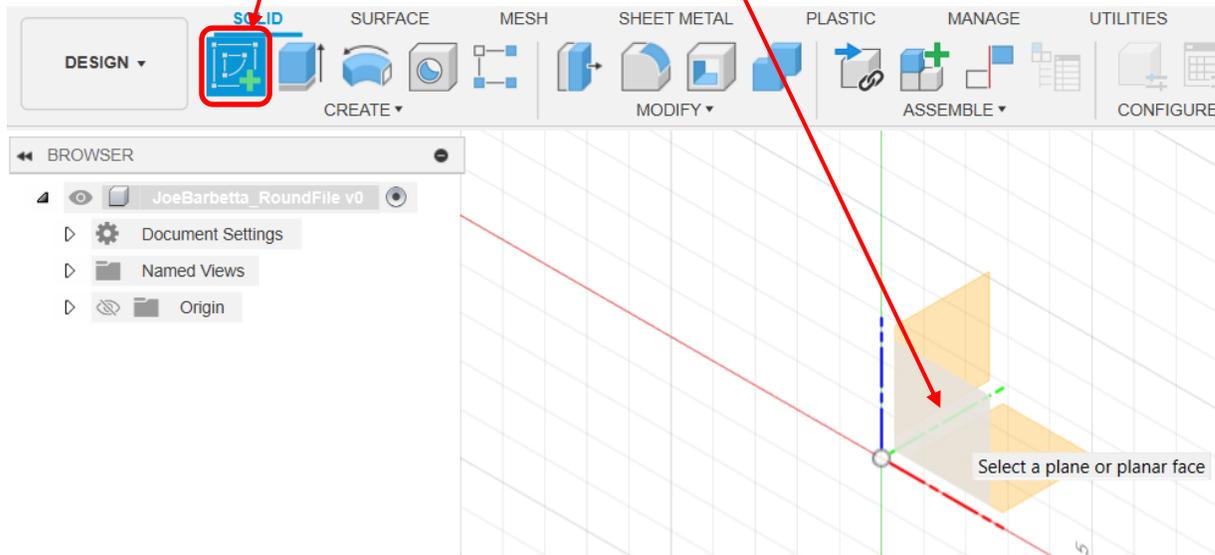
- zoom in as shown below. The scale labels can give an idea of how far one is zoomed in. The **View Cube** should indicate you are sketching on the **FRONT X-Z Plane**.
- click on the **arrow** next to the Sketches folder to open it
- right click on the **Sketch name**, select **Rename** from the menu and rename the Sketch to **Profile**
- click on the **double-arrows** for the BROWSER to hide it to make more room for your sketch
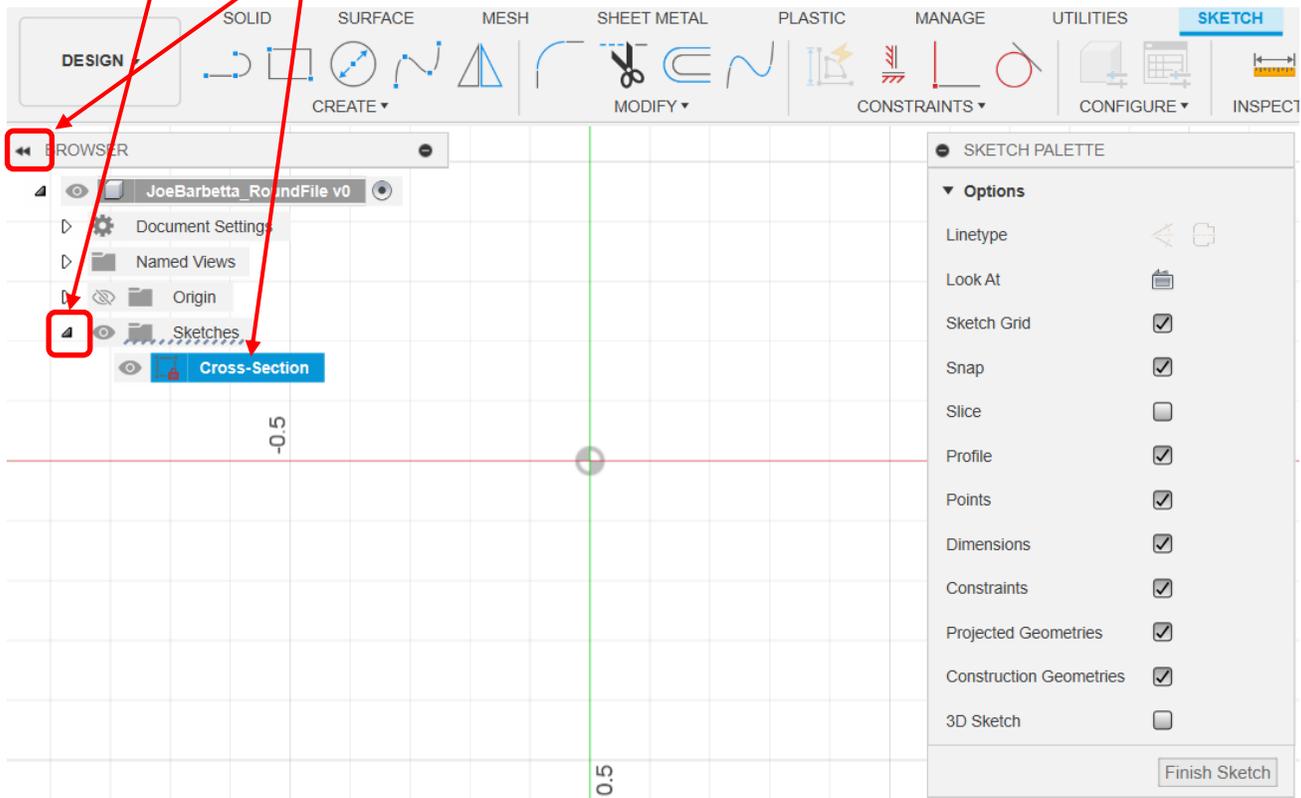
- from the **MODIFY** menu select **Change Parameters**
- if a Parametric Text message window appears, click its **OK** button
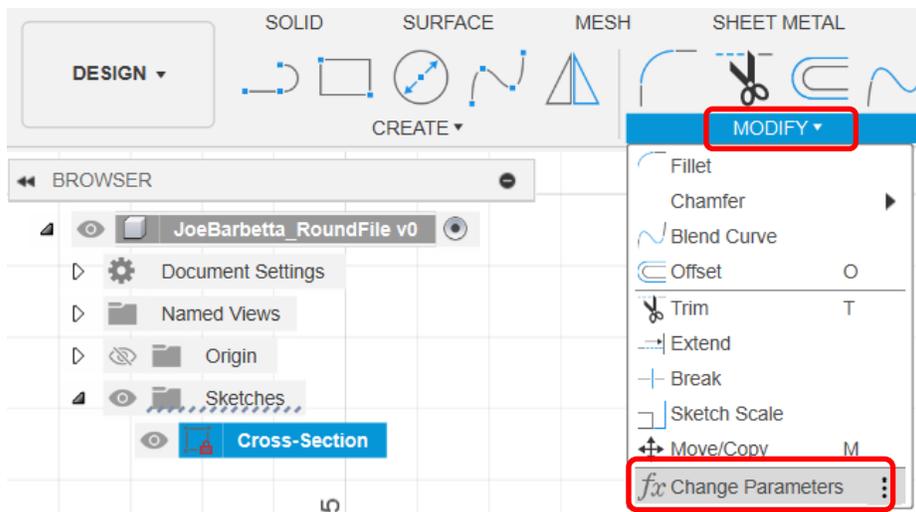
- click on the top **+** icon
- if a Parametric Text message window appears, click its **OK** button
- set the **Name** to **Diameter**, the **Expression** to **0.47**, and click **OK**
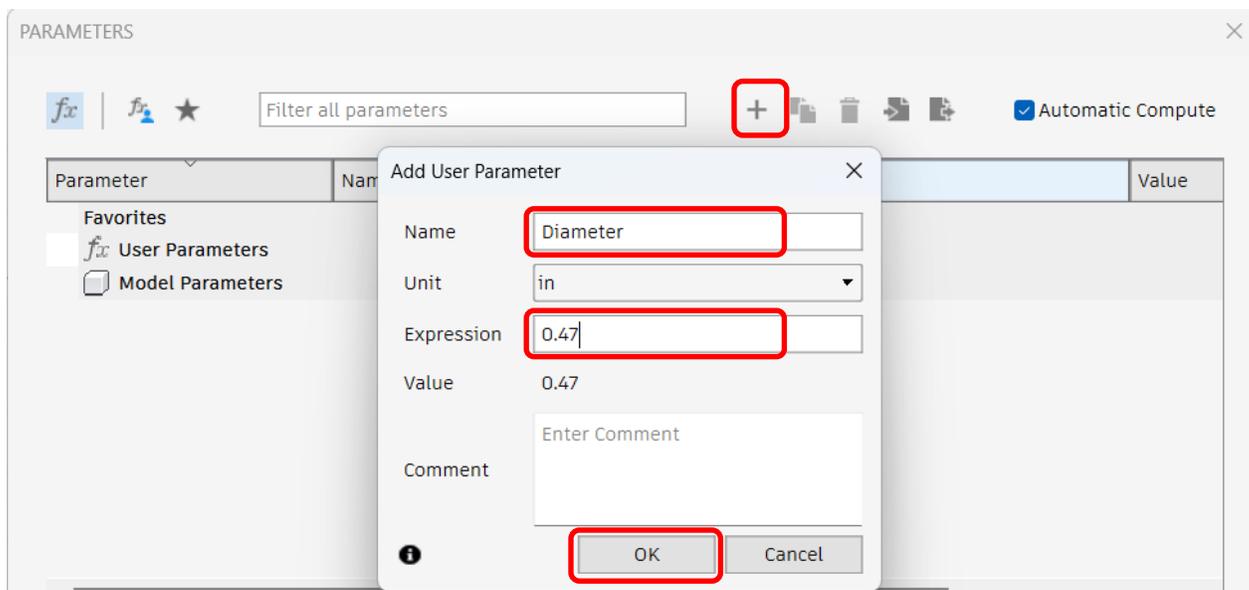
- perform the same steps to add: **SlotWidth 0.47**   and **SlotClearance 0.015** as shown below
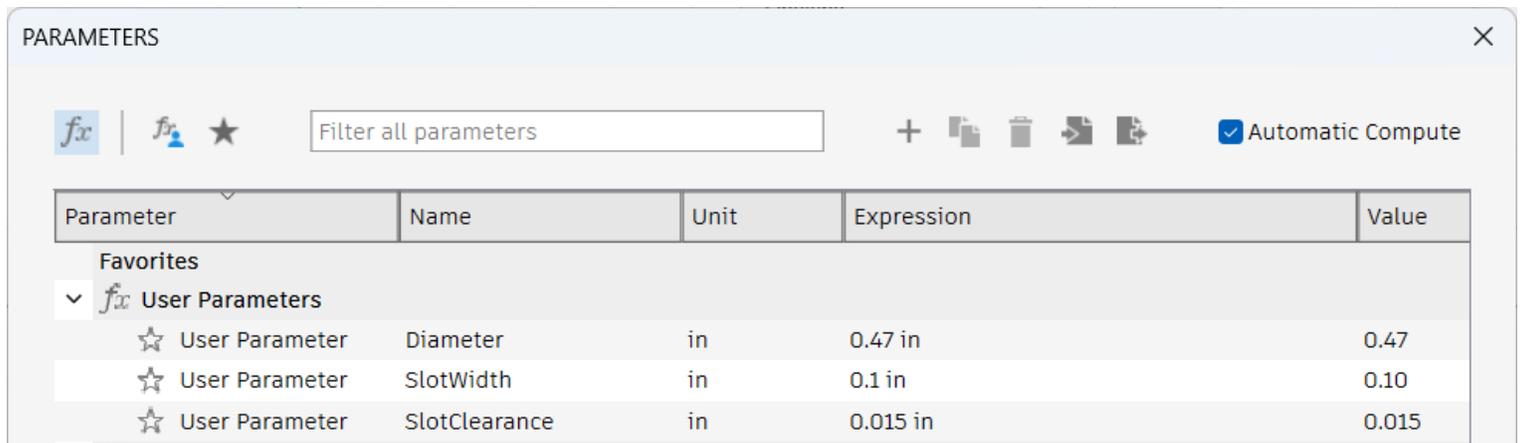
- click on the **+** icon again
- set the **Name** to **SlotBottomPos**
- in the **Expression** box type **d** and select **Diameter 0.46 in** and then **/ 4** , as shown below, and click **OK**
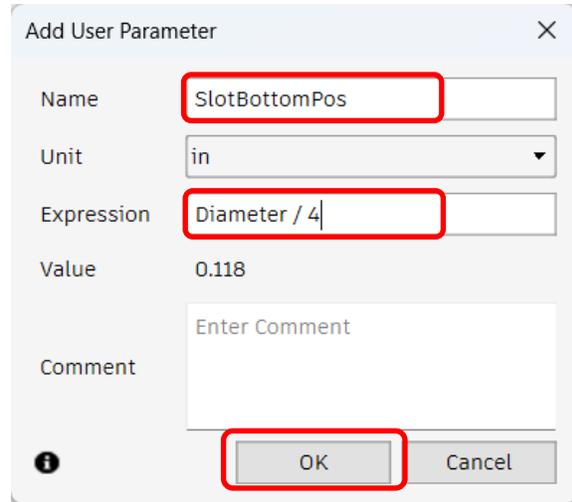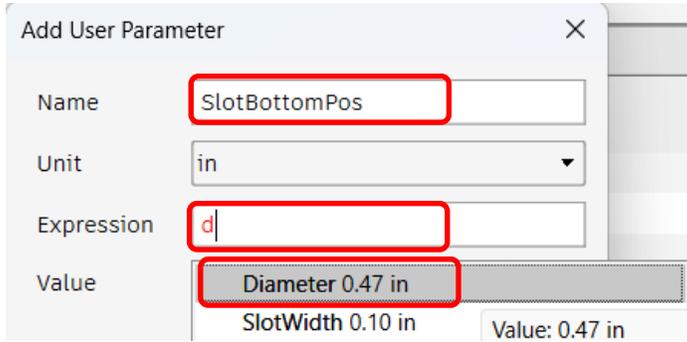
- use the **+** icon again and first grab the right side of the window to widen it
- set **Name** to **SlotDepth** and then enter the formula as shown below. When **d** is typed **Diameter** can be selected and when **s** is typed **SlotBottomPos** can be selected. Did you ever think you would be using trig in a parameter?
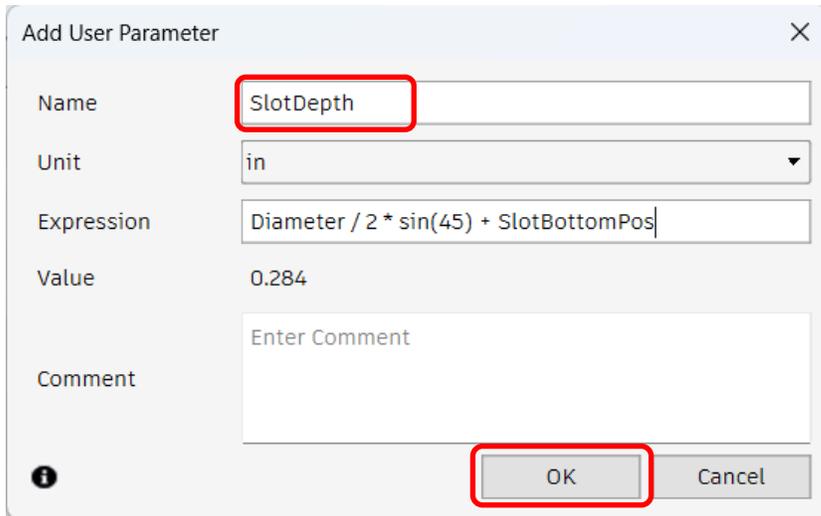
- use the **+** icon again and set **Name** to **TaperAngle**
- from the **Unit** drop-down menu, scroll down to **Angle** and select **deg**. Set Expression to **0.01** and click **OK**
It was found that if the Expression was set to 0, the Draft operation performed later will fail.
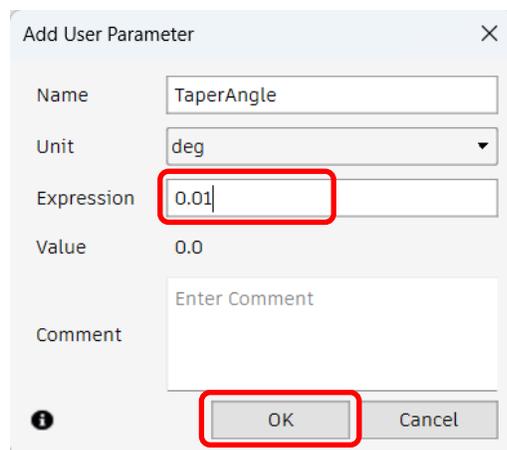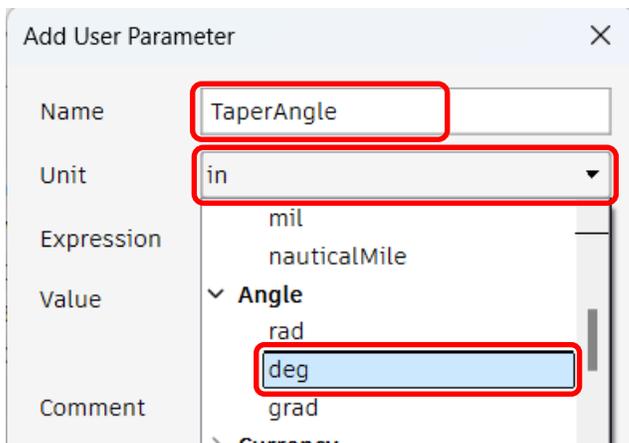
- verify the User Parameters, shown below and click **OK**

Note that one can select a row, and use the x before a Name to delete a parameter, clicking on a Name to change a name, or clicking on a Unit or Expression to make edits.

In the future, before printing a file, the Diameter, SlotWidth, SlotClearance, and TaperAngle, can be adjusted for the particular need.
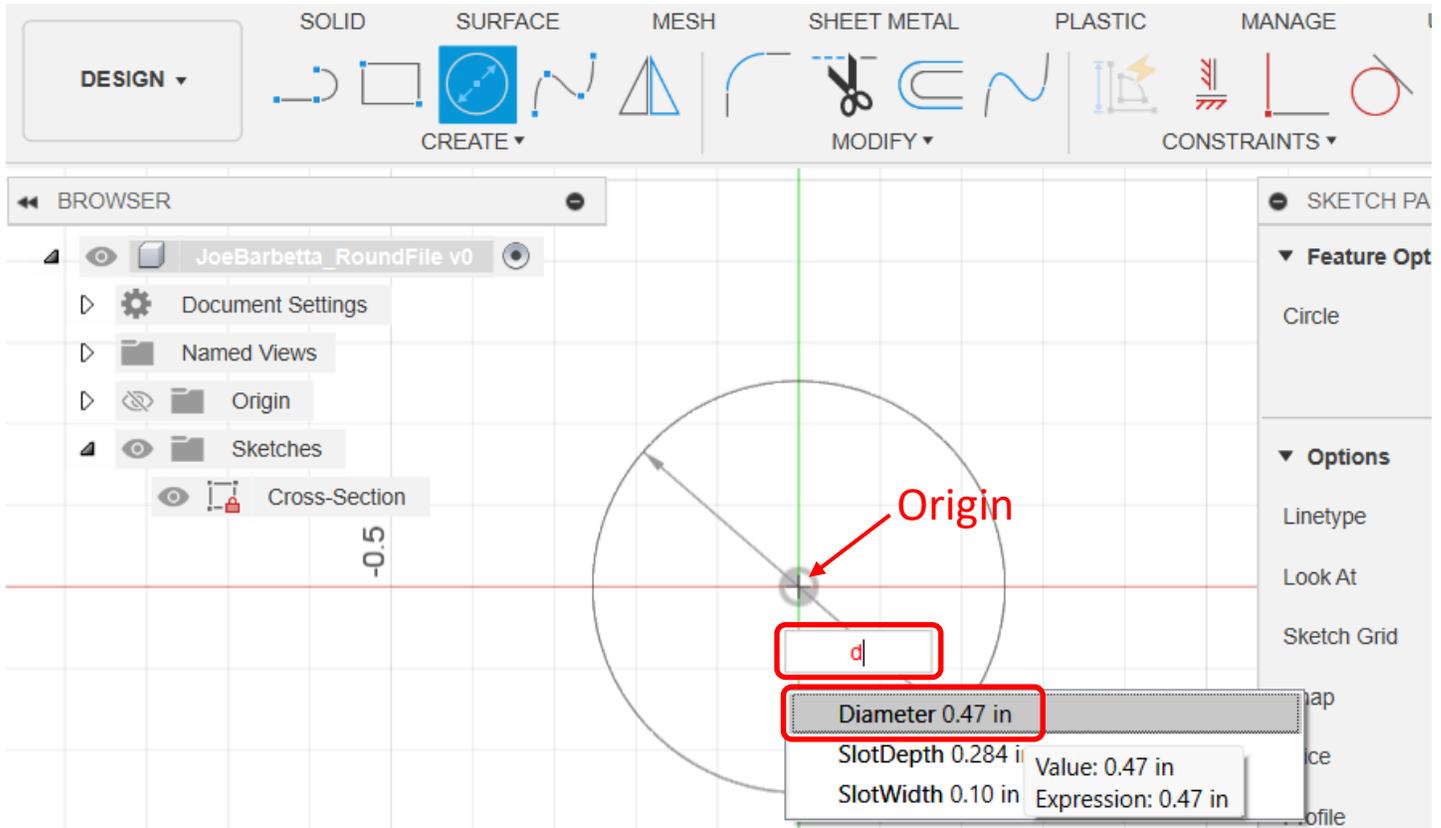
Creating the Cross-section Sketch

- select the **Center Diameter Circle** tool. If it is not visible, find it in the CREATE menu
- click on the **Origin**, extend the circle outward, type **d**, click on the **Diameter**, and then press the **Enter key**

