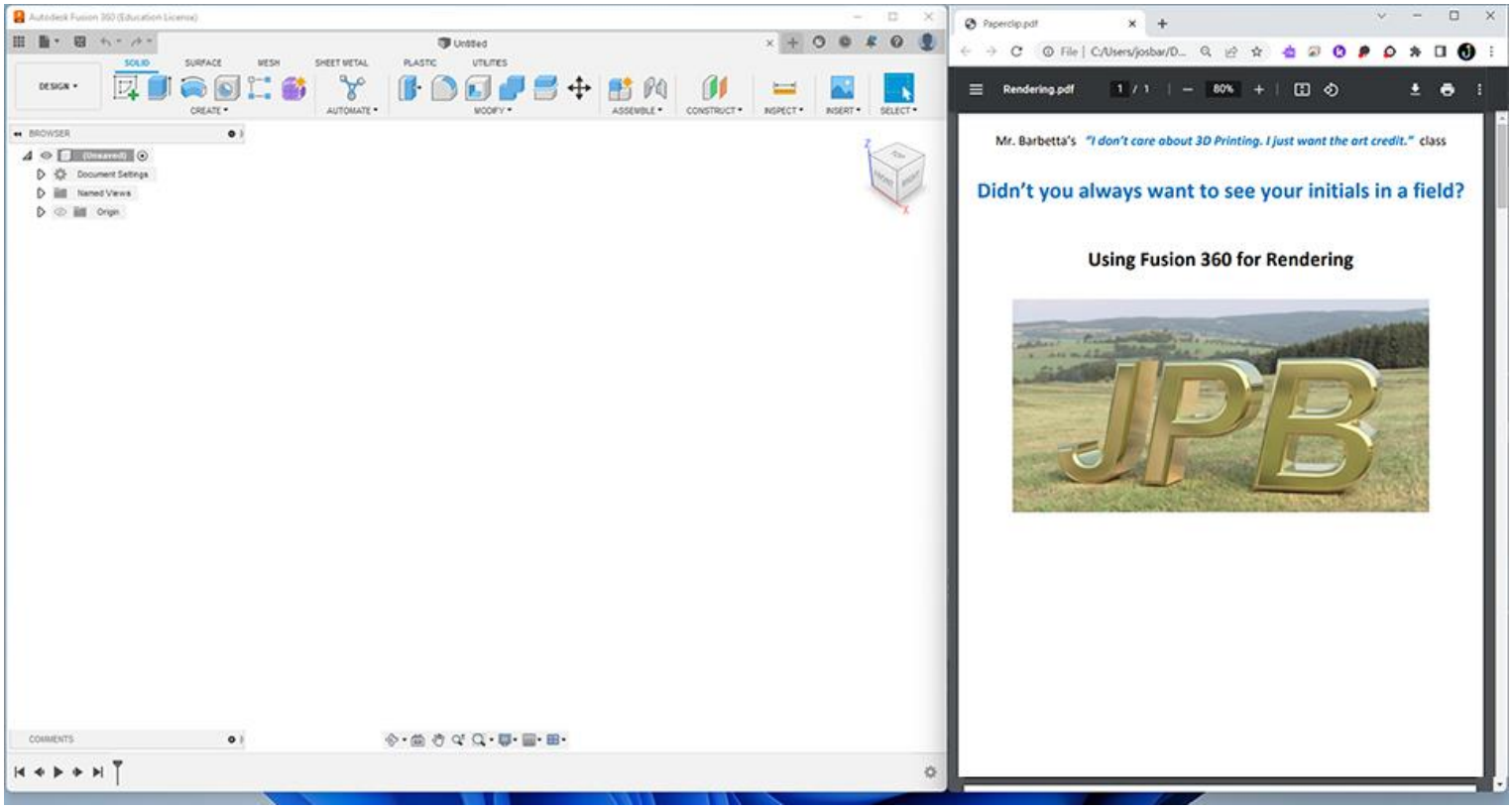
Mr. Barbetta's ***"I don't care about 3D Printing. I just want the art credit."*** class

Want some Pencil Bling?

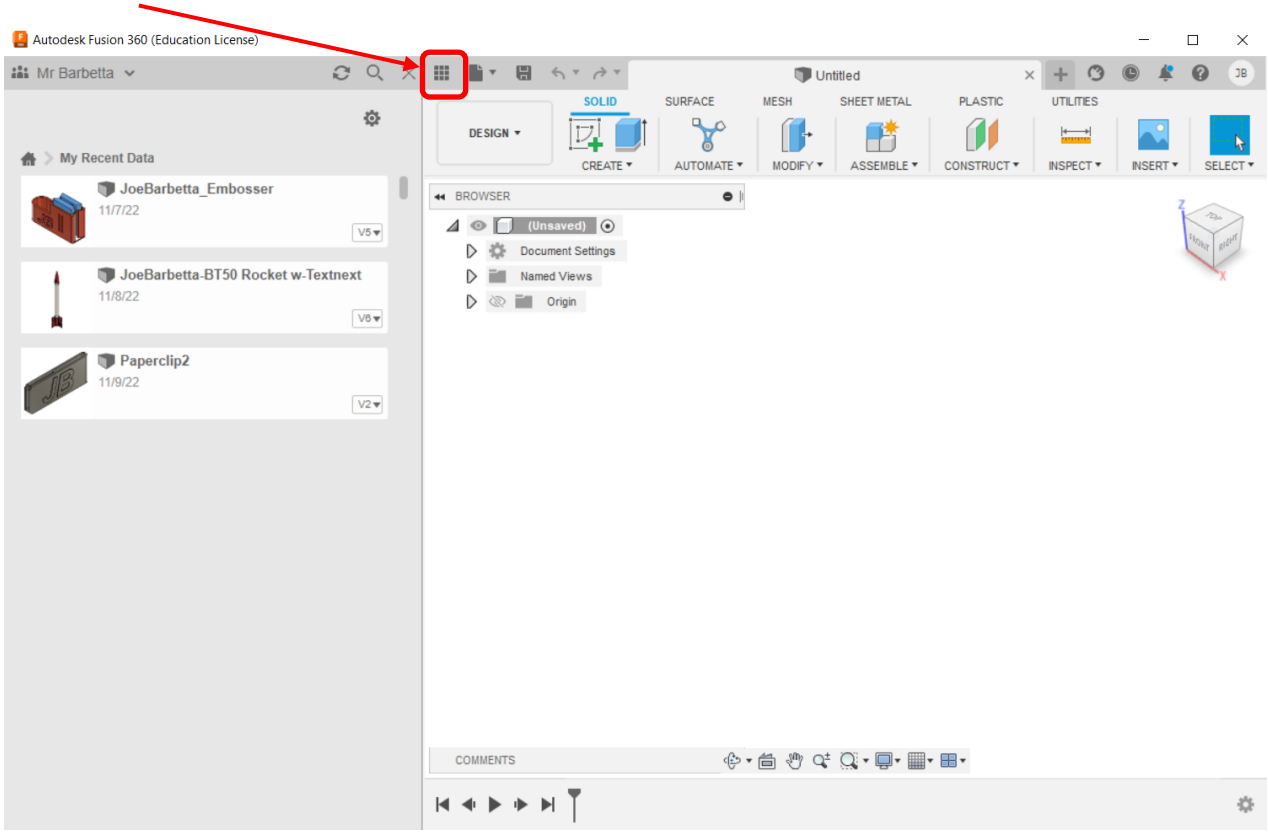
Today's lesson is sponsored by BIG SNOW and the American Dream mall.

Using This Document

The best way to follow this document is to **reduce the width of the Fusion 360 window** and have this pdf document open in Chrome browser as shown below. This document can be **downloaded from Schoology and then dragged into Chrome** and scaled down to 80%.