- zoom in to the circle, as shown below
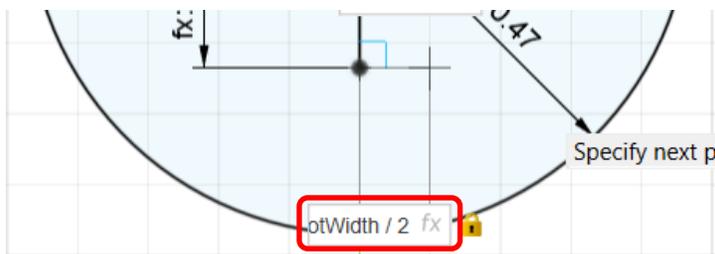- it is OK if the Diameter Dimension line looks different
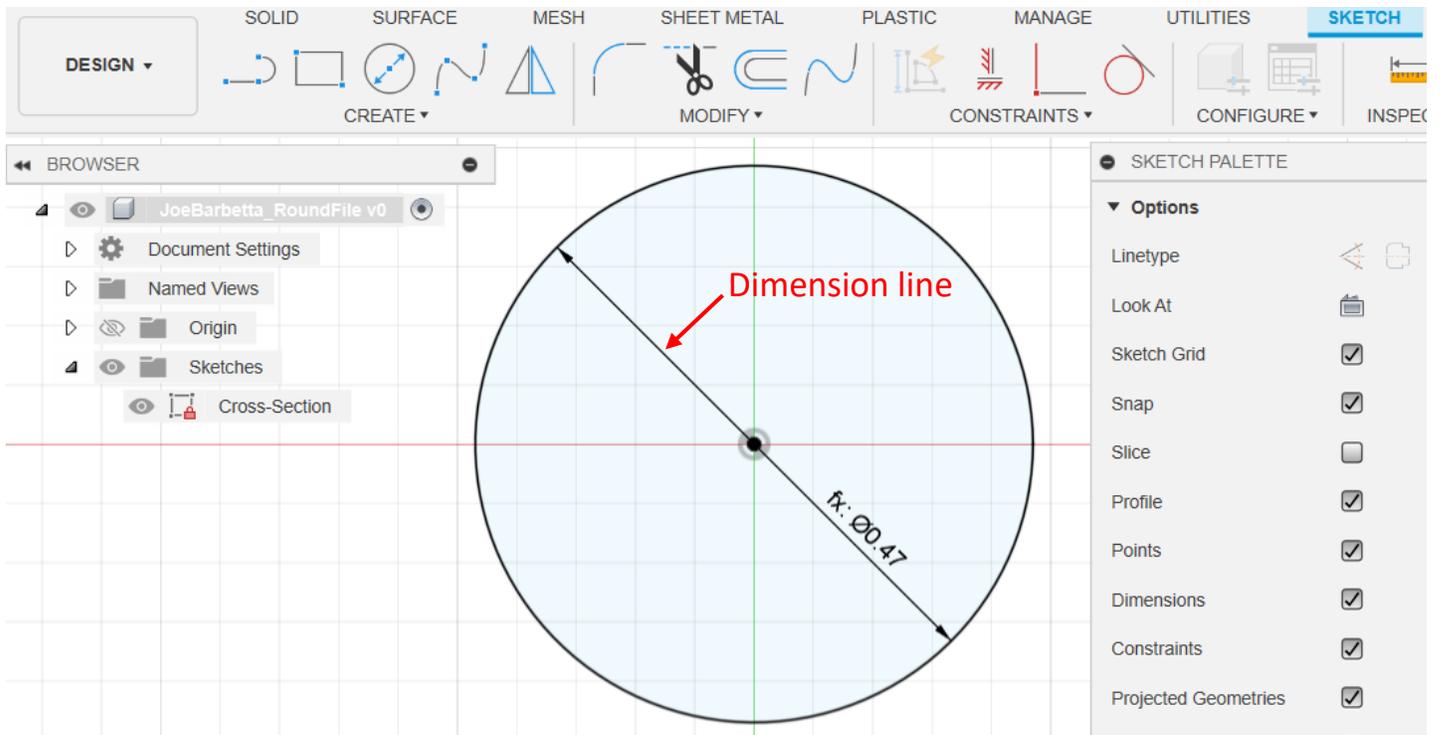
- select the **Line** tool
- click on the **Origin**, extend the line **downward**, type **s**, select **SlotBottomPos**, and press the **Enter key**
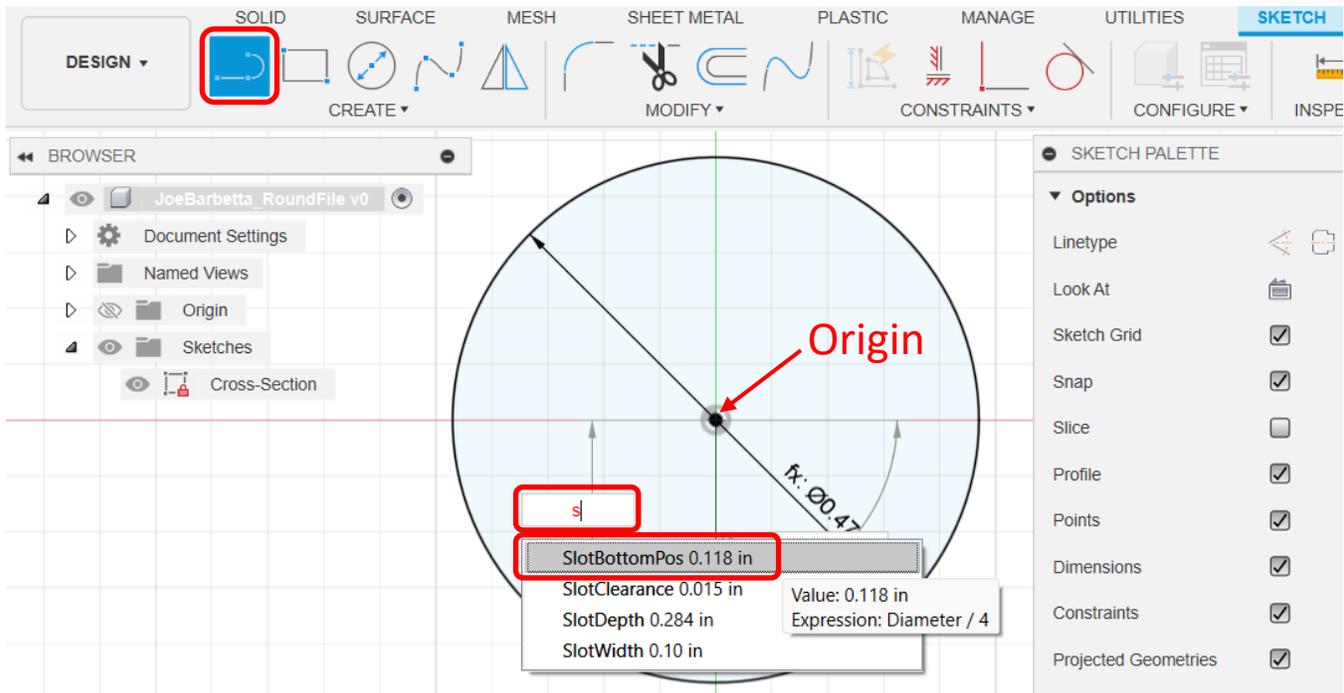
- from the **bottom of that line**, start a line, **extend it to the right**, select **SlotWidth**, but Do Not click enter yet
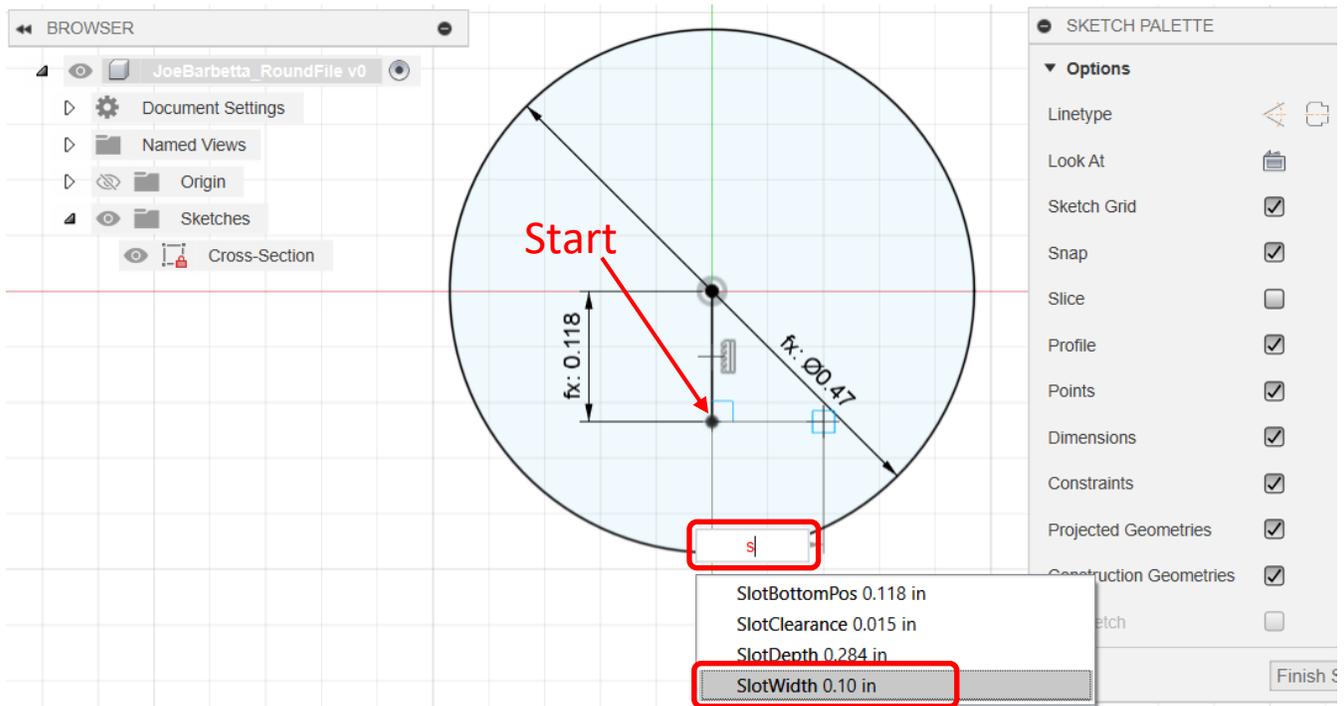
- press the **right arrow key** and then after **SlotWidth** type **/ 2** and press the **Enter key**

- click on the **0.05 dimension** and drag it up and to the right to make the dimension more visible, as shown on the next picture

This should be the result.
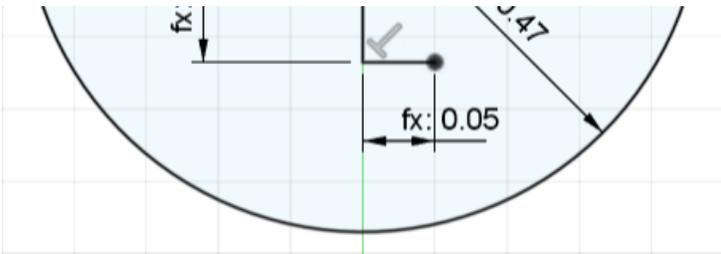
- create a 2nd line starting from the same point, but **extend it to the left** and again select **SlotWidth** and add **/ 2** after it
- drag the Dimension into place,as shown below. The Dimension may show higher up at first. It's nice to have a clean Sketch.
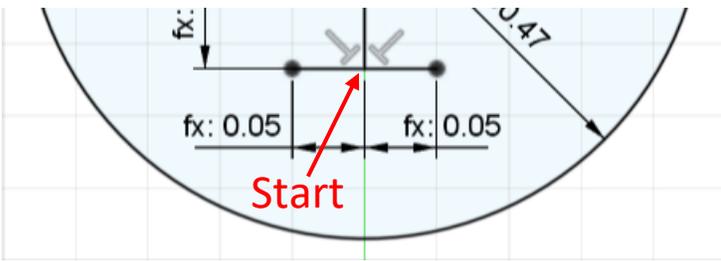

- create a line starting from the **end of a bottom line**, extend it upward, type **s**, and select **SlotDepth**
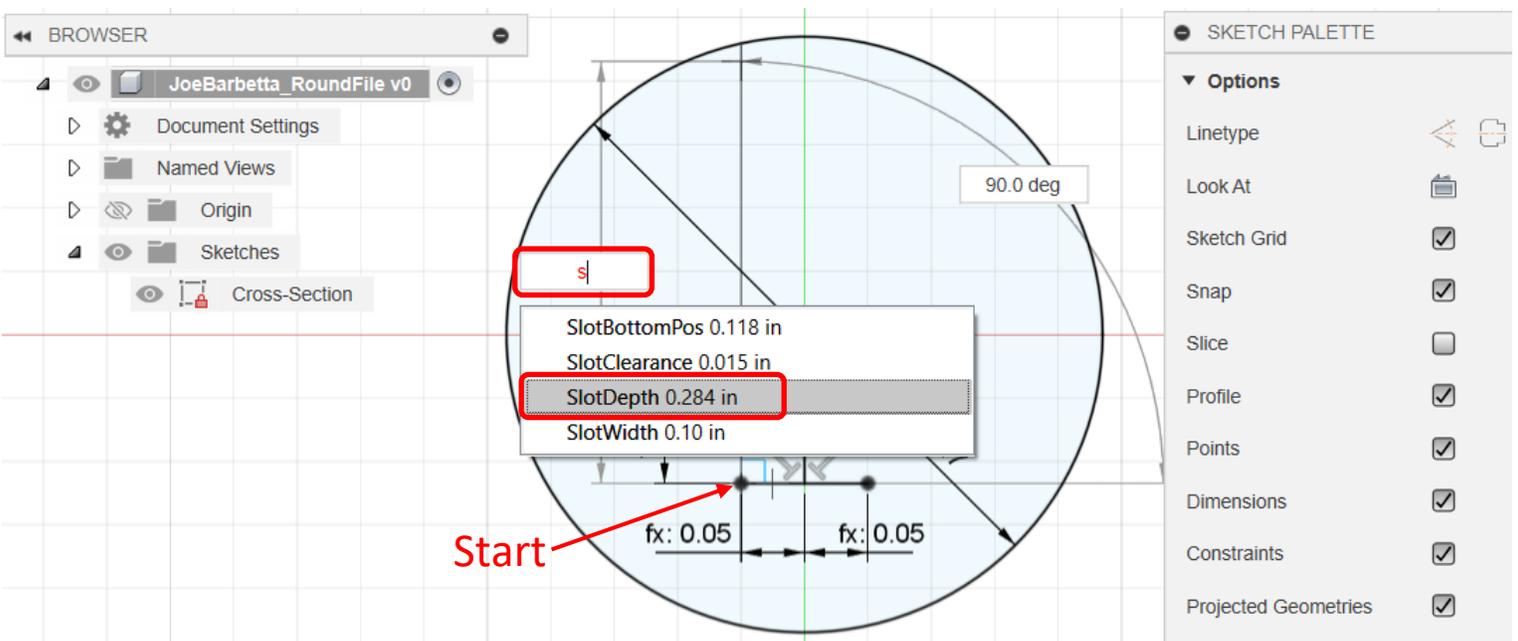
- create another line from the end of the other bottom line and upward using **SlotDepth** as well
- move the 2 **SlotDepth** dimensions to clean up the Sketch, as shown below
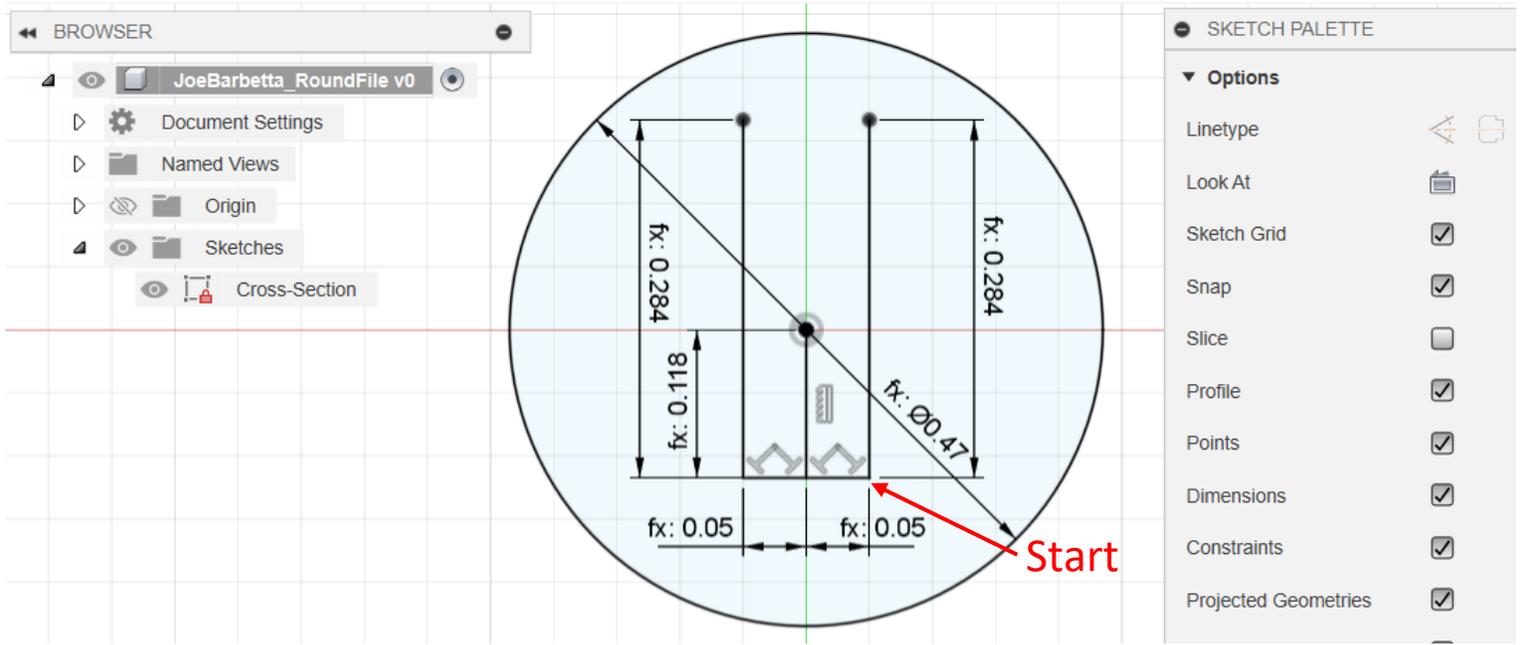
- create a line from the **top of the right side line** and extend it to the right. When it touches the circle and a **blue x** shows, click at that position.
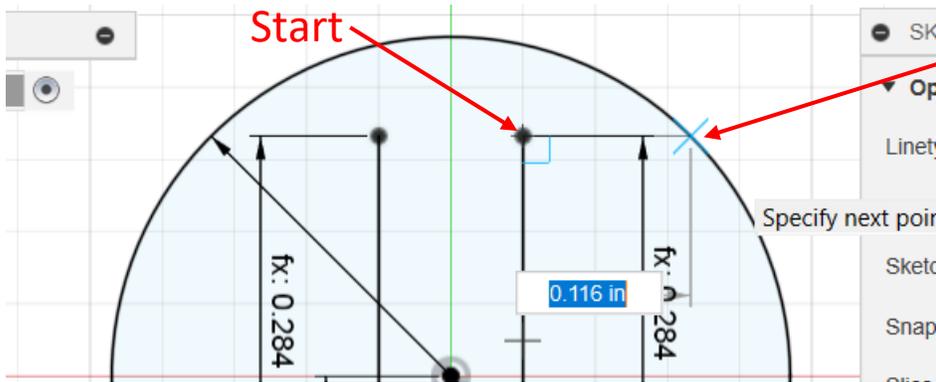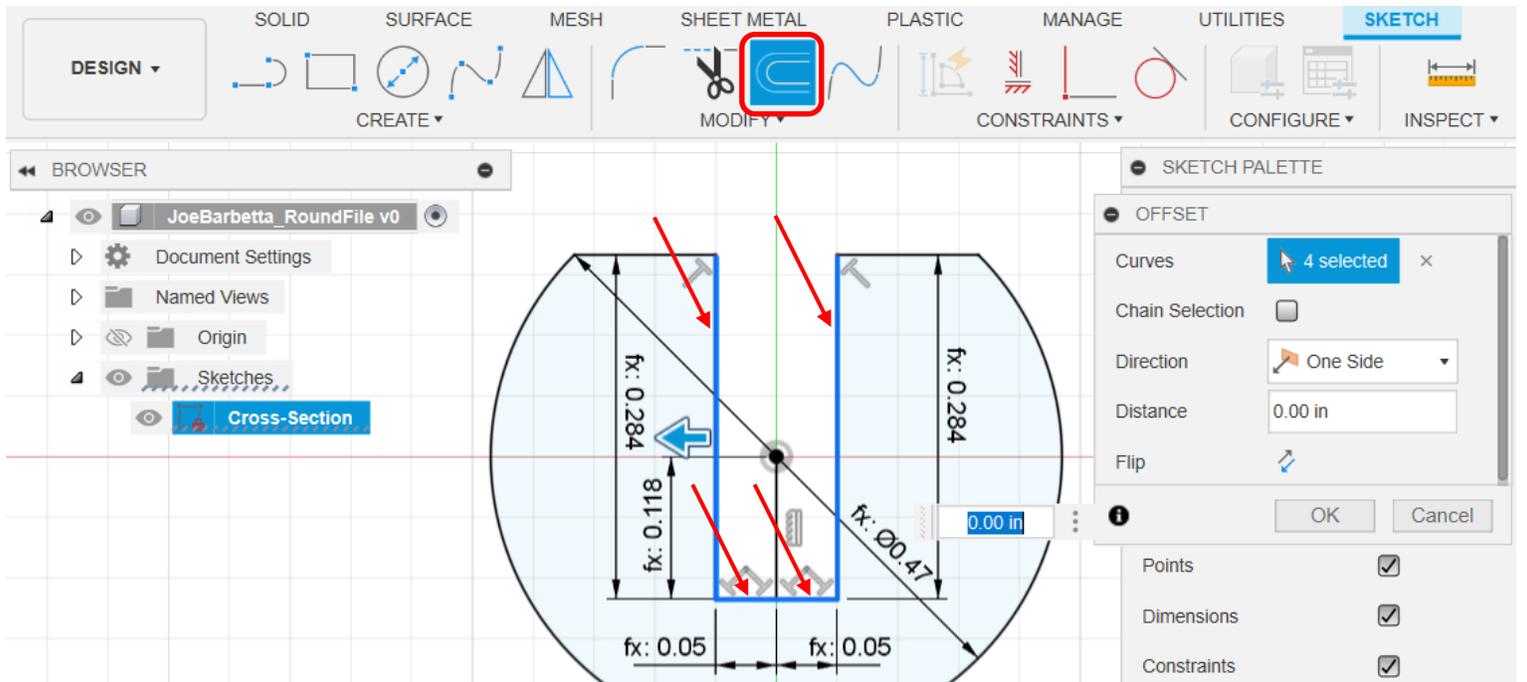