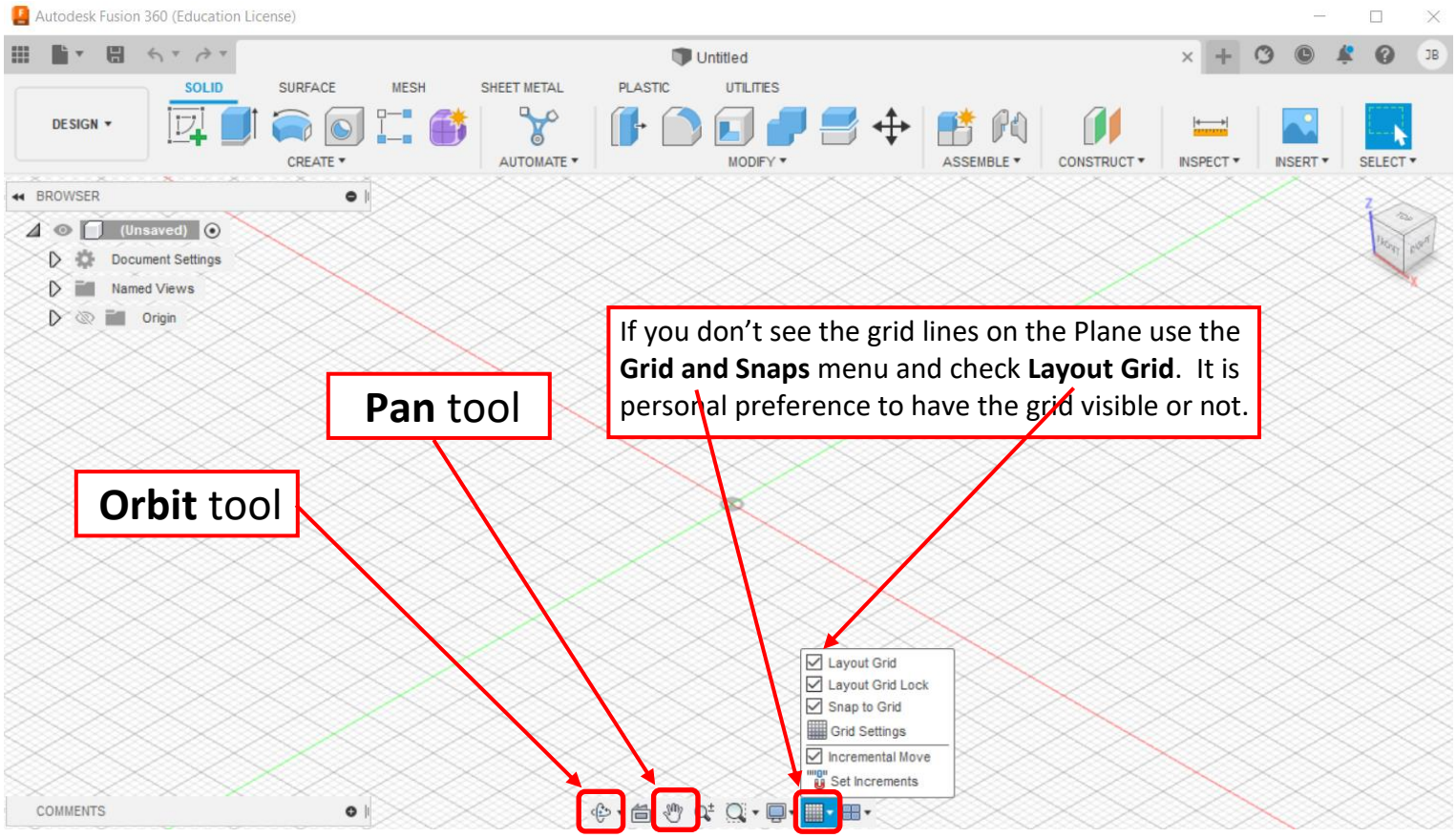
The Fusion 360 window will not allow its width to be reduced much so for smaller computer screens a trick is to click on the **Data Panel icon** and then move the window to the left with the Data Panel off the screen.