- create another line to the circle from the **top of the left side line**
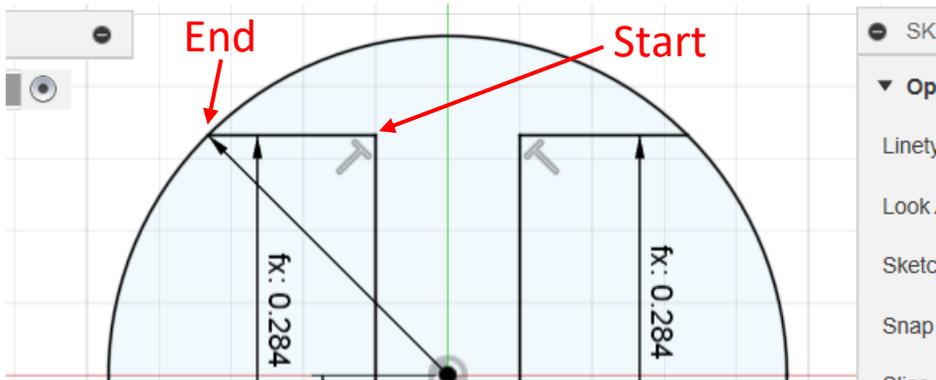
- select the **Trim** tool. If it is not visible, find it in the MODIFY menu
- click on the **top of the circle**. Ignore any warning message that may show.
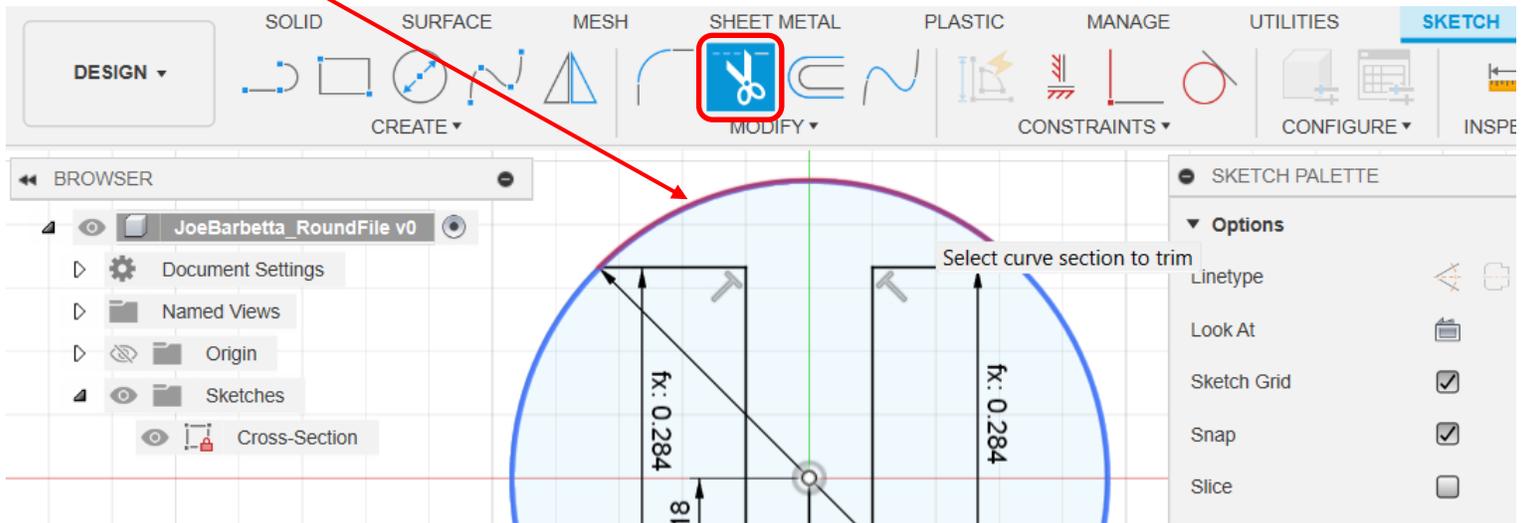
- select the **Offset** tool. If it is not visible, find it in the MODIFY menu.
- click on the **4 line segments** indicated by the arrows, which should turn the lines blue
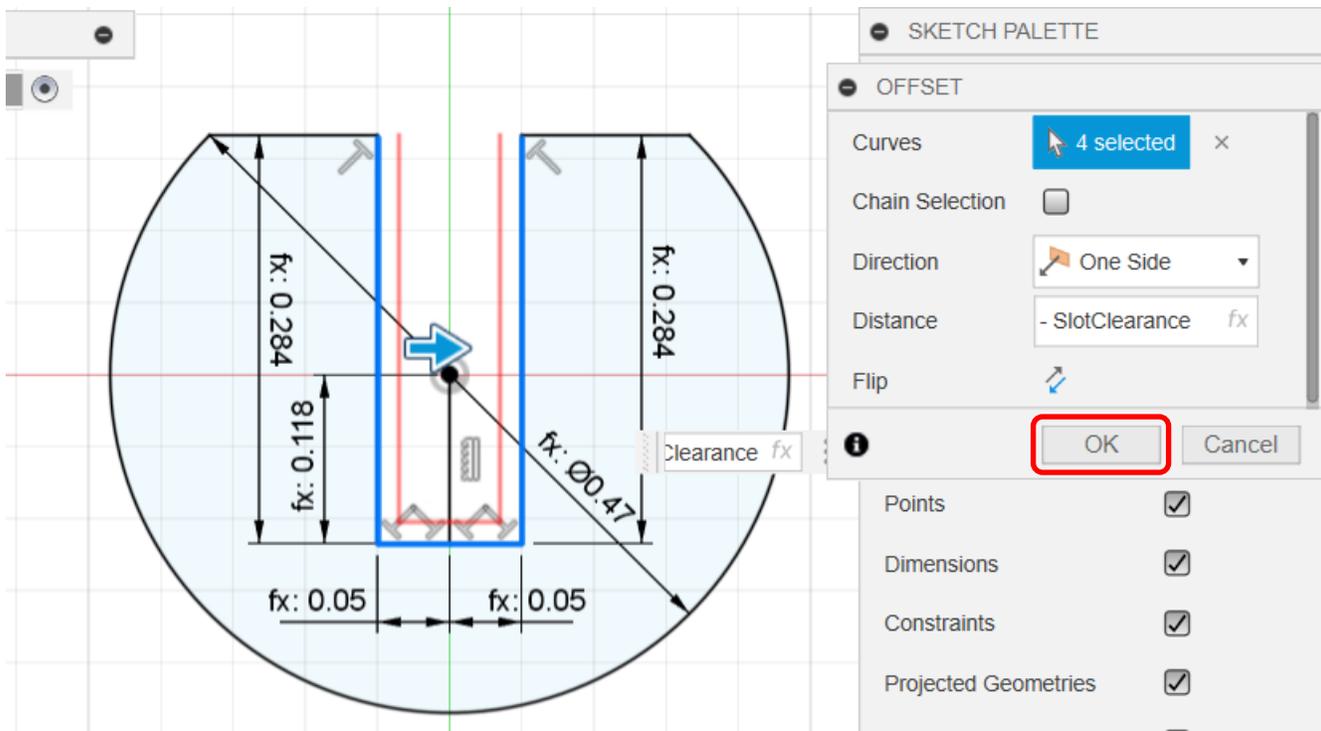
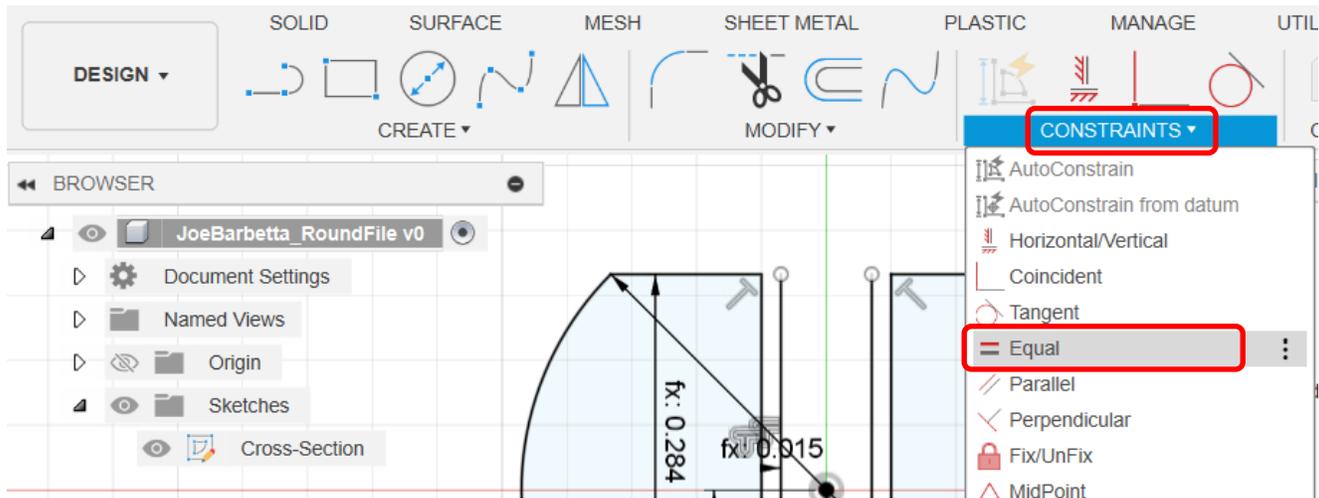- in the **Distance** box type **-** (minus sign) followed by **s** and select **SlotClearance**
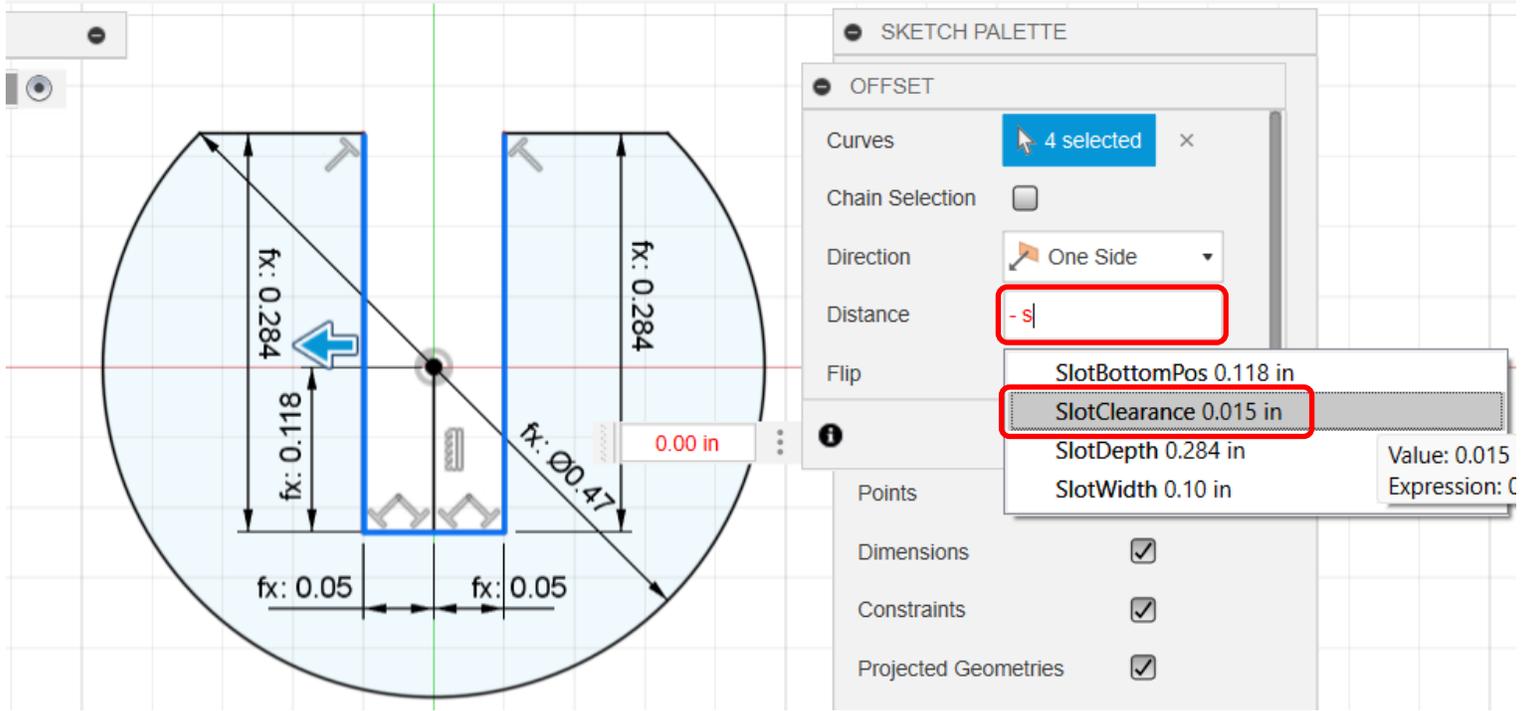
- the red lines should appear **inside the slot**. If they are outside of the slot, click the Flip icon.
- click **OK**

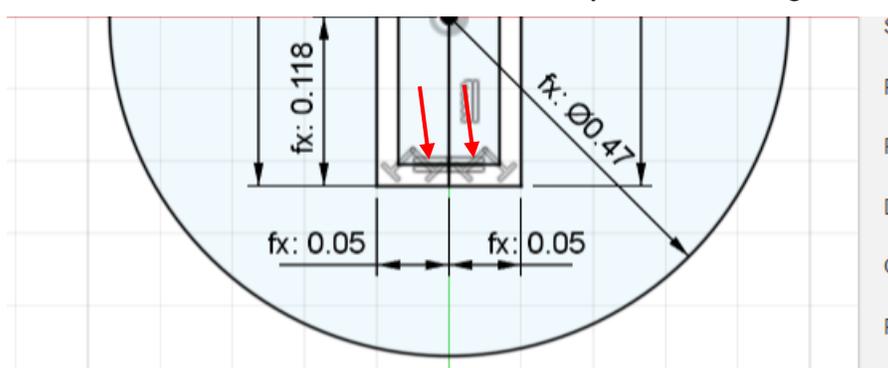- from the **CONSTRAINTS** menu, select **Equal**

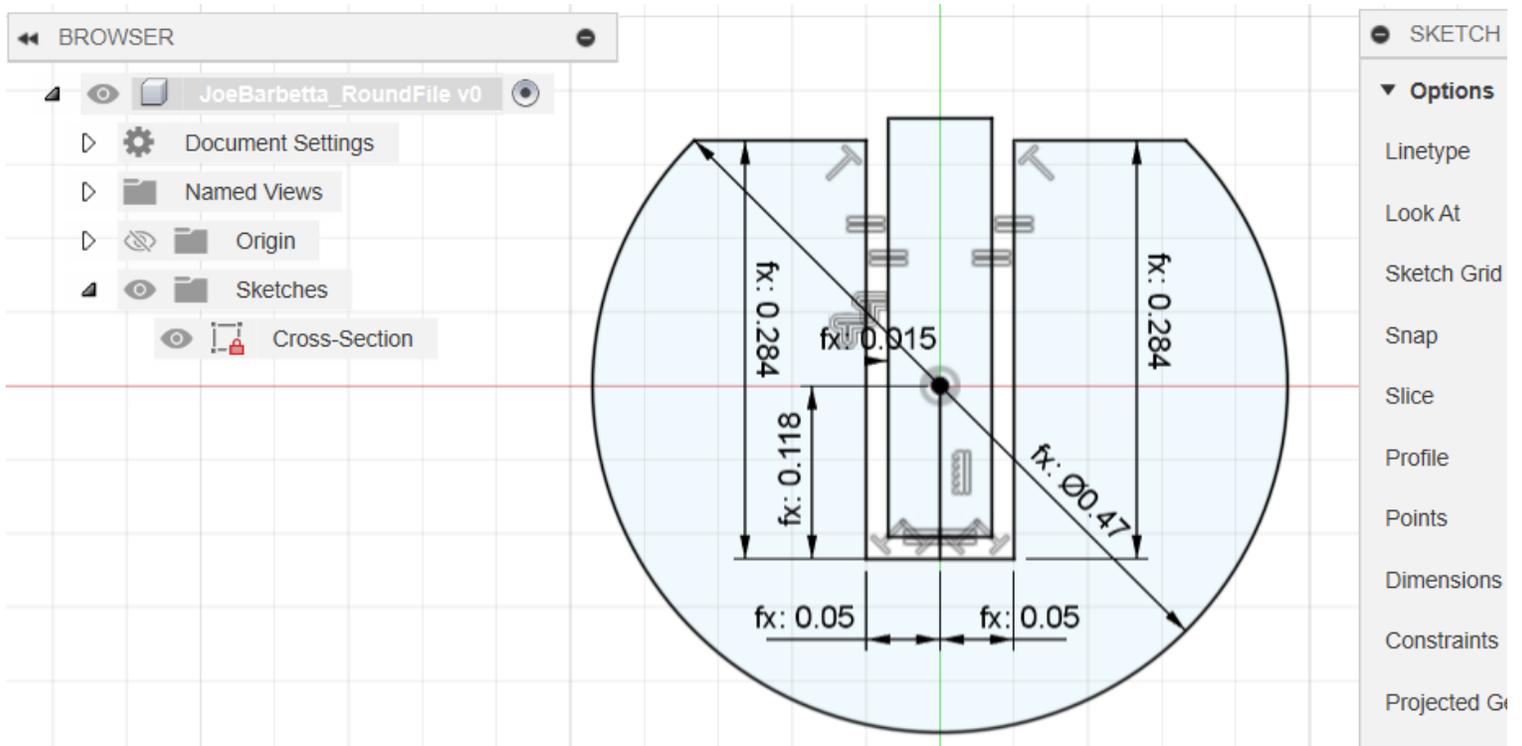- click on the **4 vertical** lines and then press the **Esc key**

- create a line between the **2 top endpoints**. No value needs to be entered.
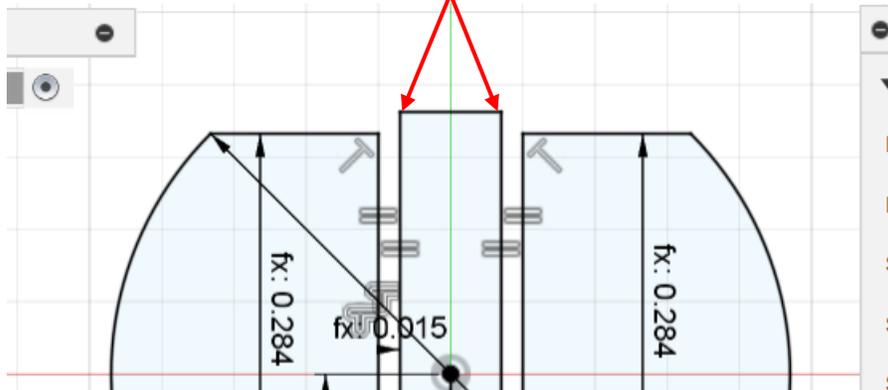
- zoom into the bottom of the circle, select the **Equal Constraint** again and click on the **2 bottom lines** indicated
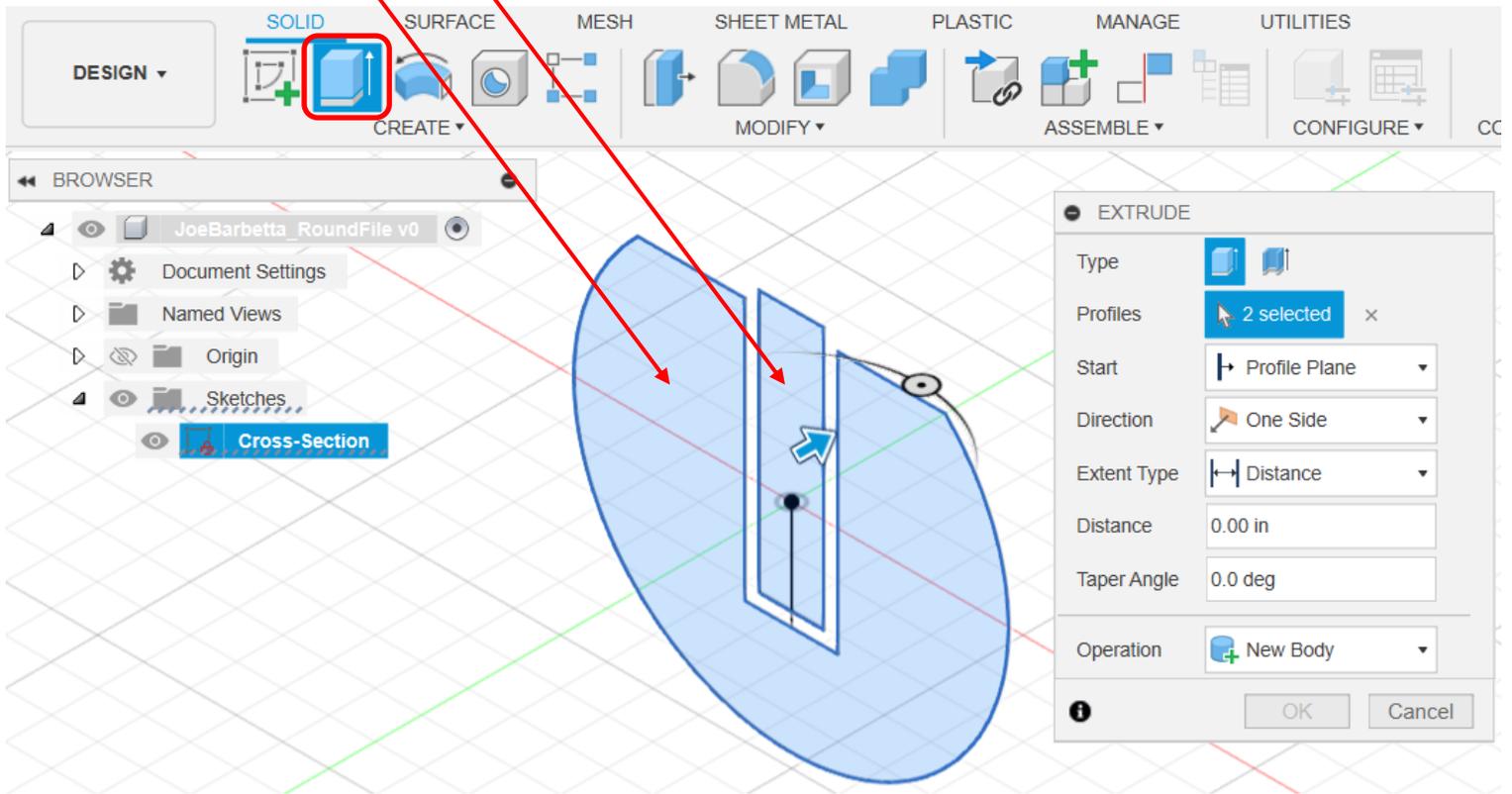
- admire your Sketch. It is OK if the Dimension lines look different.
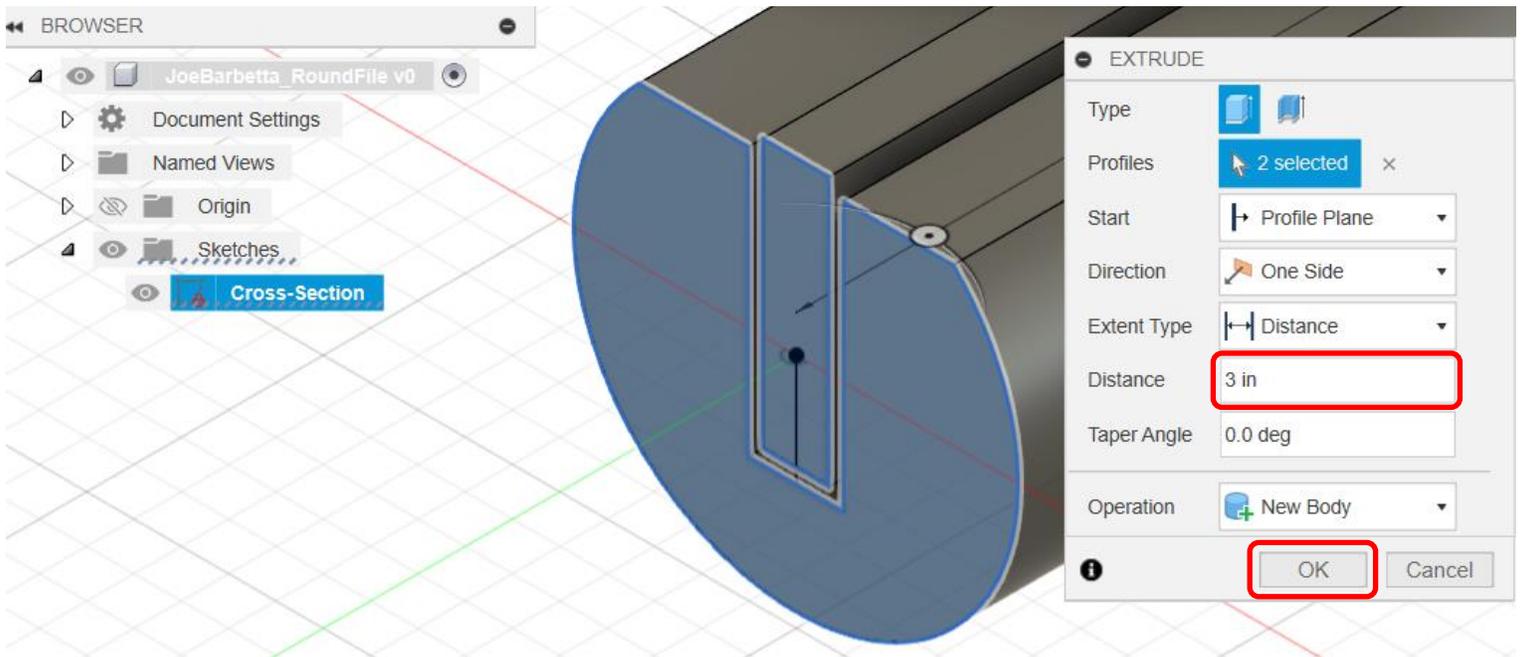- click on **Finish Sketch** at the bottom of the SKETCH window

- click on the **Home** icon at the **View Cube** and adjust the view similar to that below
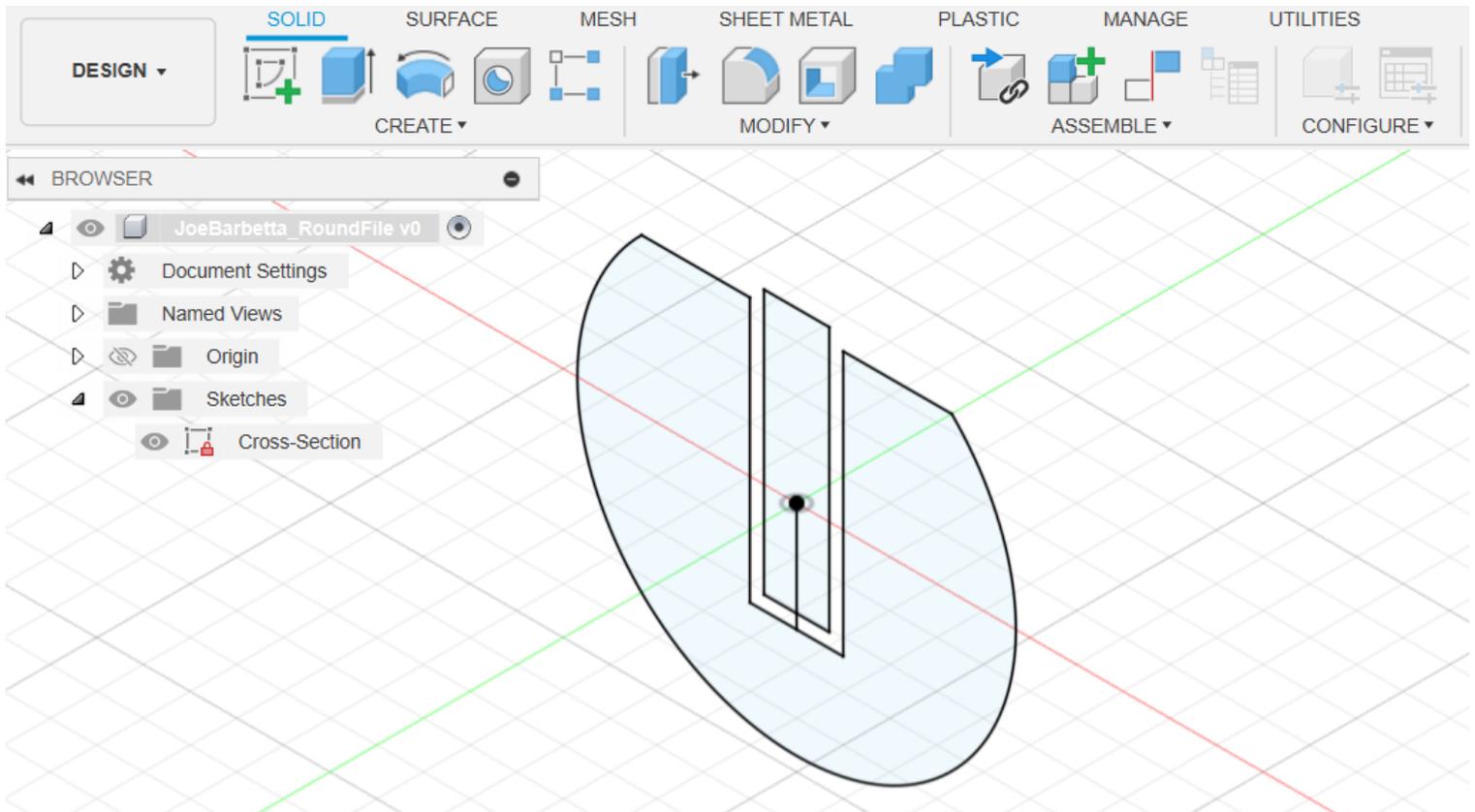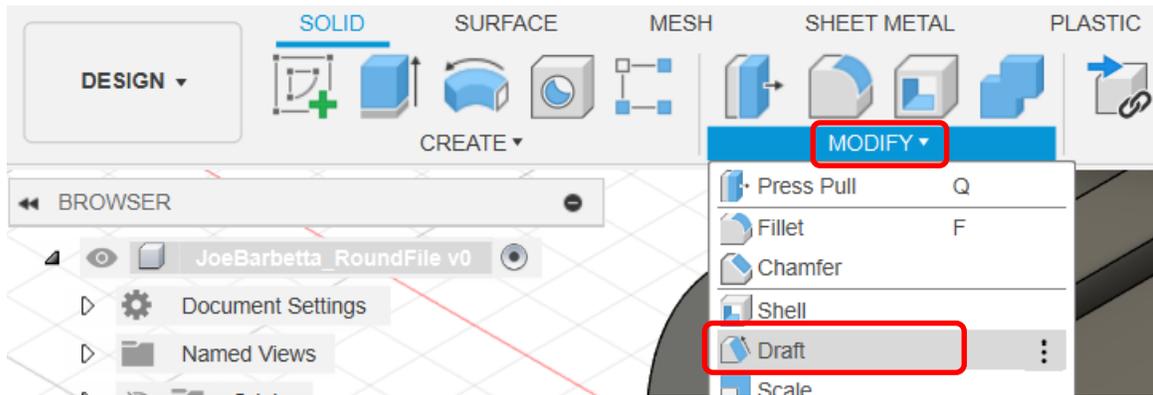
- select the **Extrude** tool. If it is not visible, find it in the CREATE menu.
- click on the interior of the **outer** and **inner** shapes
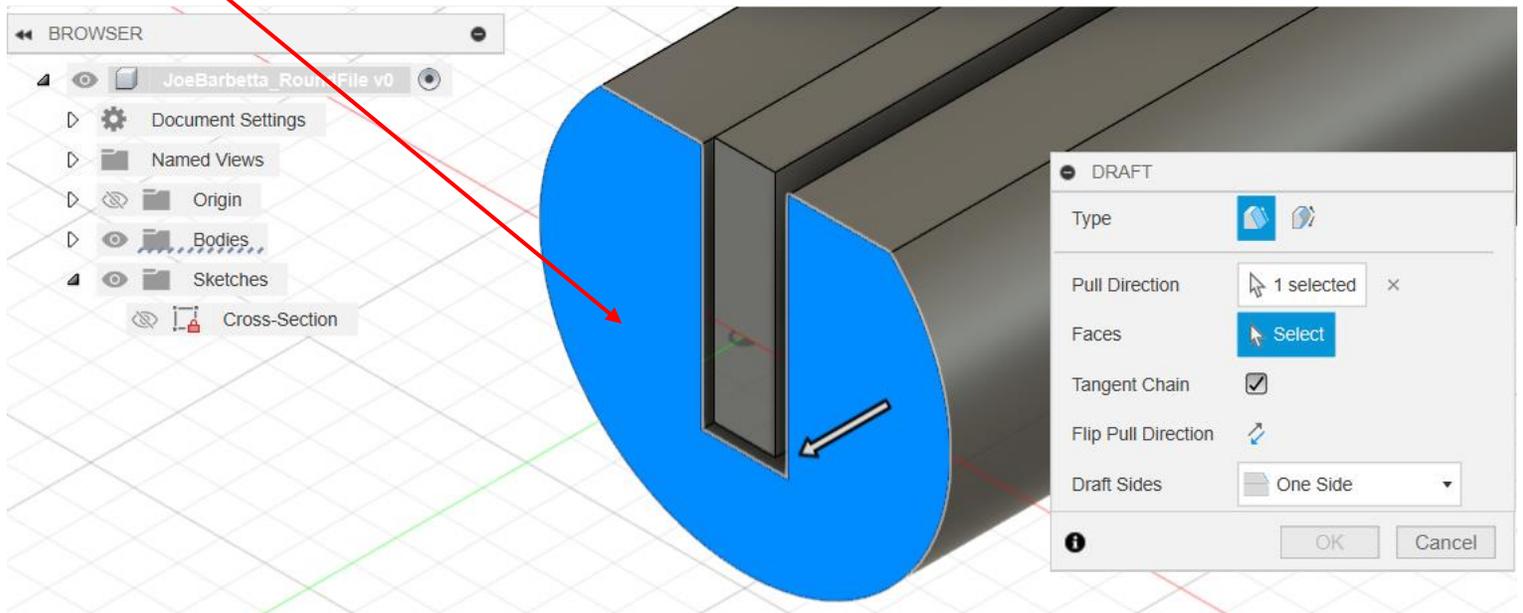

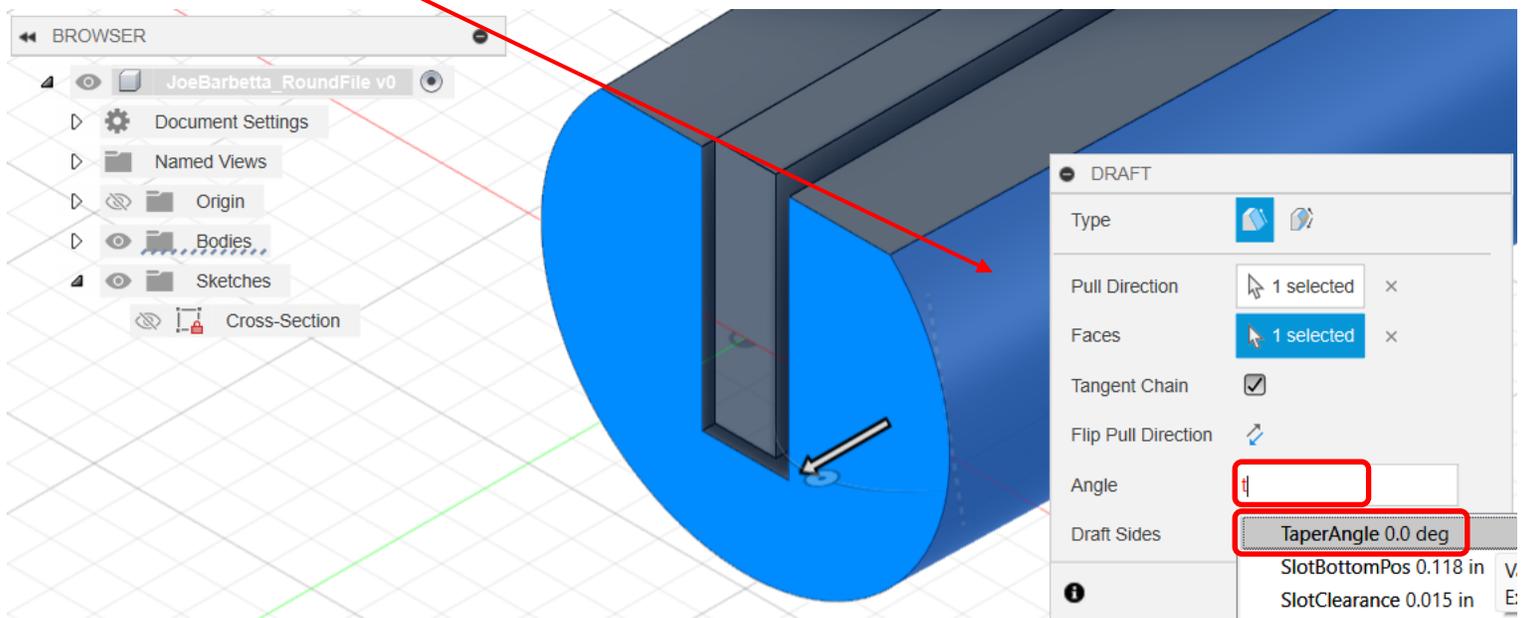- for **Distance** enter **3** and then click **OK**

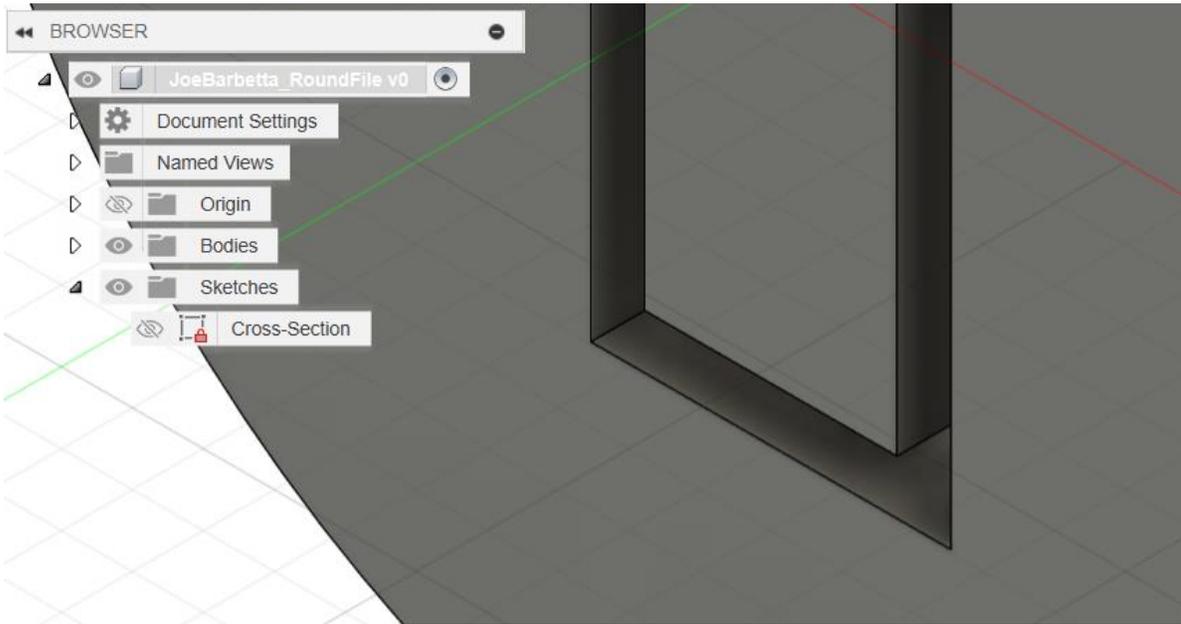- from the **MODIFY** menu, select **Draft**

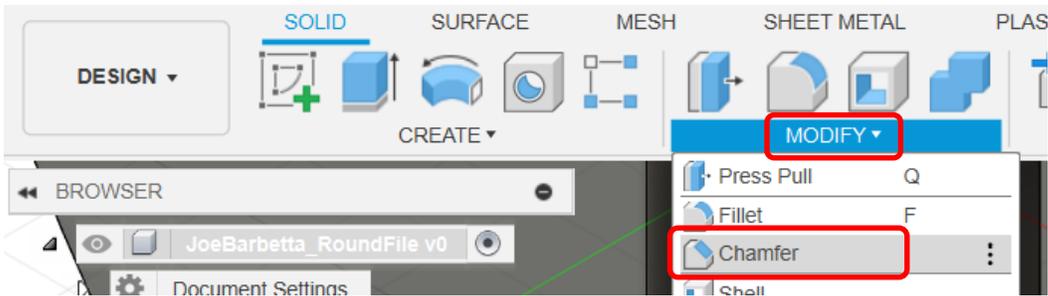- click on the **end face of the outer section**, which should turn it blue

- click on the **outer cylindrical surface**
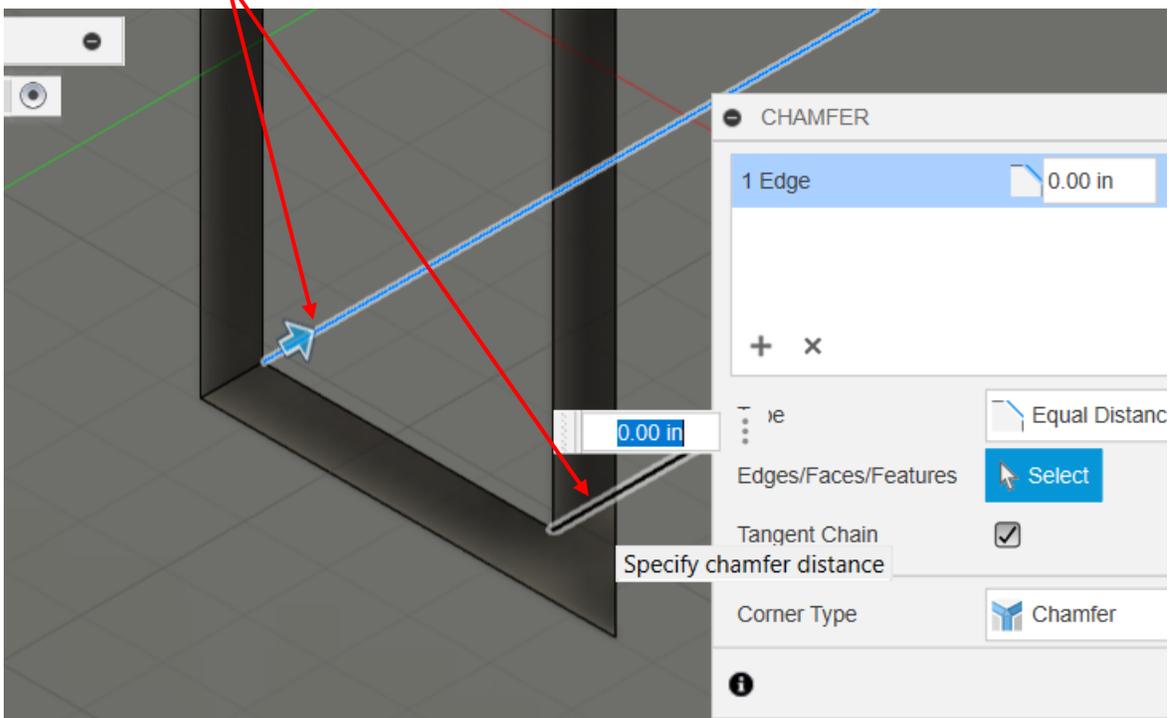- in the **Angle** box type **t** and select **TaperAngle** and click **OK**
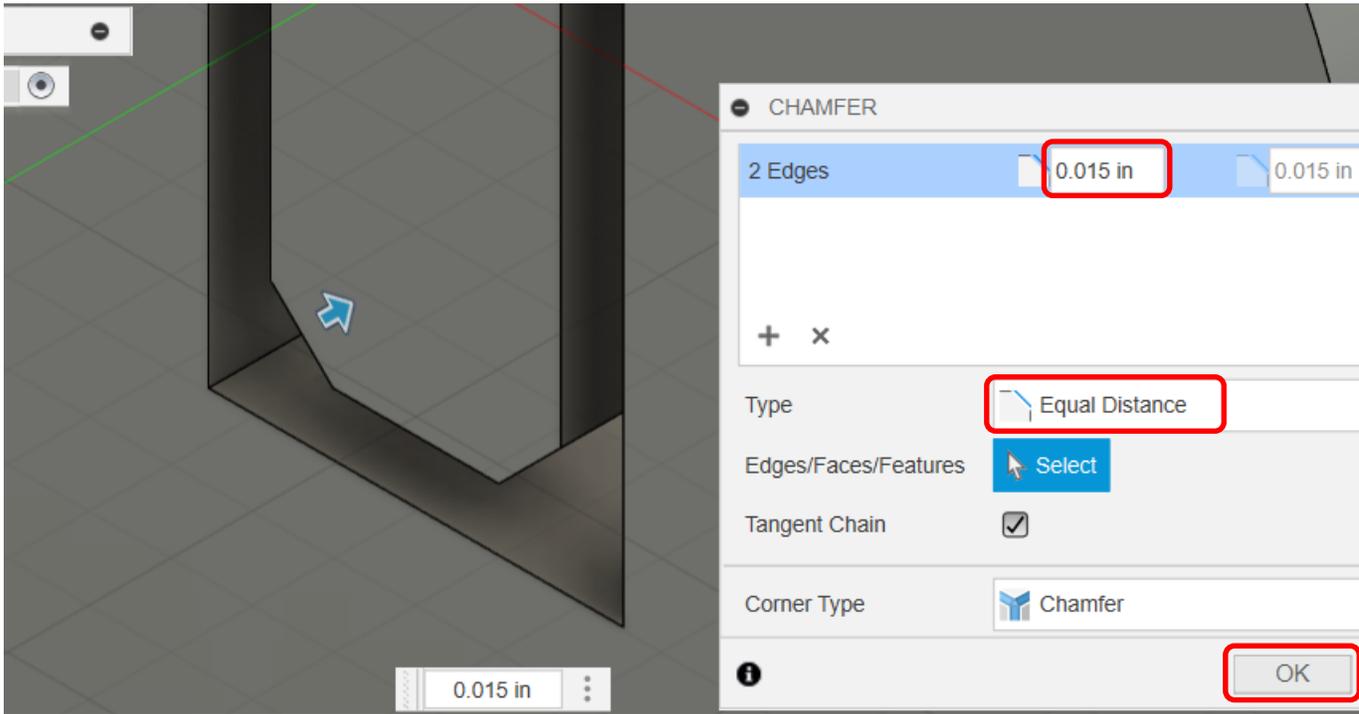
- zoom into the slot, as shown
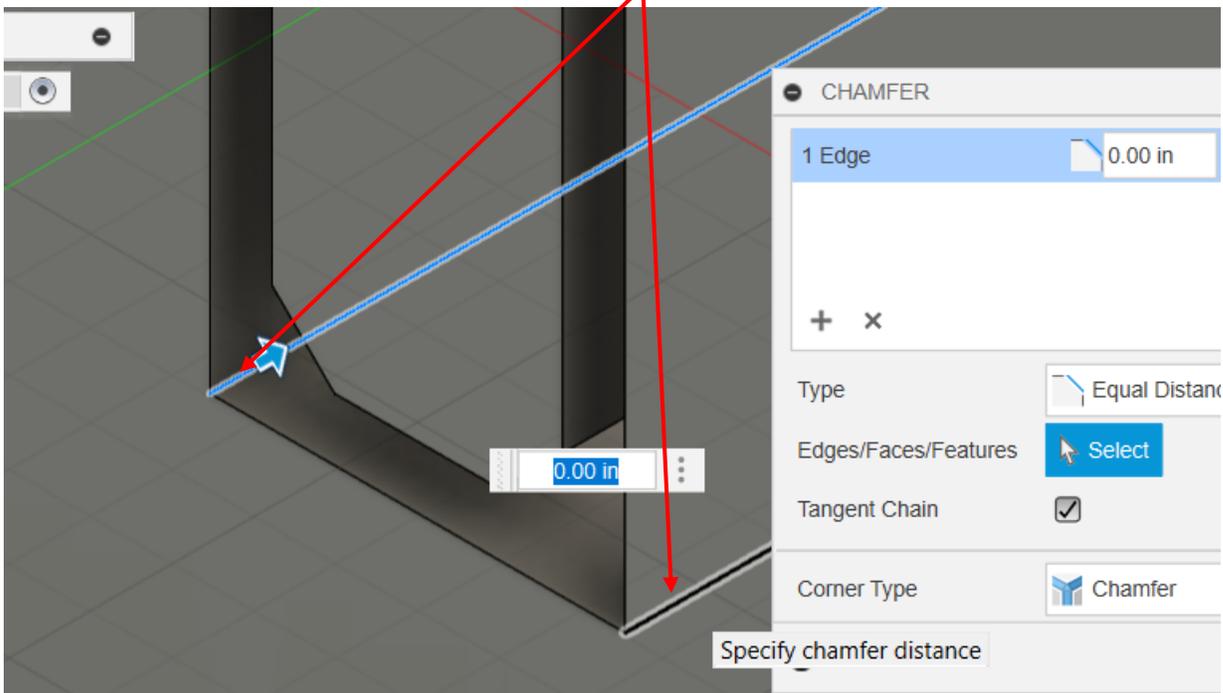