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

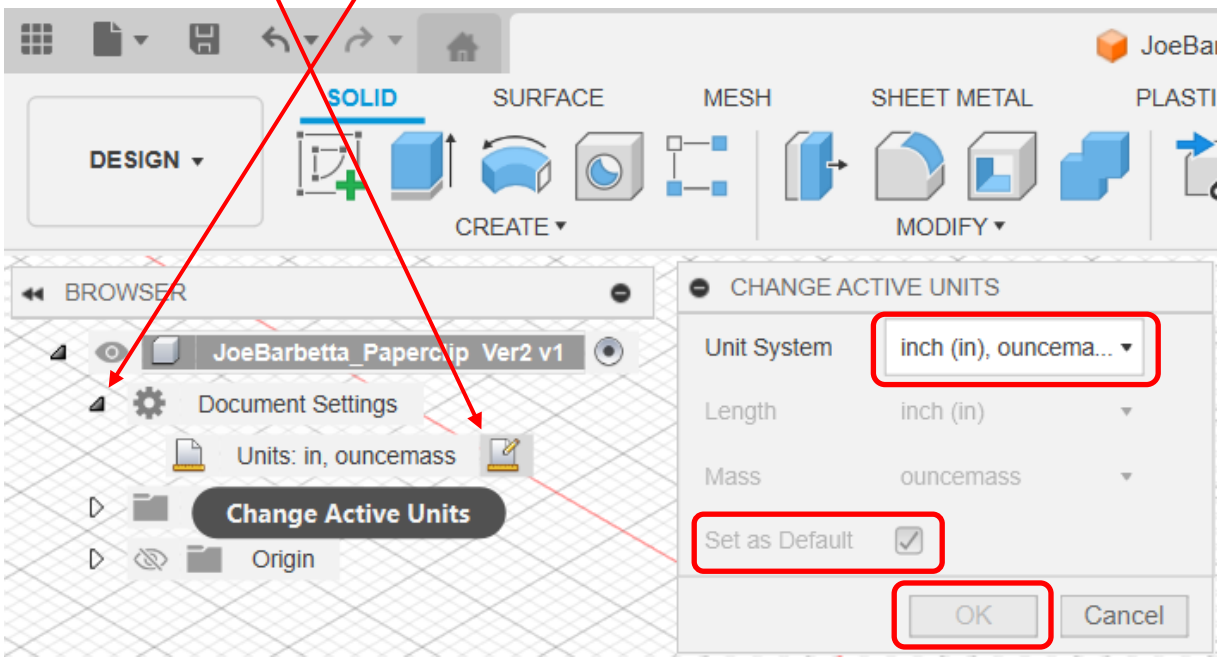
Starting a Design in Fusion (START HERE)

- 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 **_RingyThingy** e.g. **JoeBarbetta_ RingyThingy** (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.



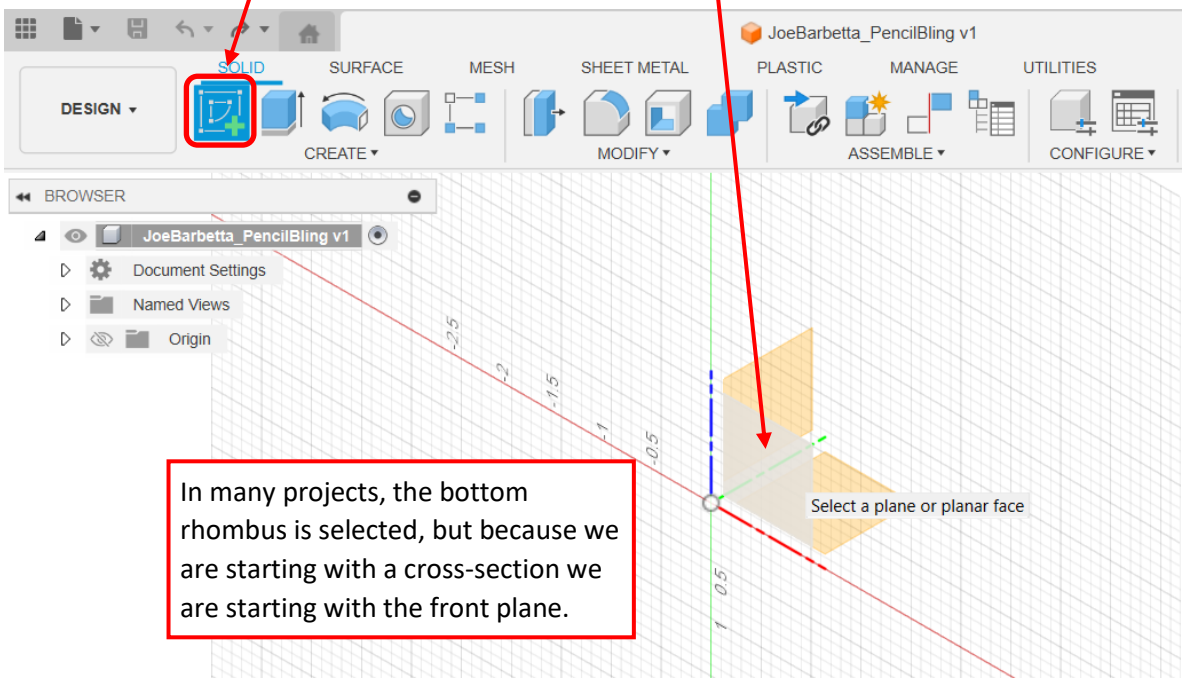
Note that the default units are in mm, which we just changed to inches.