- from the **MODIFY** menu, select **Chamfer**

- click on the **2 bottom edges of the center body**. Note that for the left edge, Fusion allows one to select through the body.
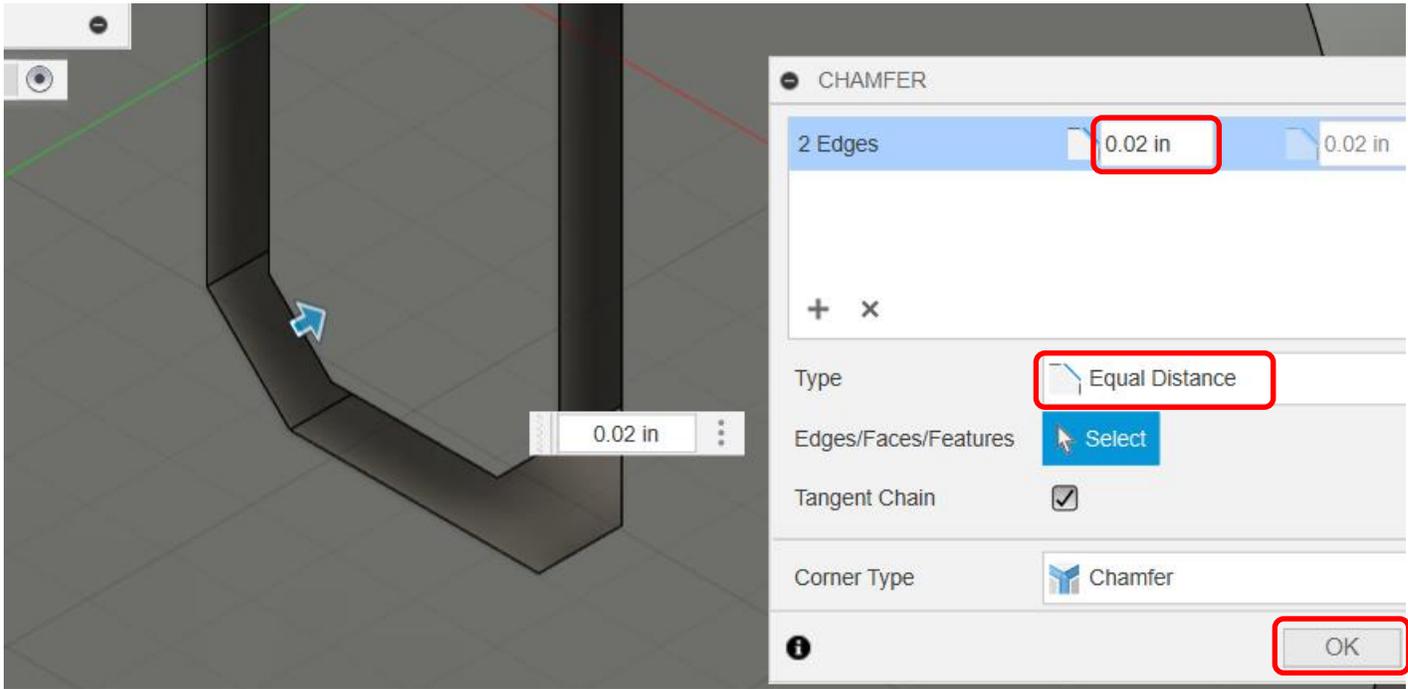
- enter **0.015**, ensure that **Type** is set to **Equal Distance**, and click **OK**
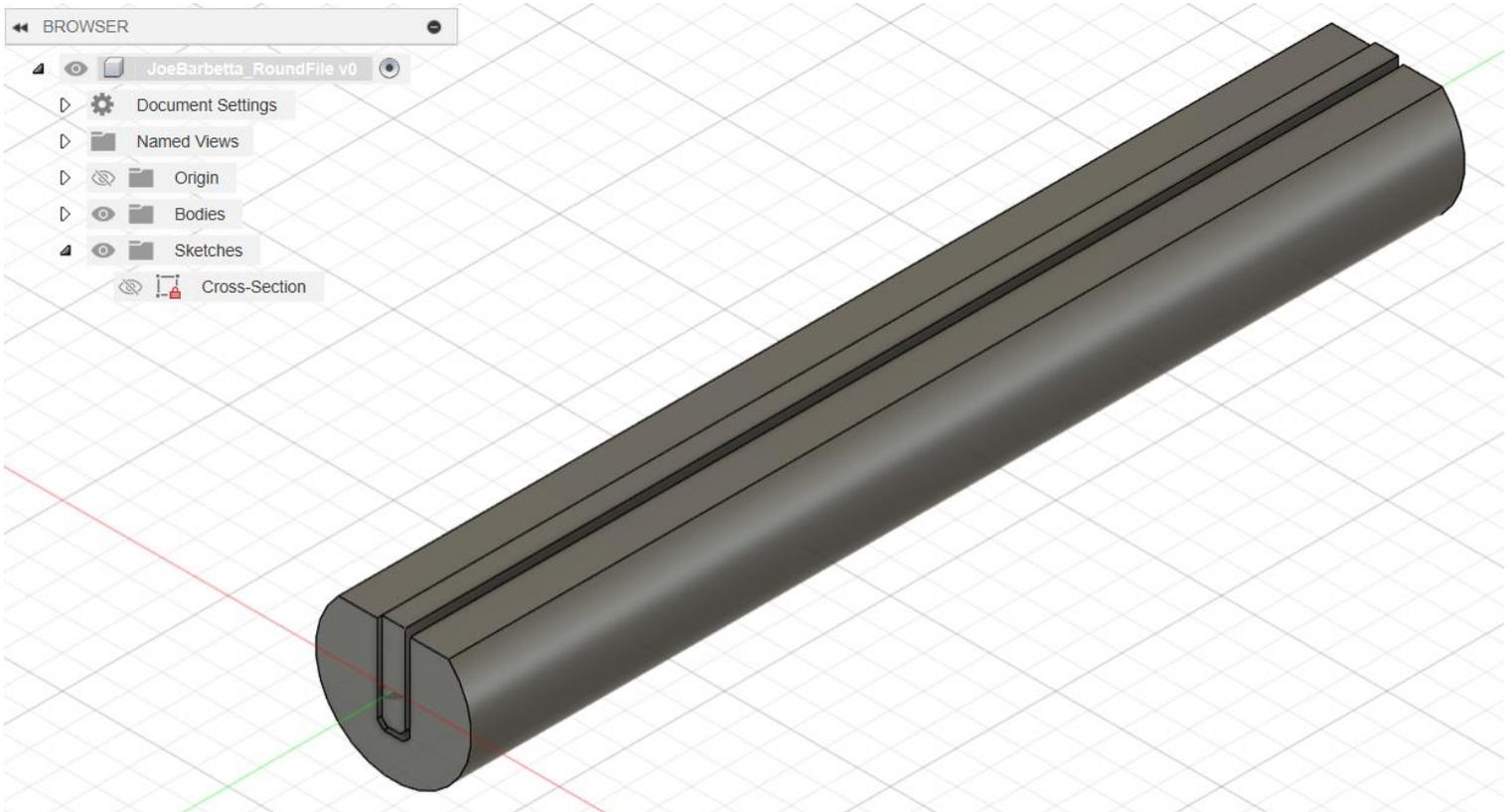

- select the **Chamfer** tool again and click on the **bottom 2 edges** of the slot

- enter **0.02**, ensure that **Type** is set to **Equal Distance**, and click **OK**
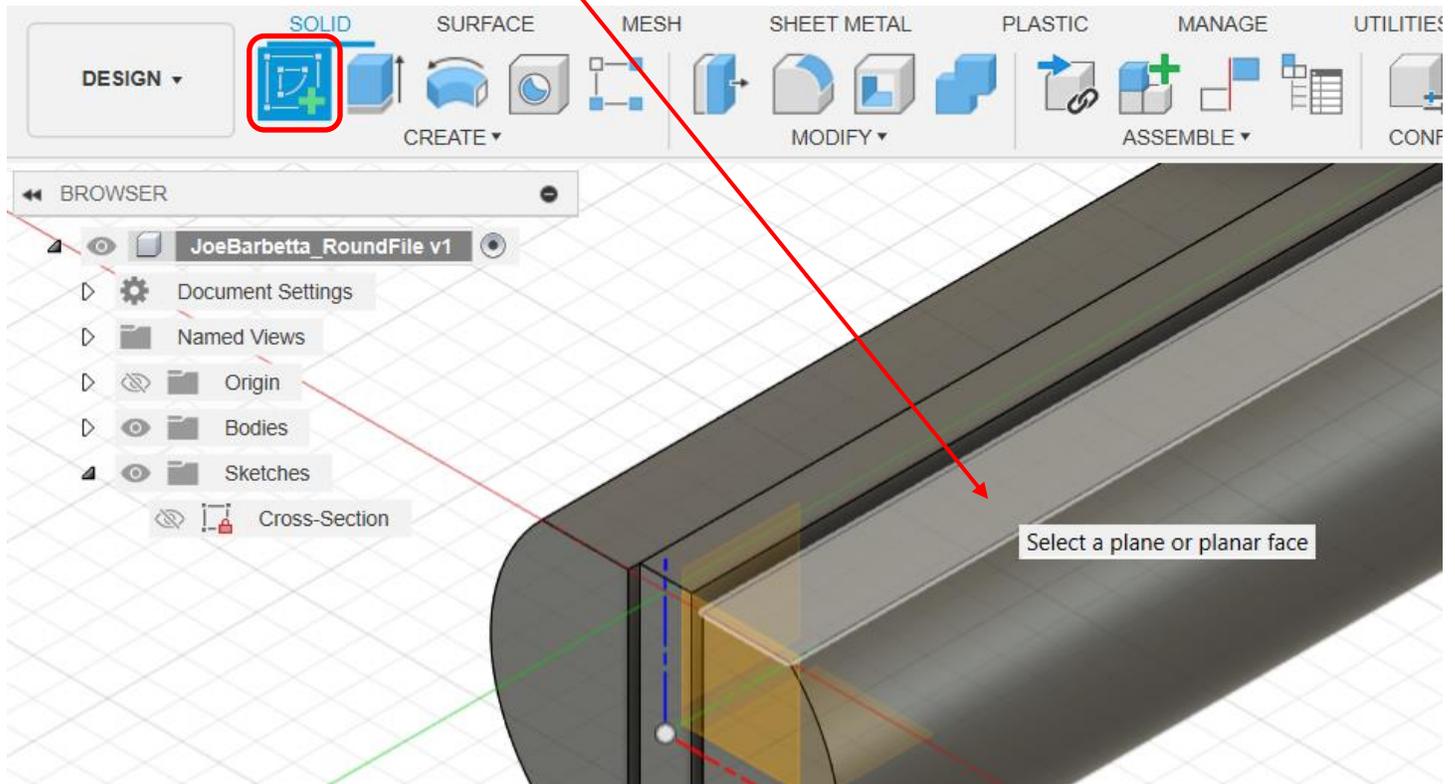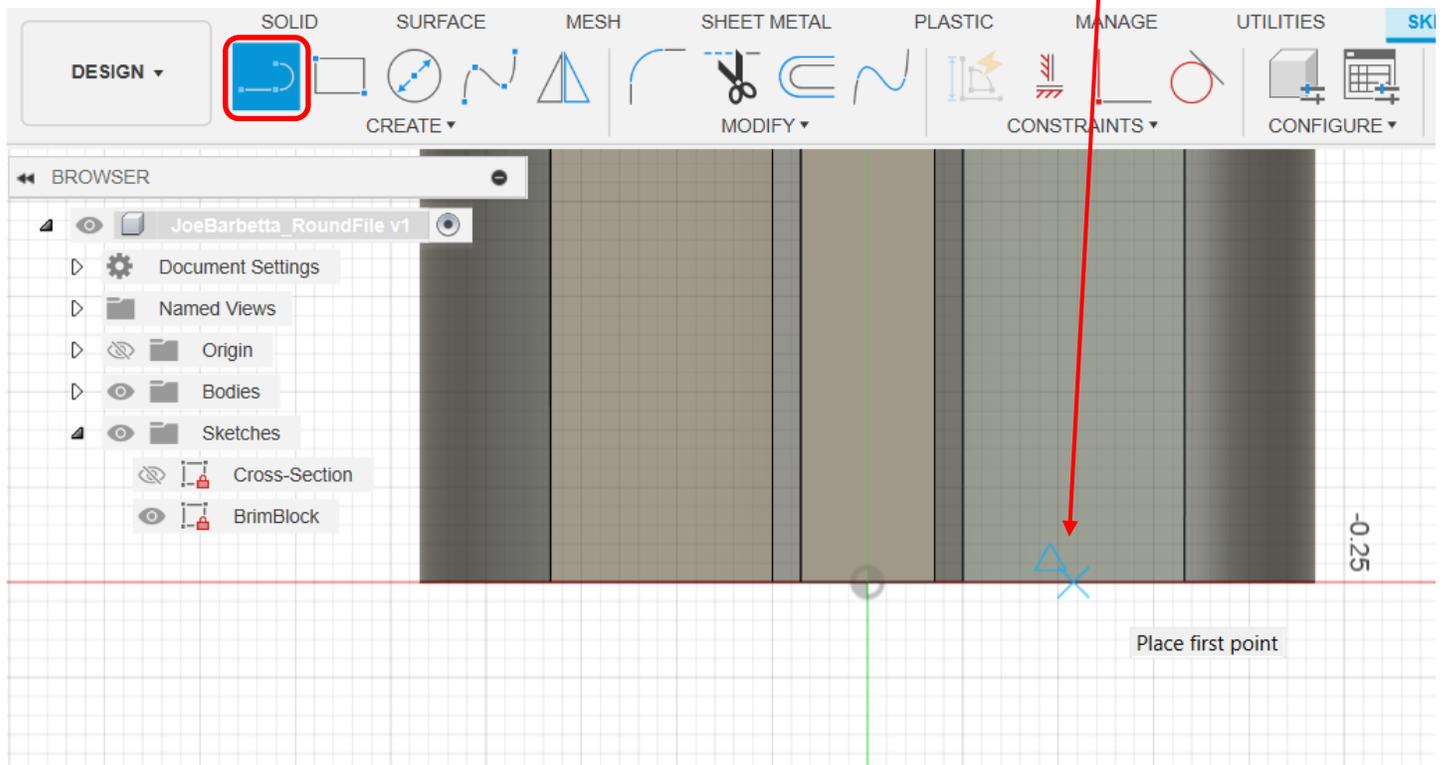

- zoom out and admire your file

Adding a Brim Block

- select **Create Sketch** and click on the **top flat surface** as indicated
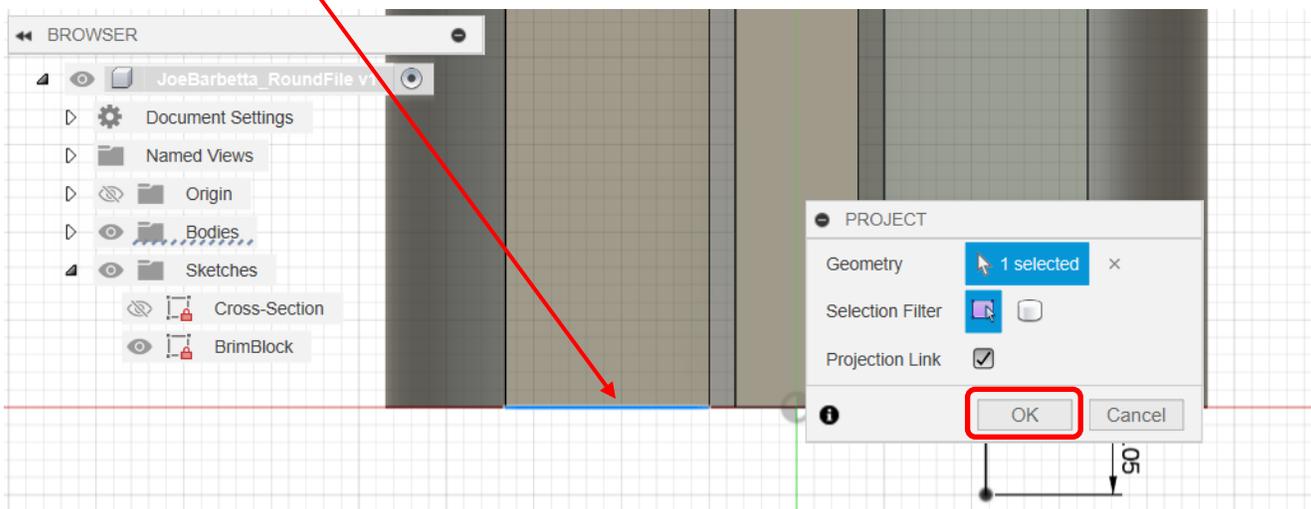

Select a plane or planar face

- zoom into one end
- select the **Line** tool and move the mouse along the **right bottom edge** and when the **blue triangle** appears and click at that point
- **extend the line downward** and enter a value of **0.05**
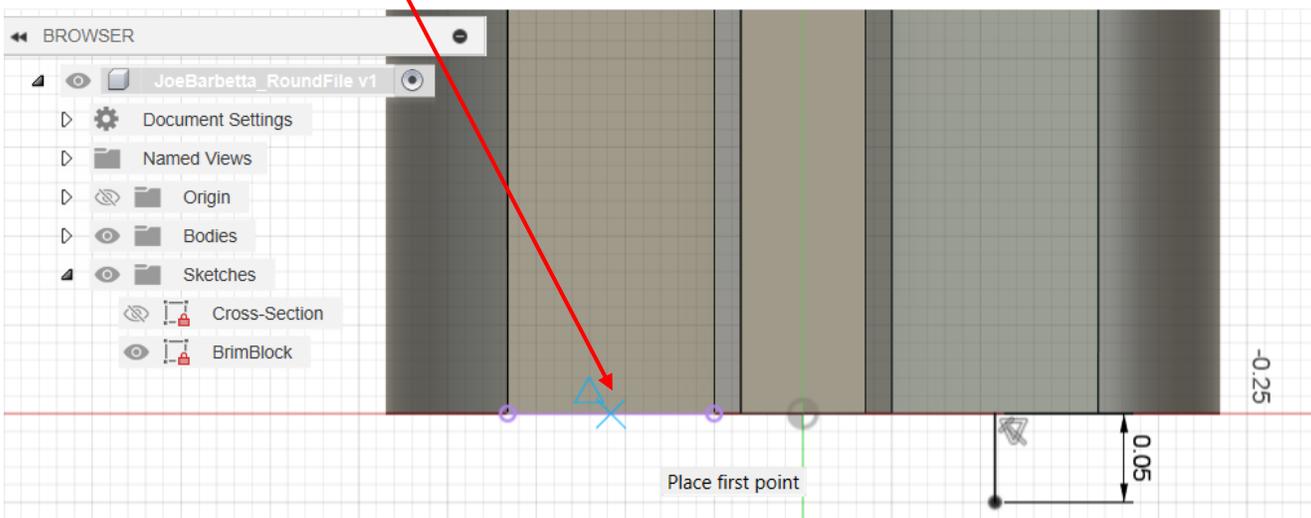

Place first point

- from the **CREATE** menu select **Project / Include** and **Project**
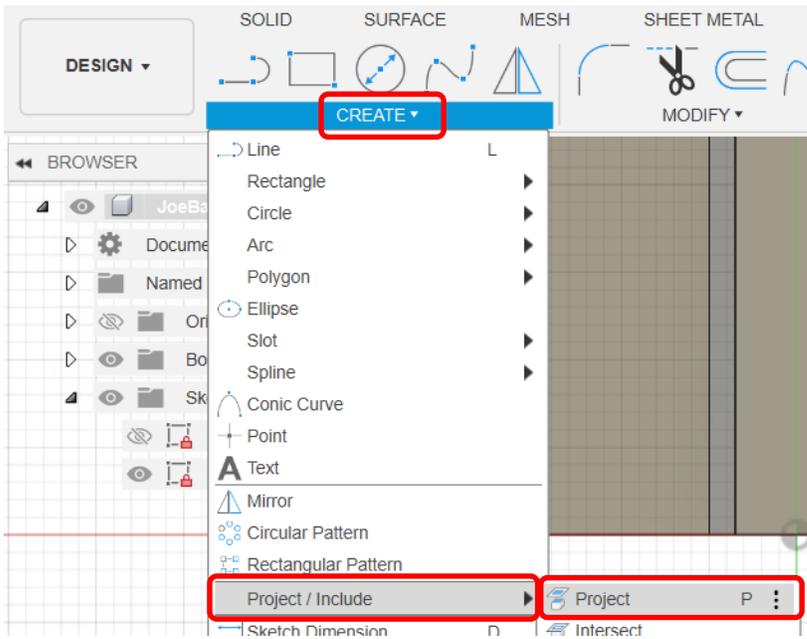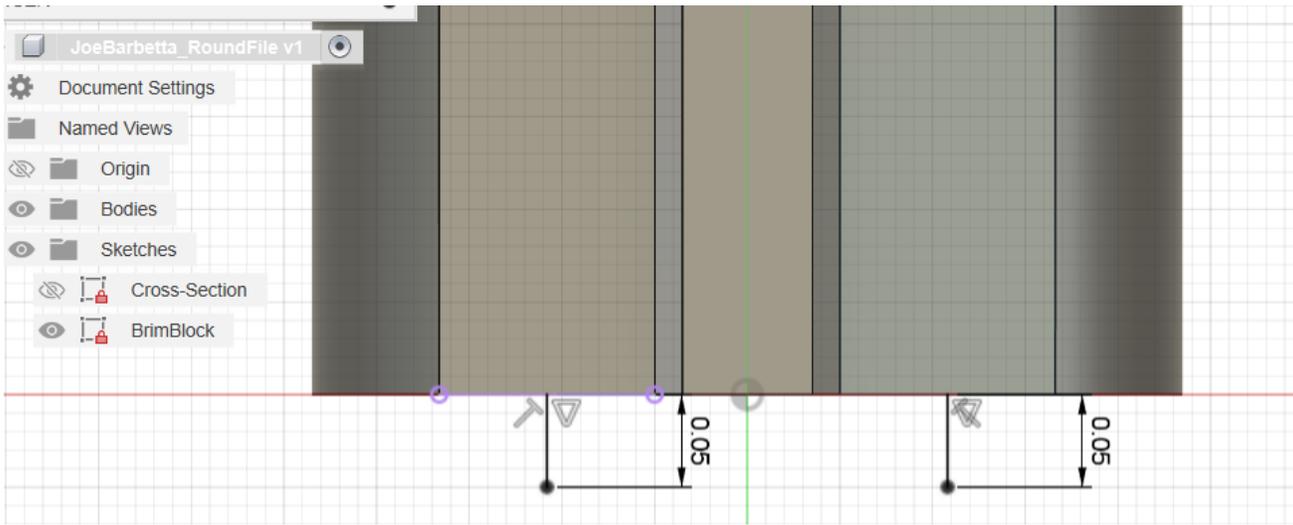

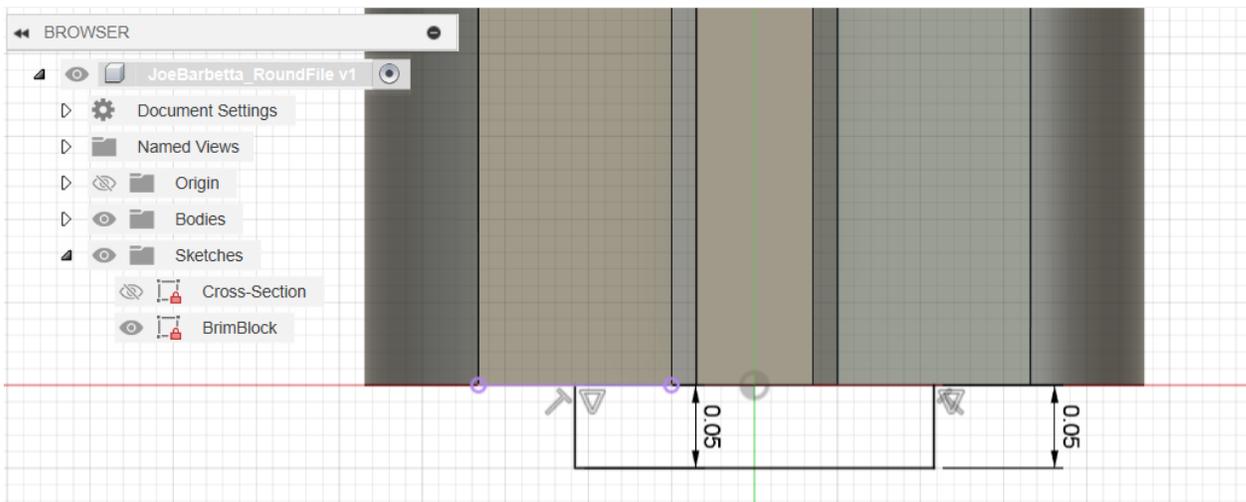- click on the **left side bottom edge**, and click **OK**

- create a line from the **center of the left side** edge downward by **0.05**, as was done from the right side edge
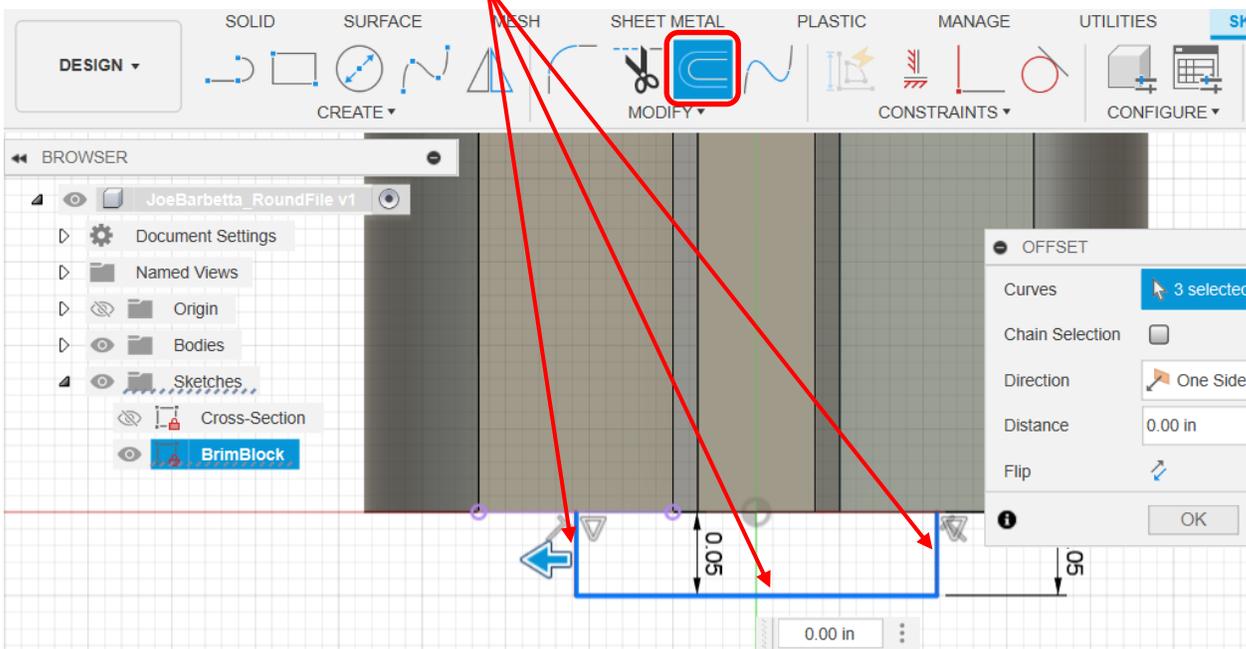
This should be the result of the 2 lines.

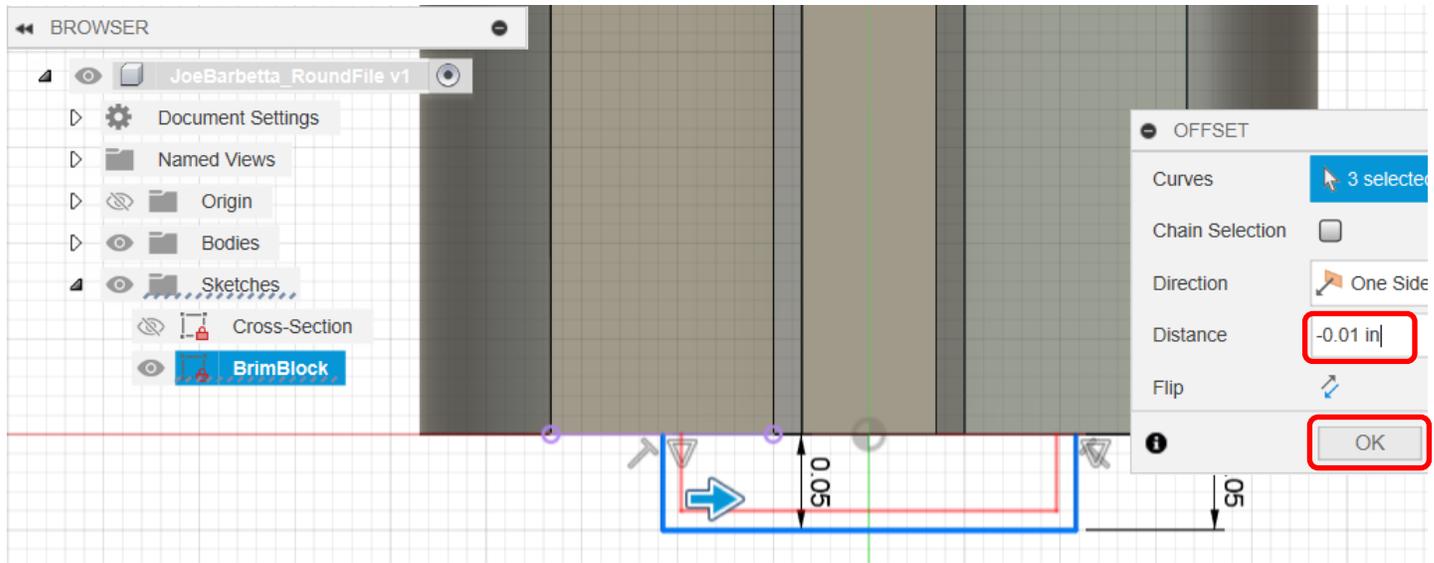- create a line connecting the bottom endpoints. No value needs to be entered.

- select the **Offset** tool and click on the **3 lines** just created

- for **Distance** enter **-0.01** (note the minus sign)
The red lines should be inside the blue lines, as shown. If they are outside of the blue lines, click the Flip icon.
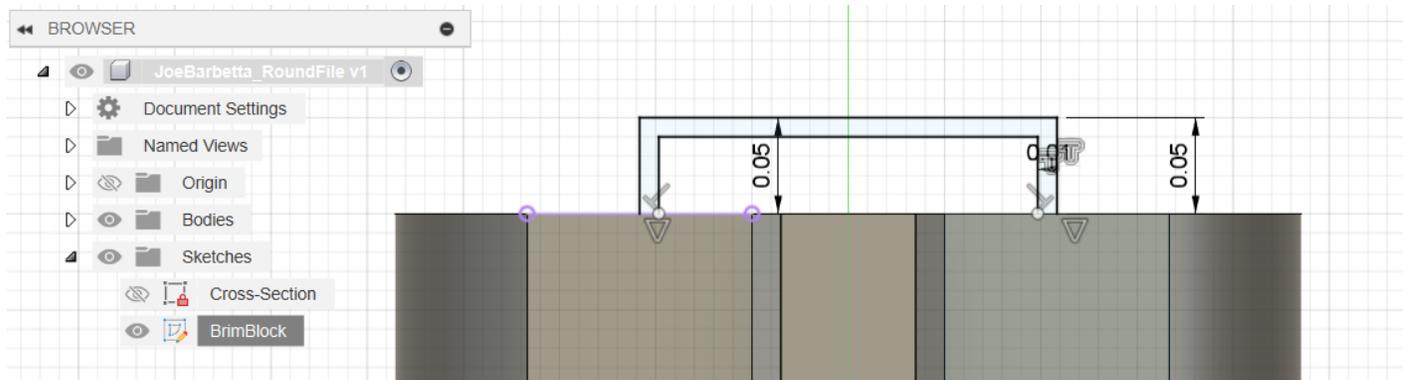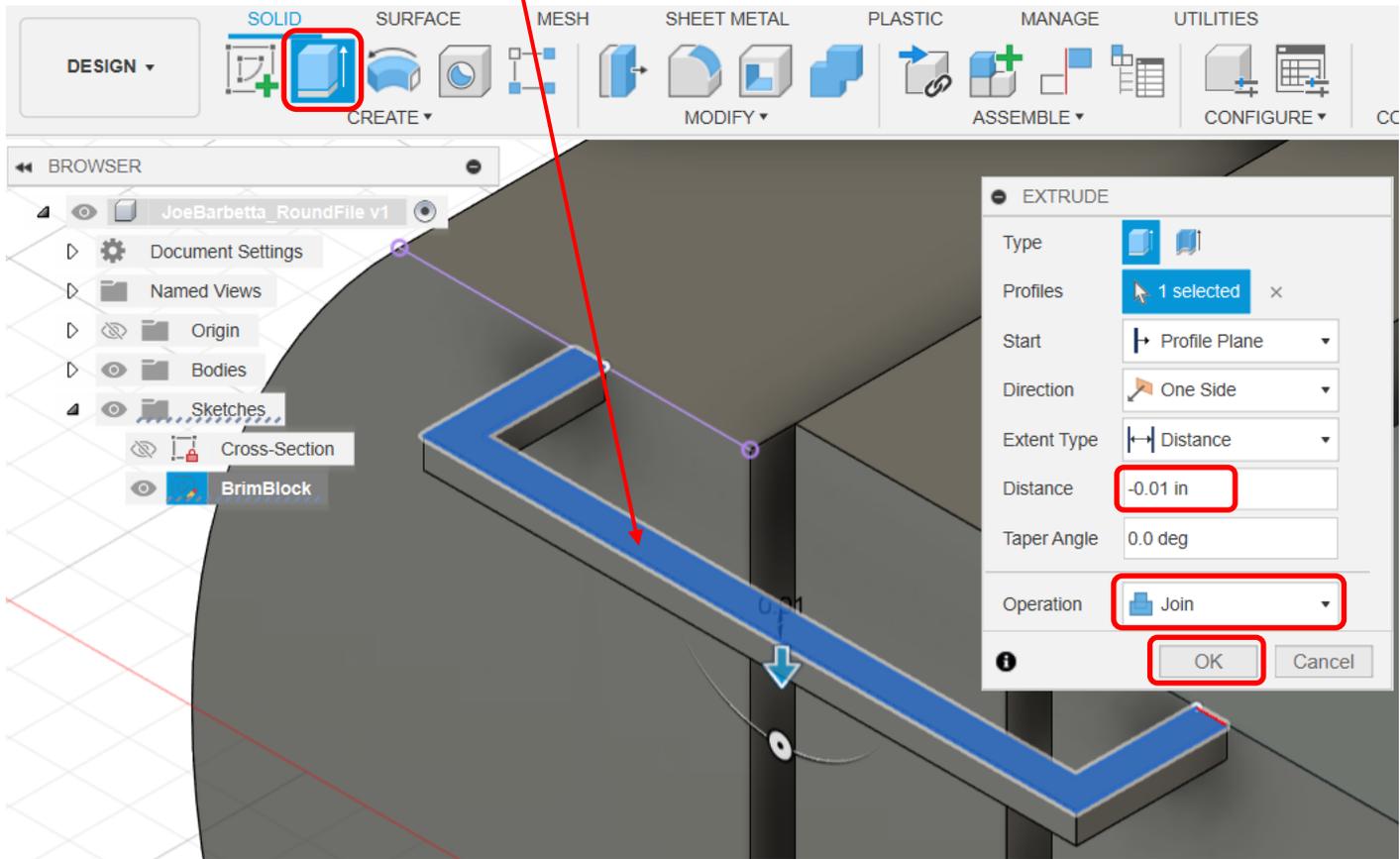- click **OK**

The result should appear as that shown below. It is OK if the dimension lines look different.
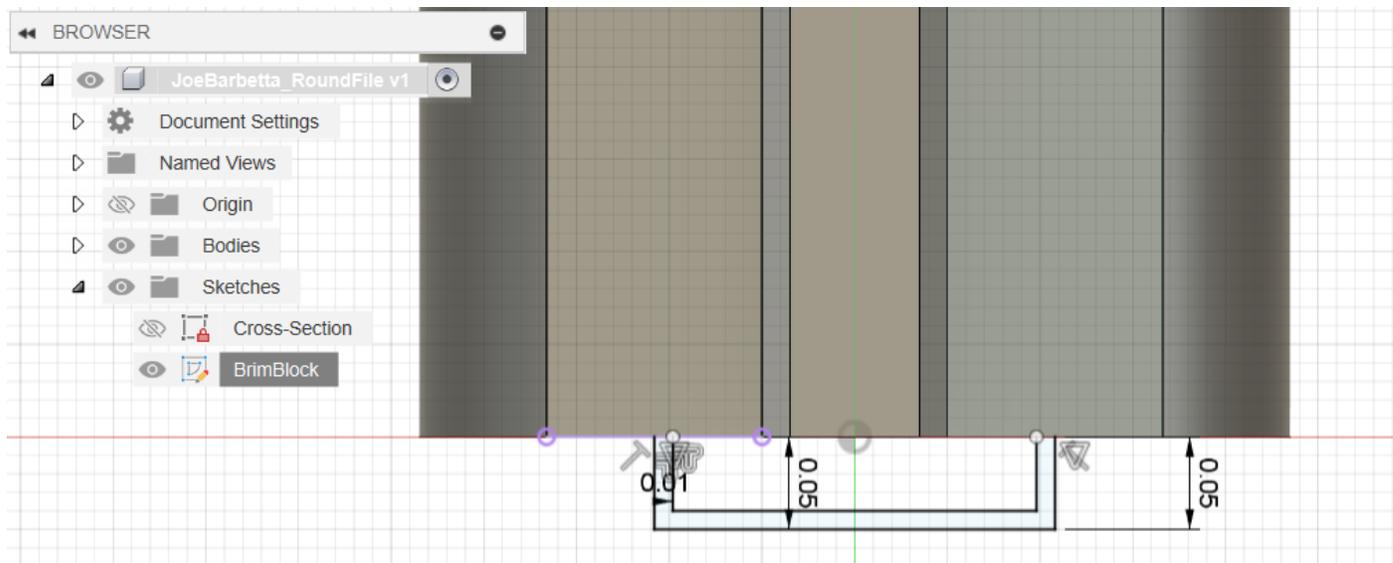

- zoom into the top of the Sketch and perform the same operations of creating lines, projecting, and the offset. The result should look like that below.

- click **Finish Sketch**
- zoom into the end of the body
- select the **Extrude** tool and click on a **region that was just created**
- for **Distance**, enter -**0.01** (note the minus sign), ensure that the **Operation** is set to **Join**, and click **OK**
As shown below the material should extrude downward. If it extrudes upward, change the sign of the Distance value.


- click on the **eye** icon for the **BrimBlock** Sketch to make it visible again and zoom into the other end of the body
- select the **Extrude** tool again and extrude the region by a distance value of -**0.01** (note the minus sign) as was done on the opposite end. Click **OK**.
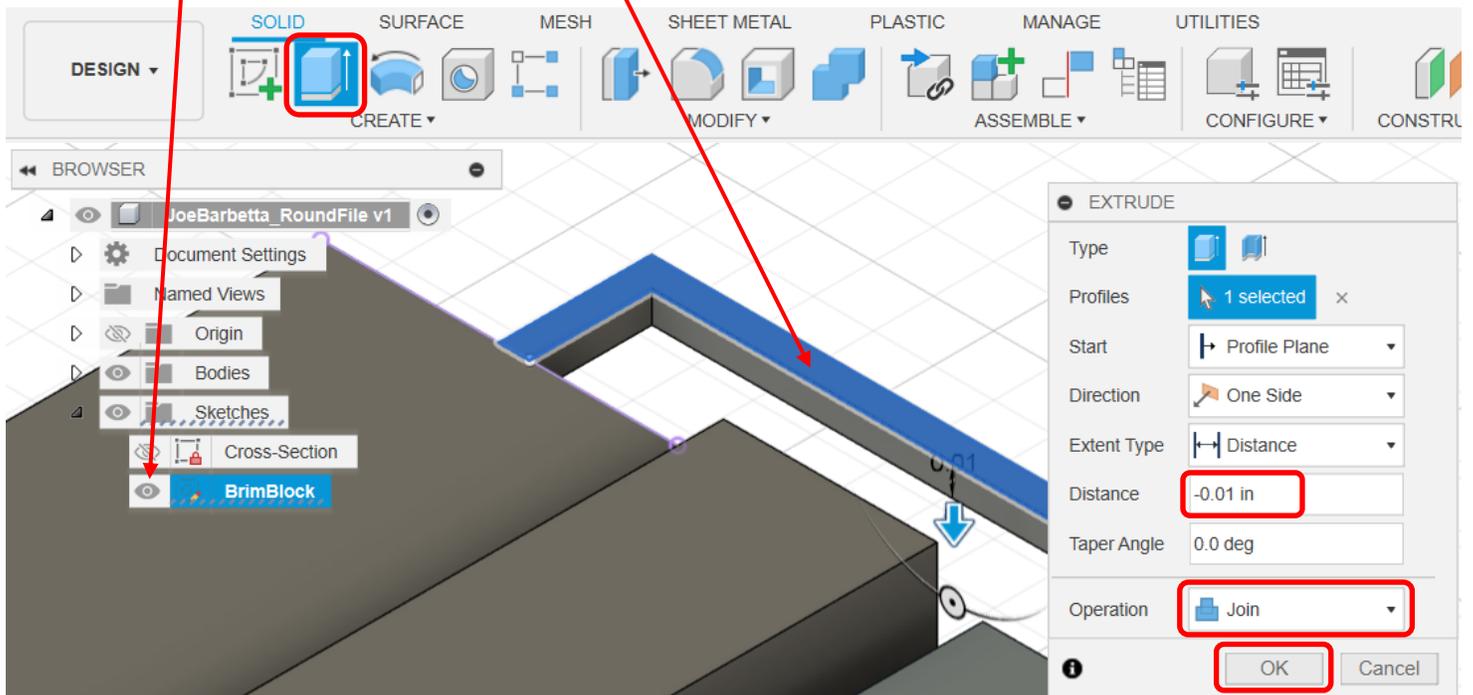
- click on **eye** icon again to hide the Sketch
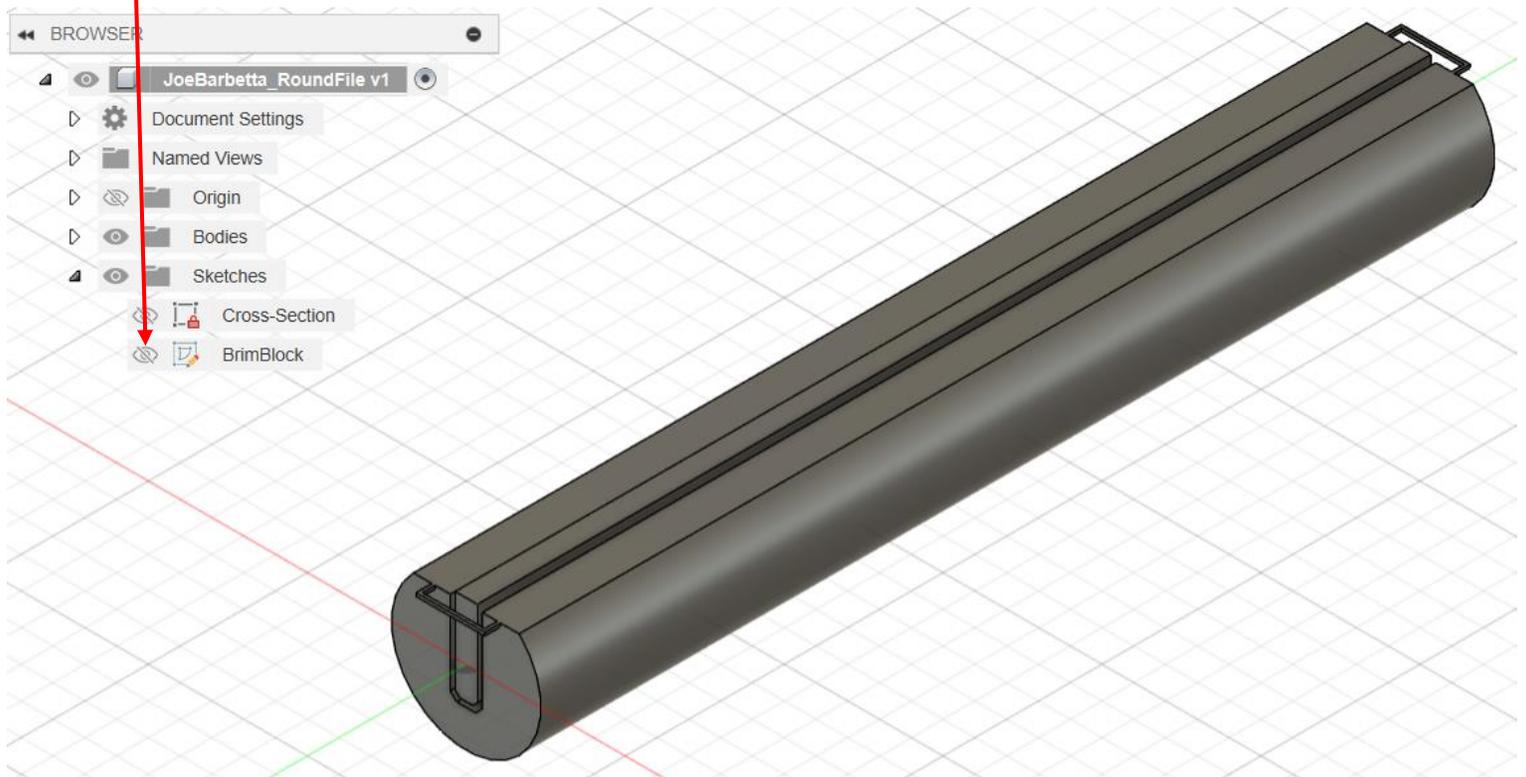
The purpose of the Brim Block that was created at each end is to prevent Brim material from being printed inside the slot. The bottom view shows the file parts in Cura in the Preview mode after slicing. The blue material on the build plate is a single layer of plastic that is printed around parts to help them adhere to the build plate. After printing the brim material is peeled off of the prints. It can be difficult to remove brim material from inside the slot and the Brim Blocks will prevent Cura from adding brim material in the slot. The Brim Blocks will just peel away with the brim material.
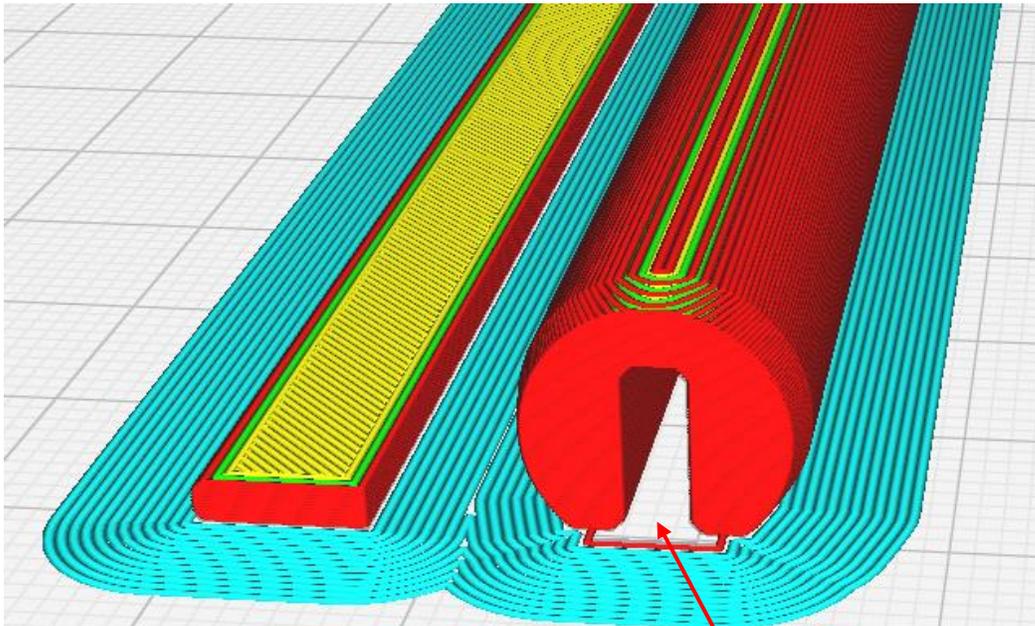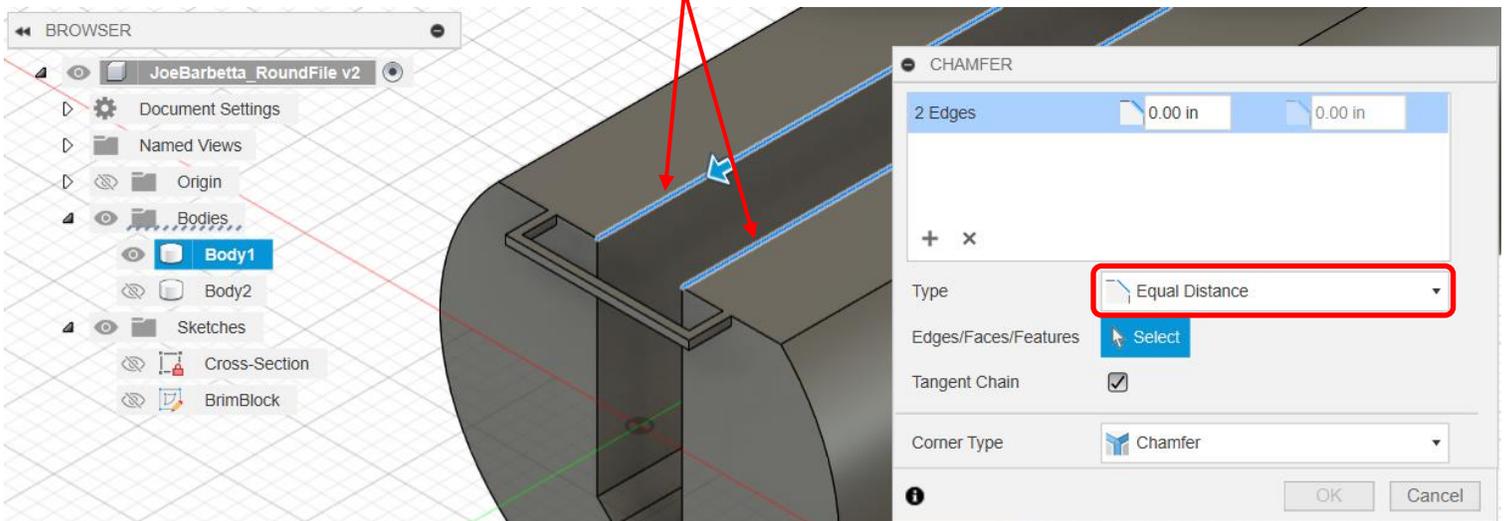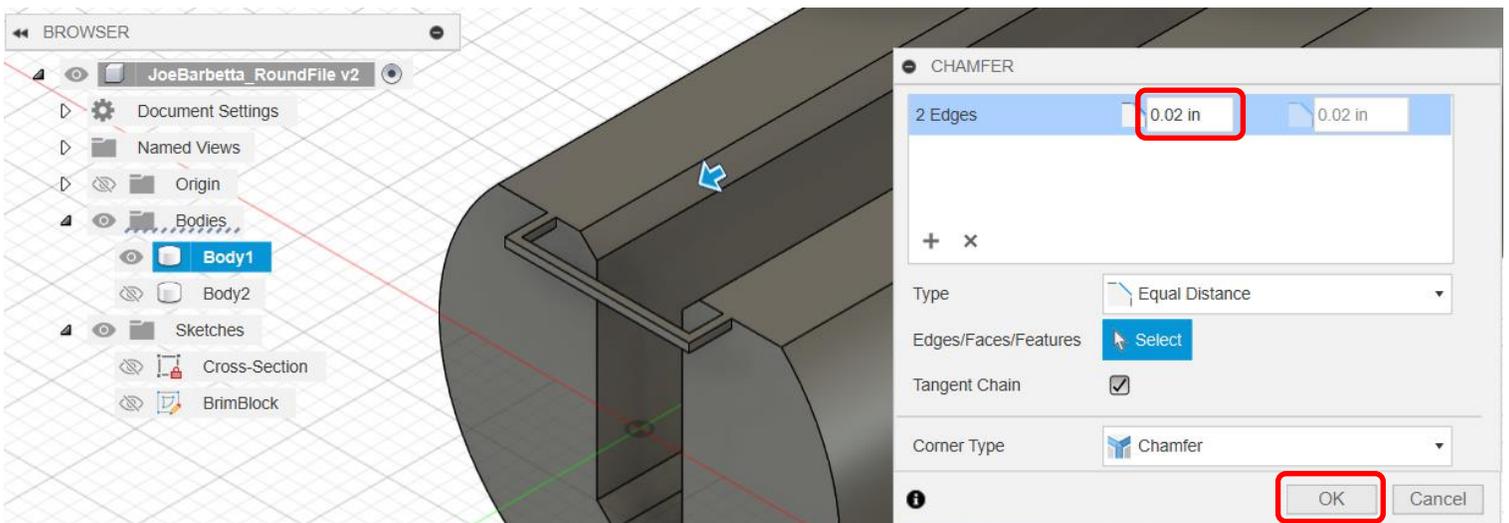