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

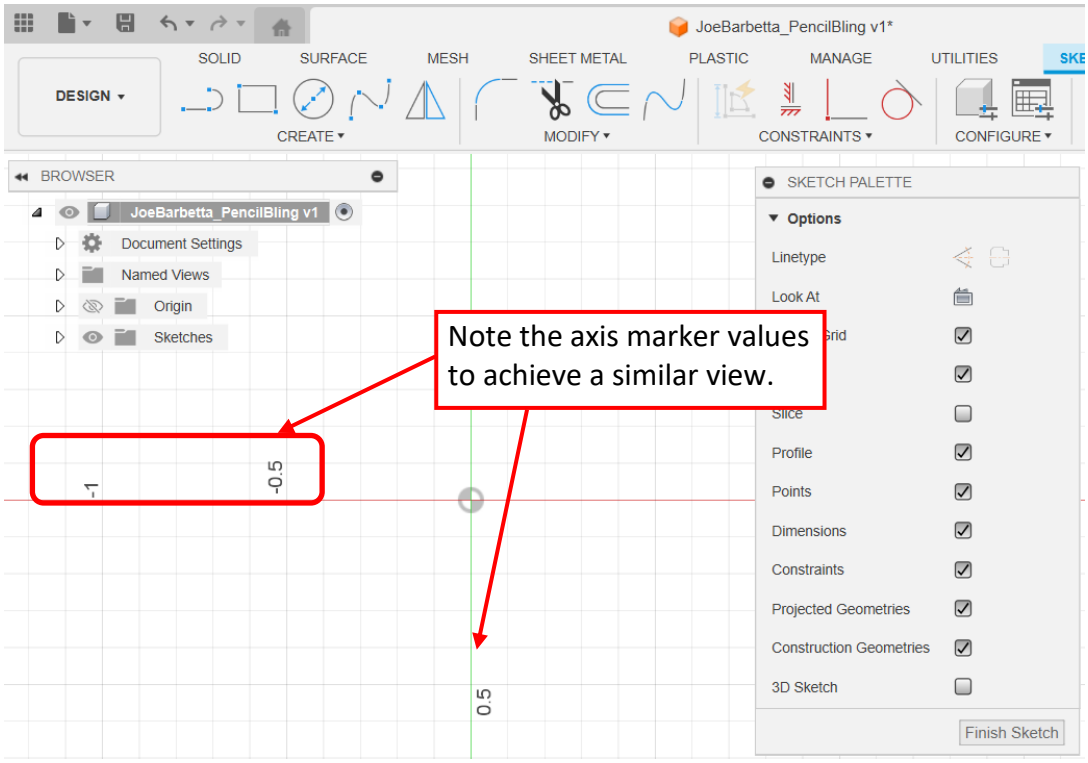
Creating the First Sketch

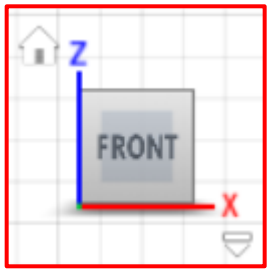
Note that a Fusion expert may tell you to create a Component first. Just say **"Dude. I'm just making some pencil bling."**

- select the top **Create Sketch** tool and click on the **front rhombus** to select the X-Z Plane.
If a tool can't be found, one can always look in the **CREATE** and **MODIFY** menus for it.