No brim material in slot

A Few More Tweaks

- zoom into one end of the body
- click the **arrow** for the **Bodies** folder and then click on the **eye** icon for **Body2**, which will hide the inner wedge
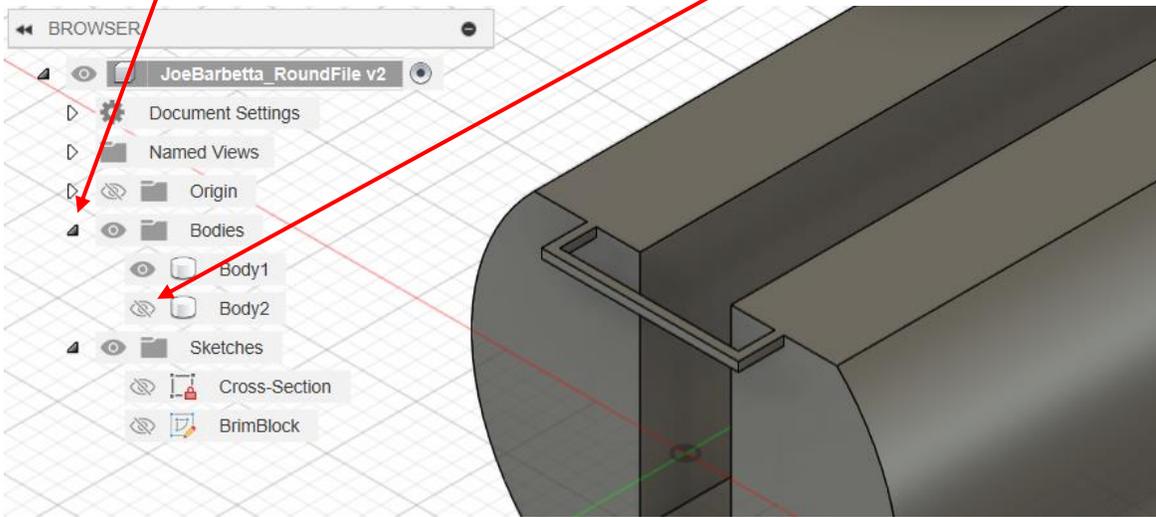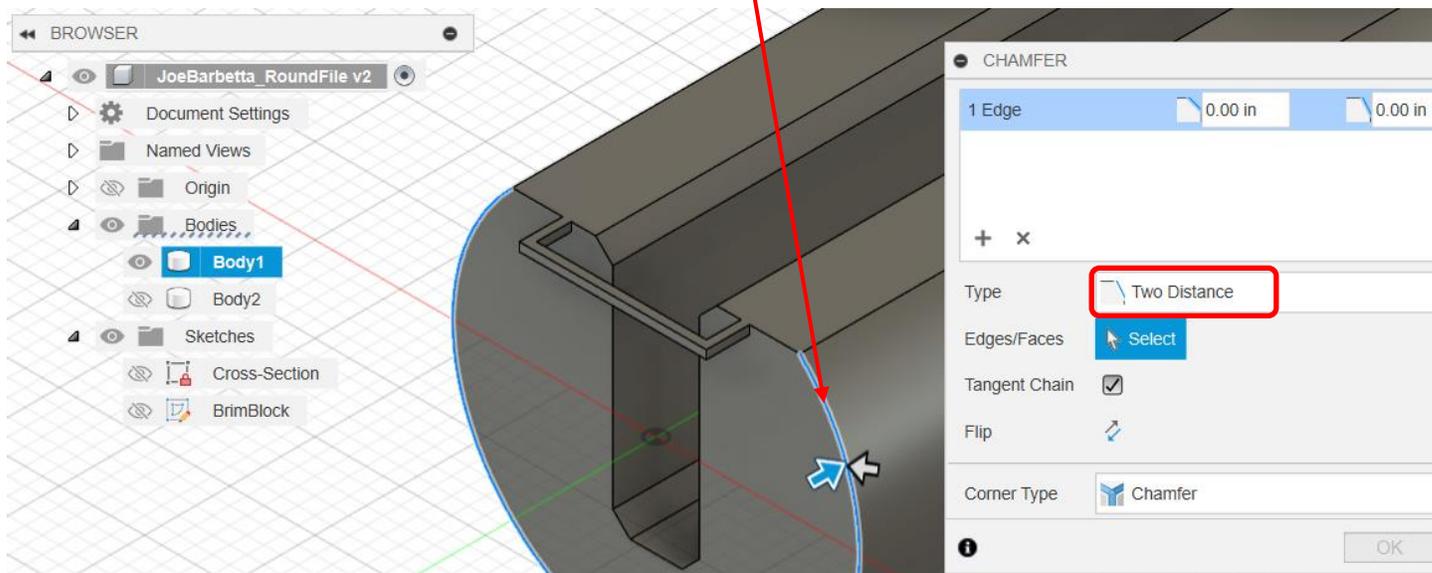

- from the **MODIFY** menu, select **Chamfer**
- ensure **Type** is set to **Equal Distance** and click on the **two slot edges**
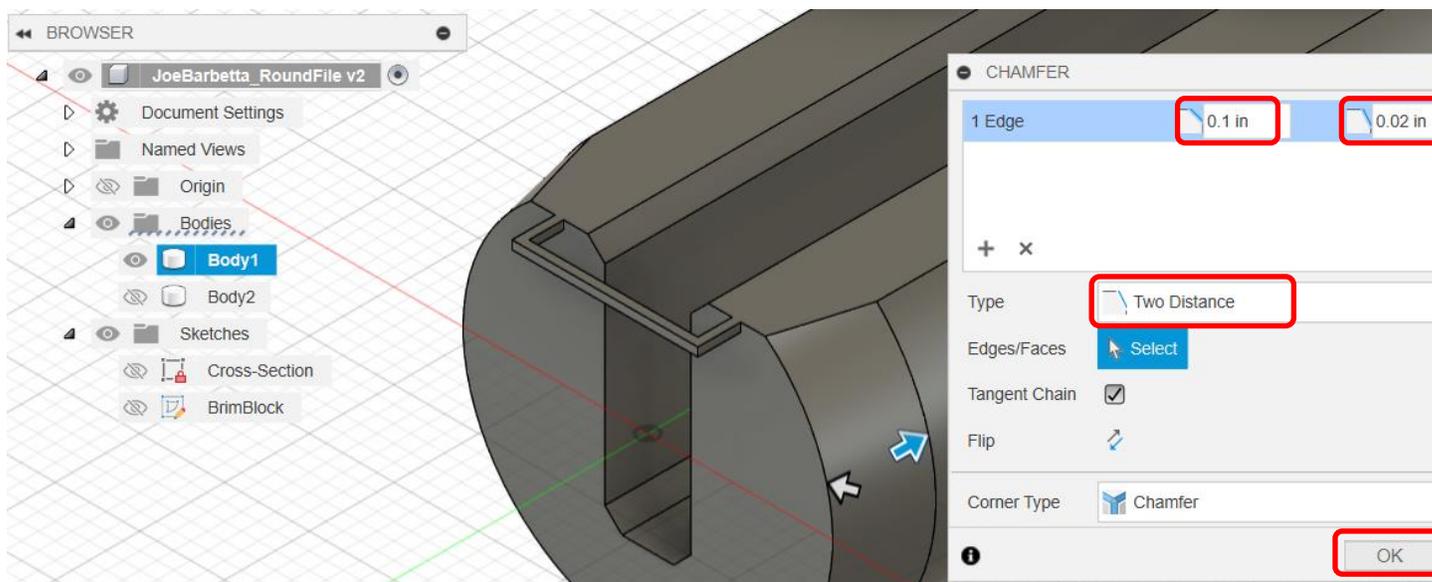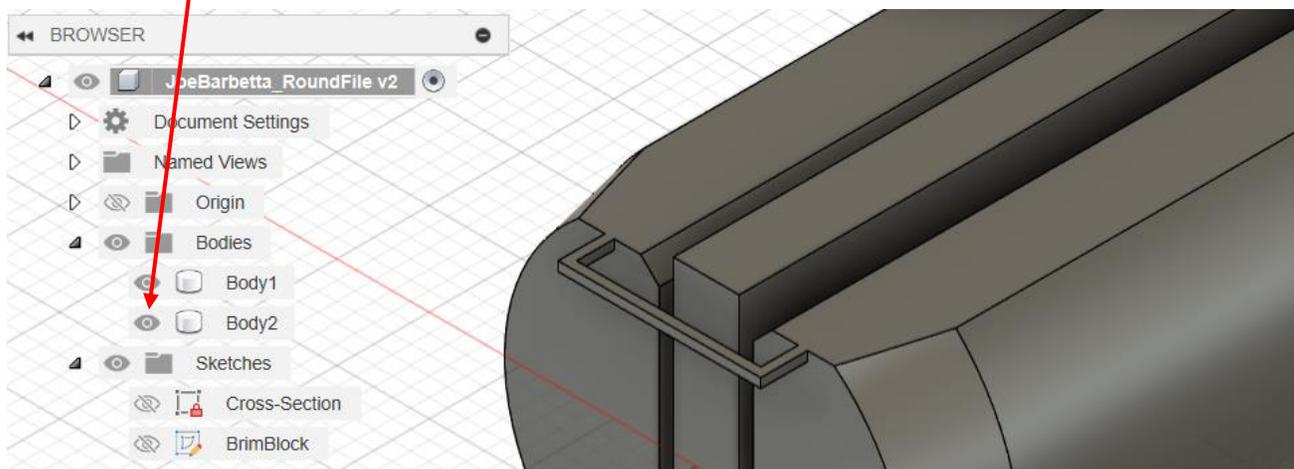

- enter **0.02** and click **OK**

- from the **MODIFY** menu, select **Chamfer** again
- set the **Type** to **Two Distance** and then click on the **circular edge**

- set the **right** value to **0.02** and then the **left** value to **0.1** in that order and click **OK**

- click on the **eye** icon for **Body2** to make it visible again

Changing User Parameters

The round file can be tailored to accommodate different diameters and sandpaper thicknesses by changing the User Parameters that were first set at the start of the project. The initial parameters work well to create a file to fit in a 1/2 hole and hold 180 grit sandpaper that it 0.015 thick.

- from the **MODIFY** menu, select **Change Parameters** at the bottom of the menu
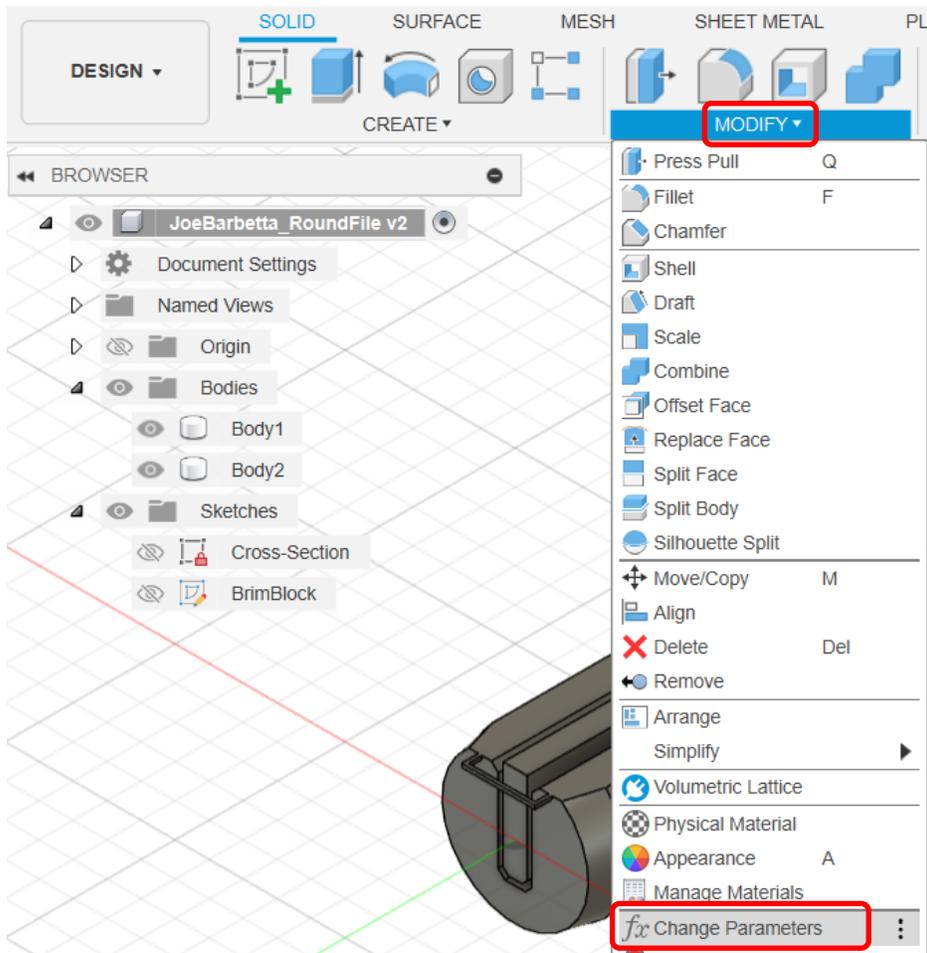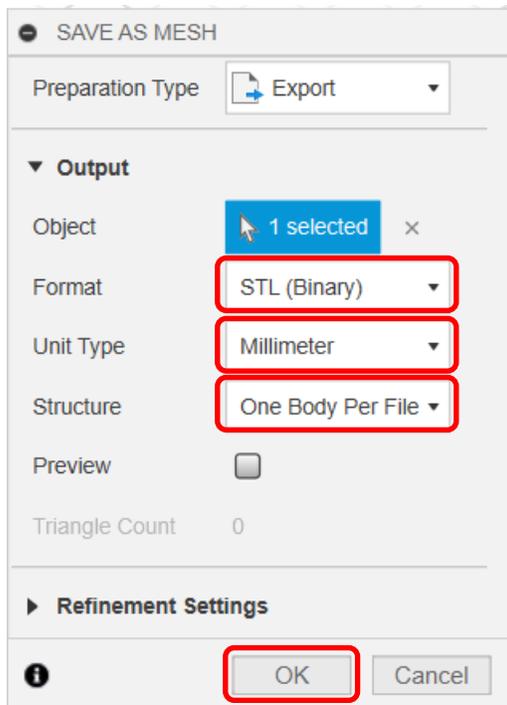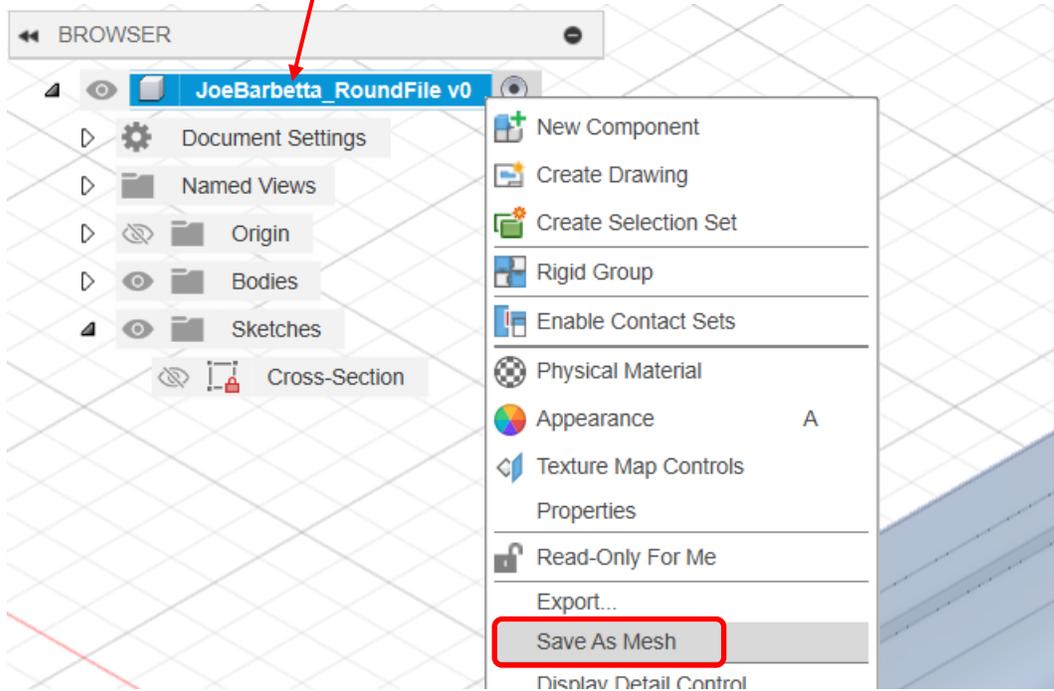