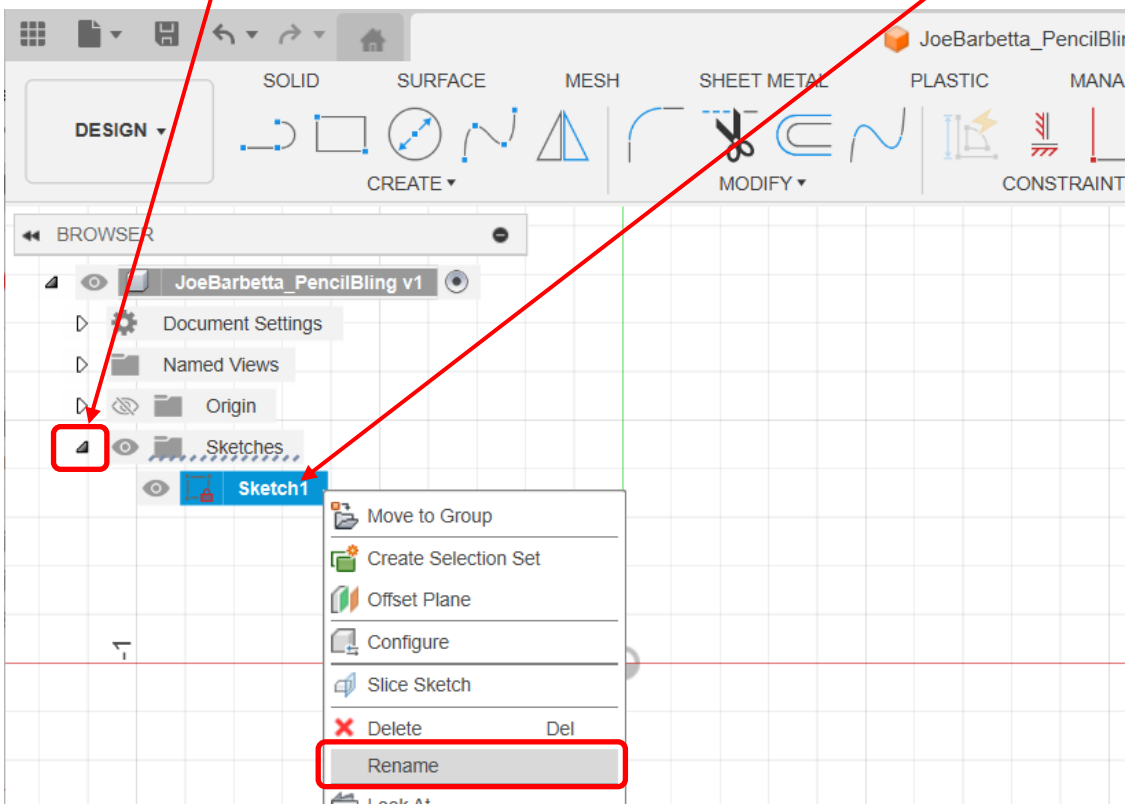
- zoom in as shown below. The scale labels can give an idea of how far one is zoomed in. The **View Cube** should indicate you are sketching on the **Front X-Z Plane**.





Whenever a new Sketch is created, it should be named.