- click on the **Expression** for **TaperAngle** and change the value to **1**, which should cause the far end to expand.
- set the value back to **0.01** and close the PARAMETERS window
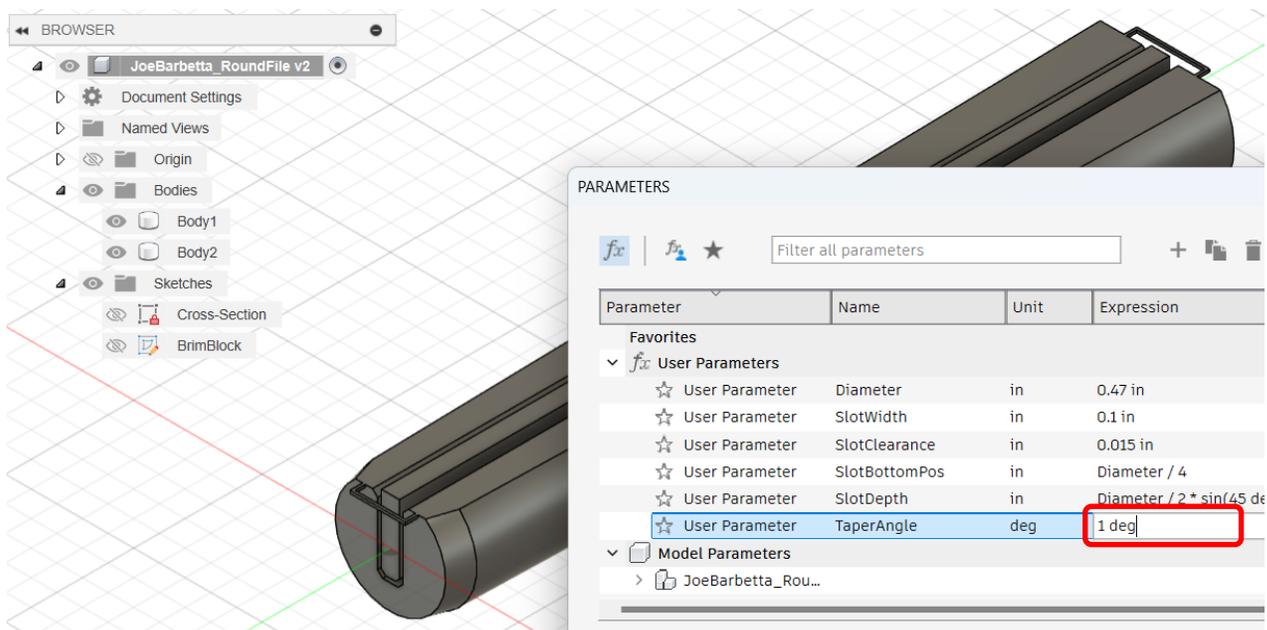
Creating STL files for printing

- right click on the **Project Name** and select **Save As Mesh**

- ensure that **Save to my computer** is checked
- make not of the save location. By default it should be the **Downloads** folder.
- click **Save**
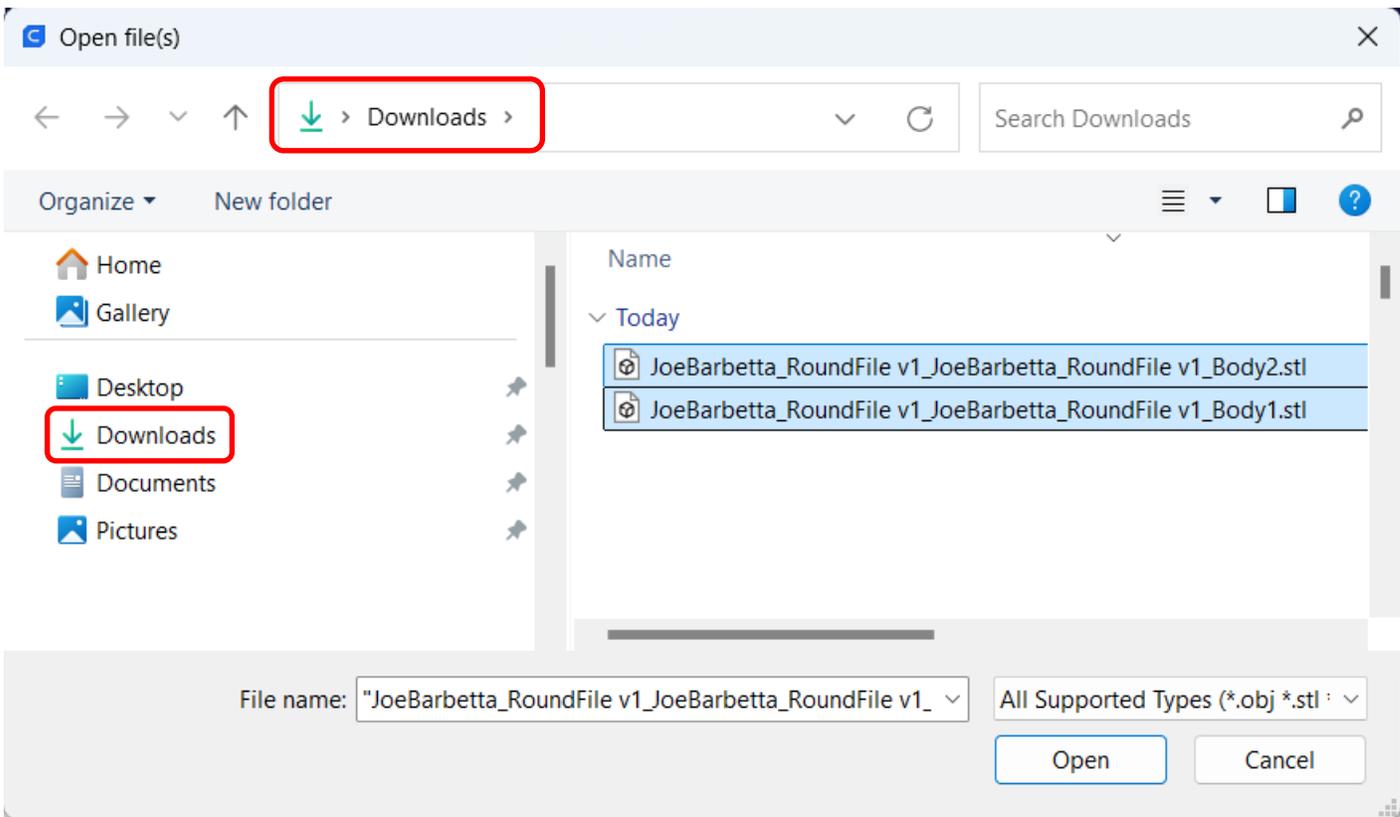
Opening the STL Files in the Cura Slicer Program

- run **Cura**
- from the top **File** menu, select **Open File(s)…**
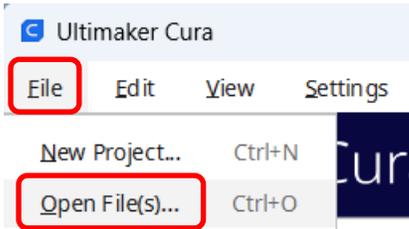
- if the present folder is not **Downloads**, select **Downloads** on the left section
- click on the **2 files** with **RoundFile** in the name. If there is only 1 file, revisit the Create STL instructions. The **Structure** must be set to **One Body Per File**.

The 2 bodies will show in Cura.

In the picture here the body was zoomed into. The colors may be different depending on Cura settings.
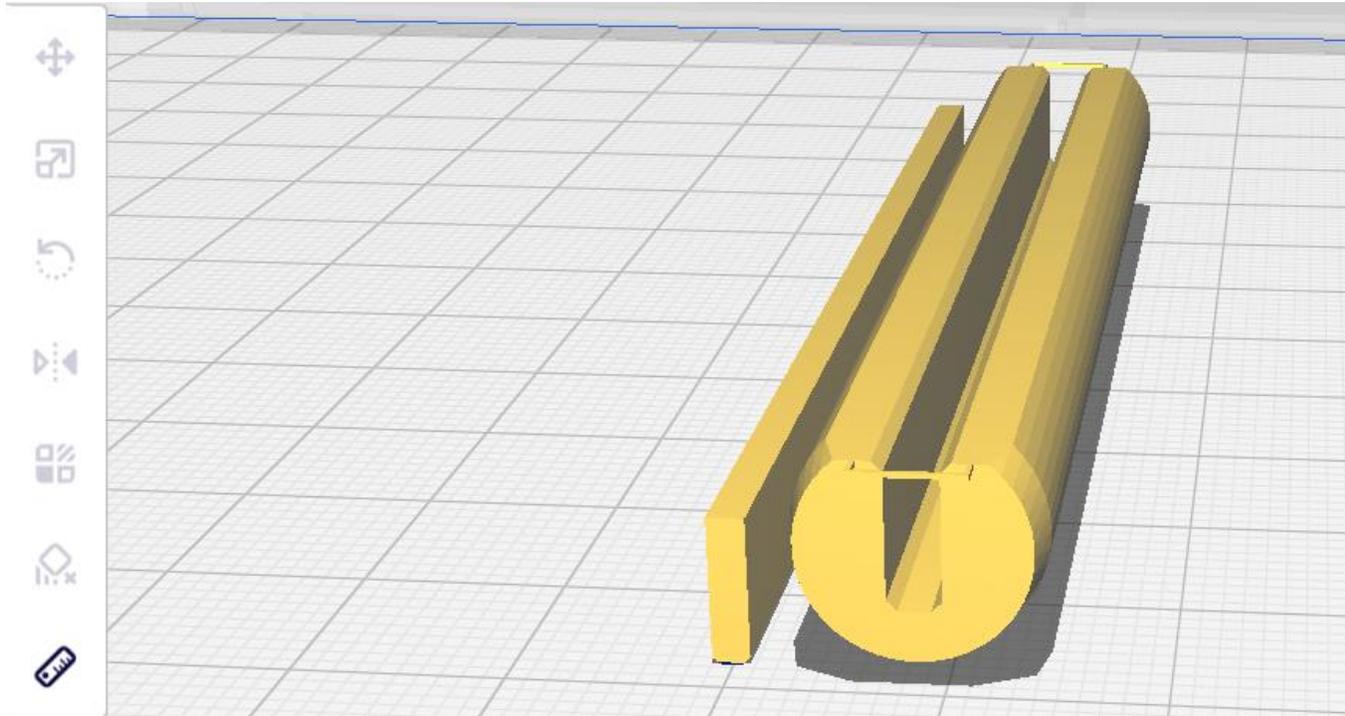

- click on the **cylindrical body** to select it
- select the **Rotate** tool and then select the **"Select face to align to the build plate"** icon
- click on the **top flat surface** of the cylinder, which should flip it over
Note that an alternative method is to use the arrows of the rotation circles.
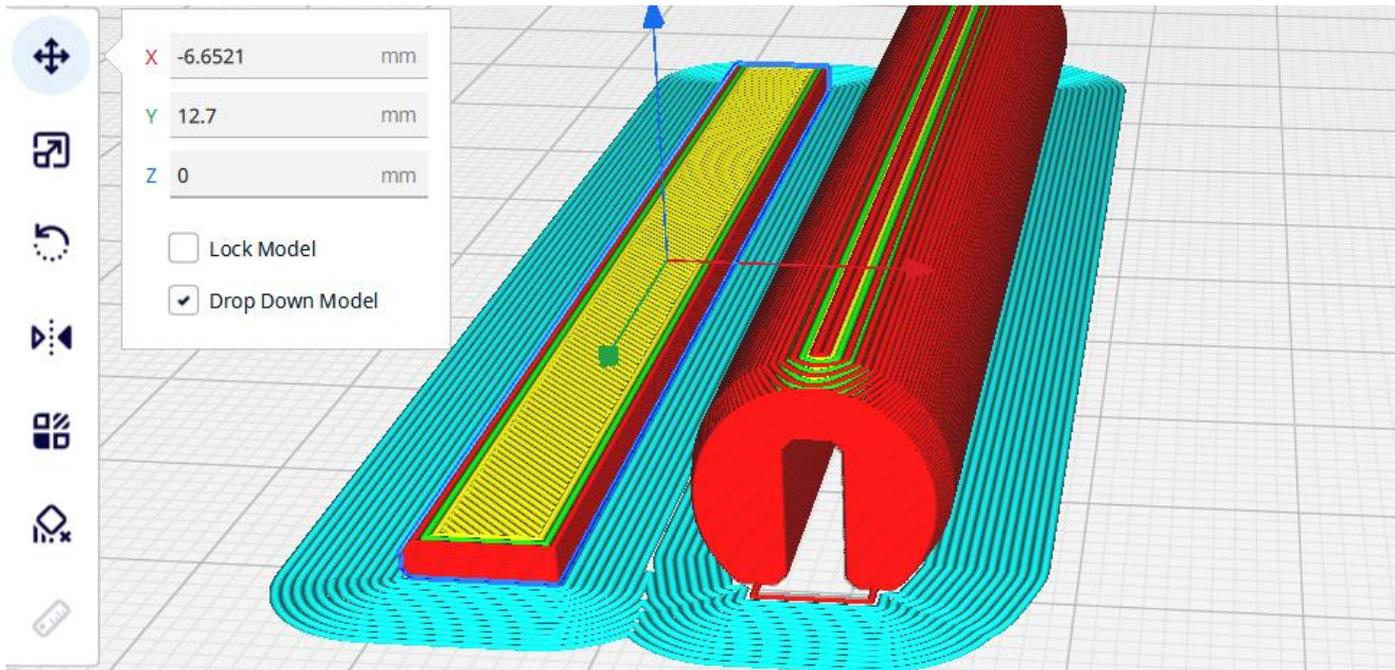

Snap Rotation
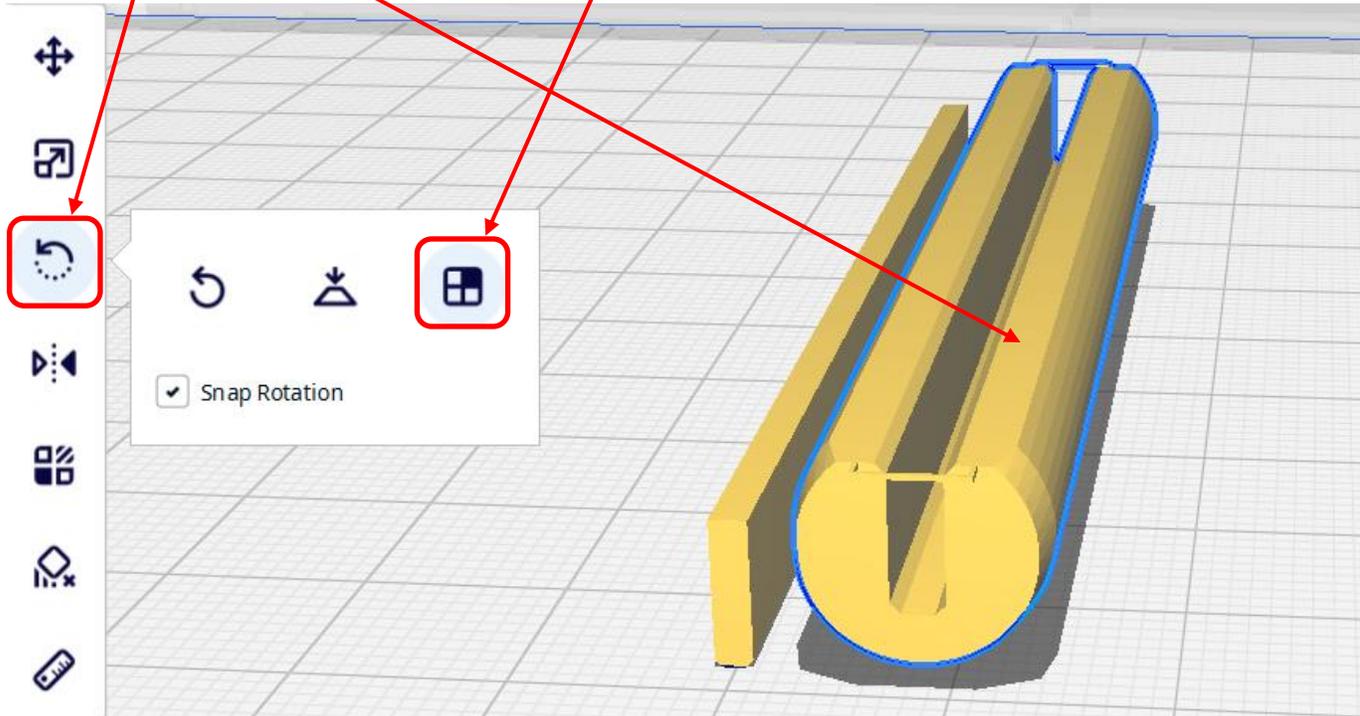
- click on the **slot wedge** body
- select the **Rotate** tool and then select the **"Select face to align to the build plate"** icon
- click on the **side surface** of the **wedge**, which should flip it over

Snap Rotation
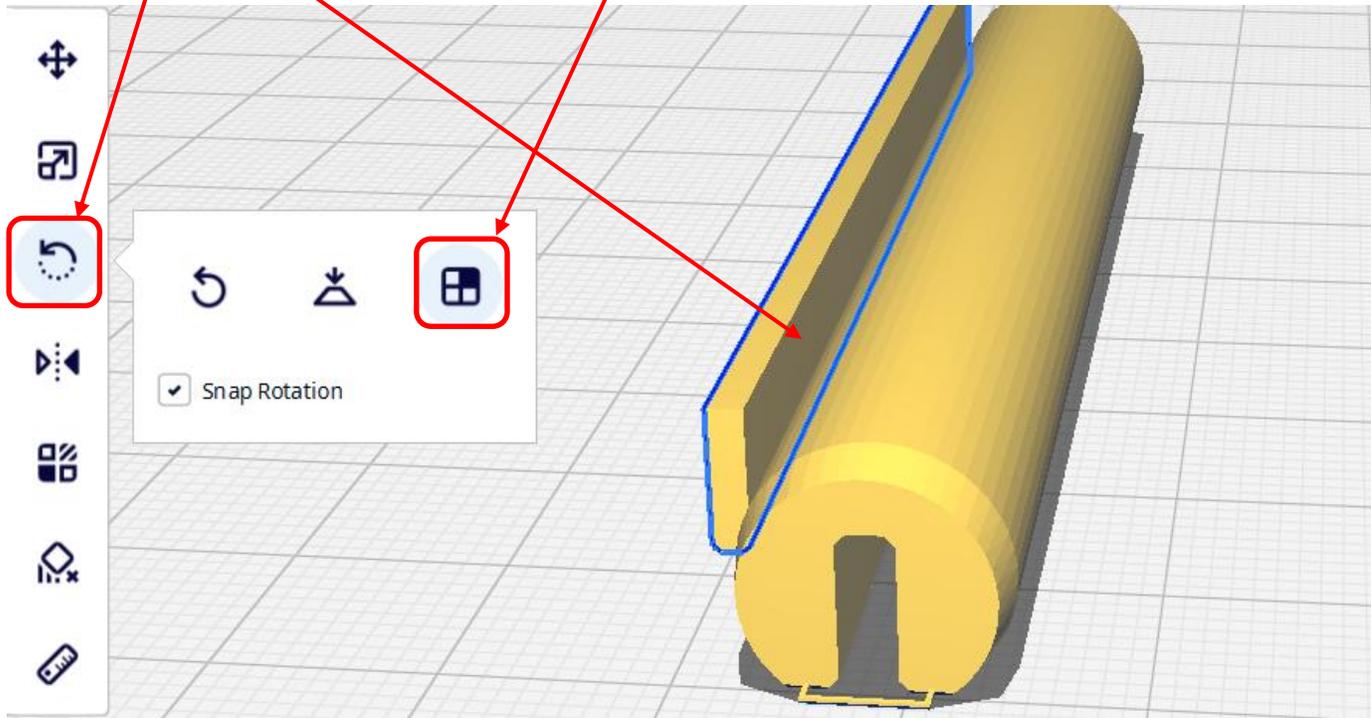
- select the **Move** tool and move the **wedge** to the left, as shown below

X -6.6521 mm

Y 12.7 mm

Z 0 mm

Lock Model

Drop Down Model
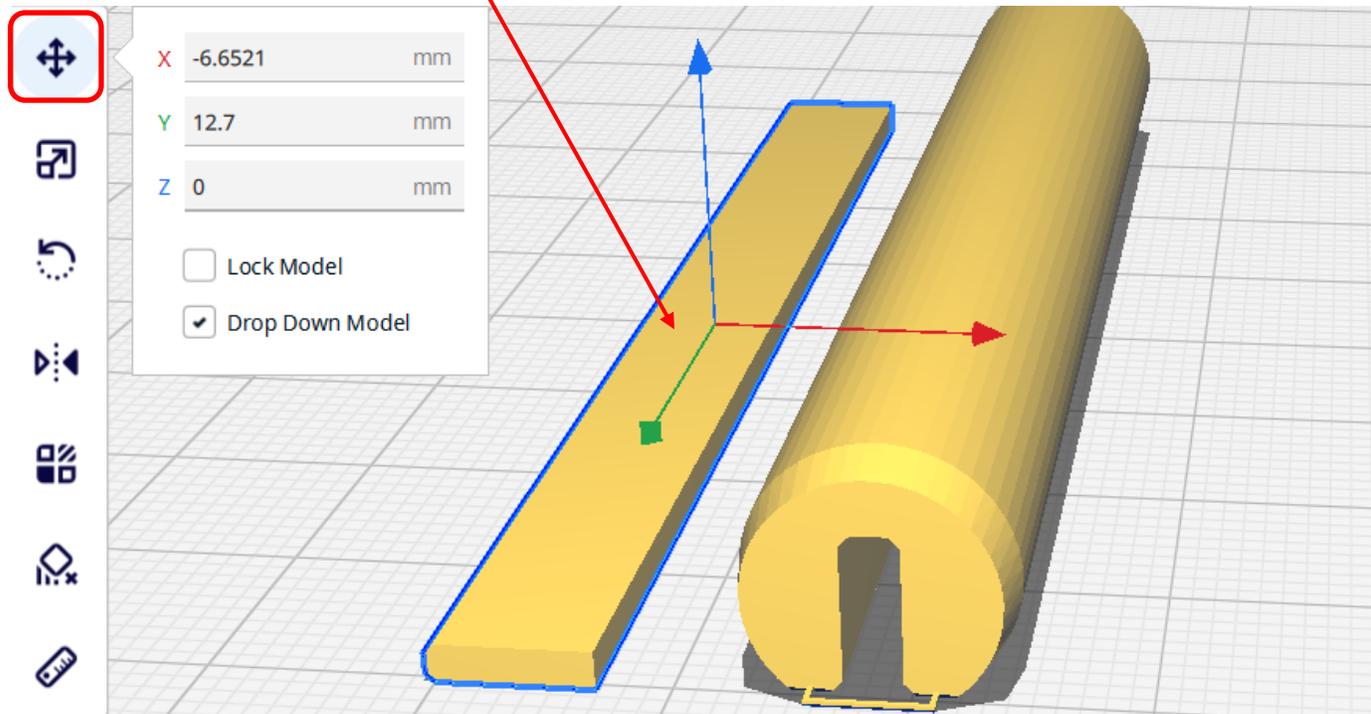
Assembling the File

A piece of sandpaper can be cut to the proper size to cover the cylinder and have extra material to fit in the slot. The wedge can then be pushed into the slot to secure the sandpaper. The sandpaper used here is 3M Pro Grade Precision 180 Grit, which is about 0.015" thick. If it is too difficult to insert the wedge, the SlotClearance parameter can be increased. In this case a vise was used to help push the wedge it.

Do Not use your best scissors to cut sandpaper! Use scissors you do not care about. The sandpaper will dull the scissors.