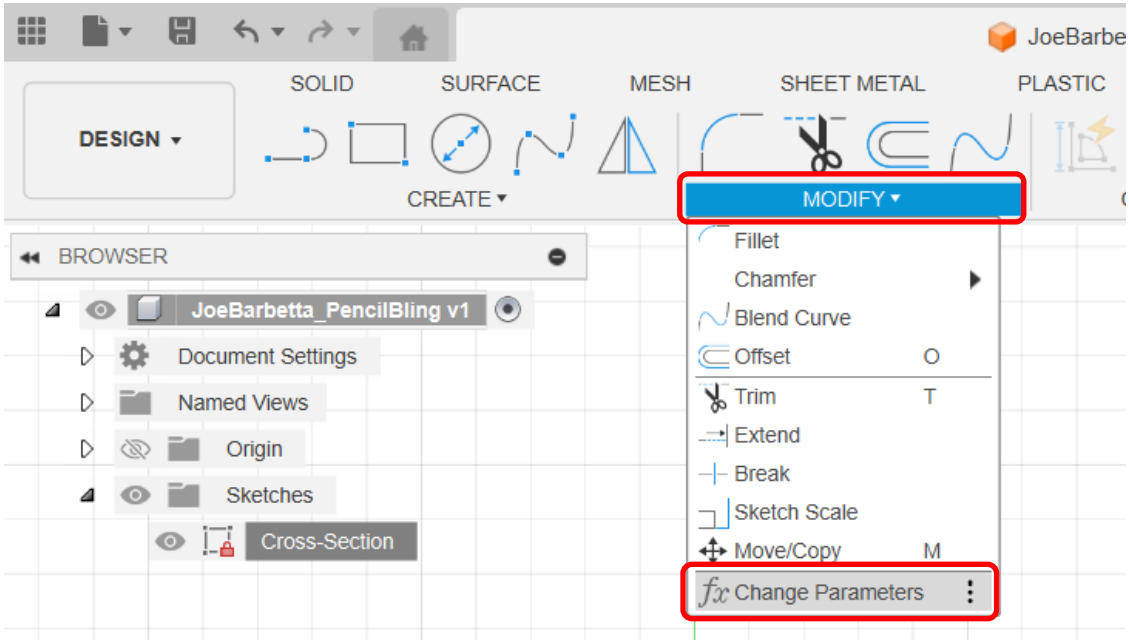
- click on the **arrow** to open the Sketches folder, **right-click** on the default name **Sketch1** and select **Rename**. Change the name to **Cross-Section**.



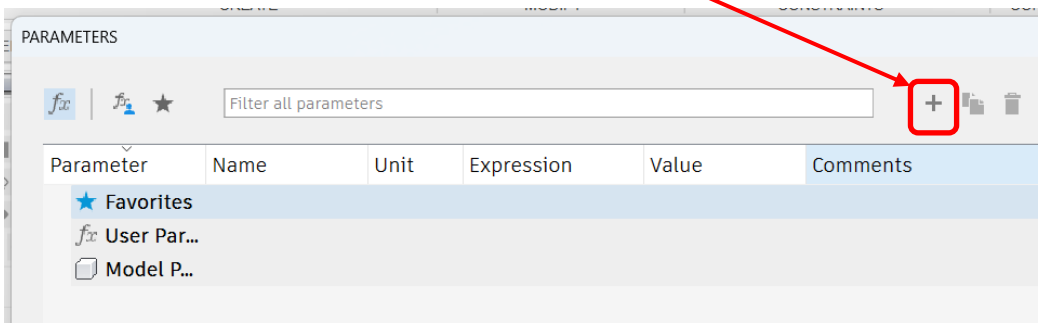
Parametric Modelling

"Parametric Modelling" is a powerful feature, wherein variables can be created, which can then be used for dimensions. One can later change values using this window to make adjustments to a design. We will set a parameter for the Hole Diameter to make it easier to adjust the hole diameter to best fit on a pencil.

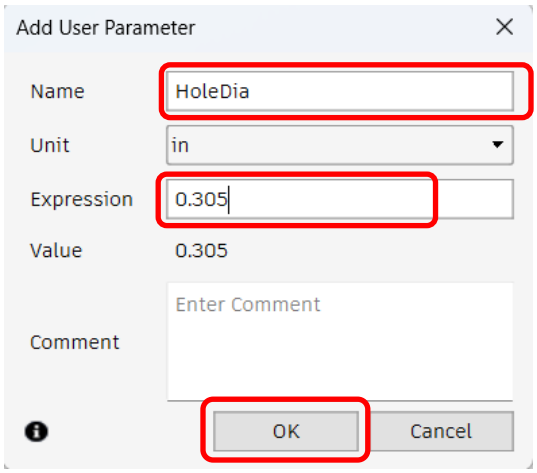
- from the **MODIFY** menu select **Change Parameters**



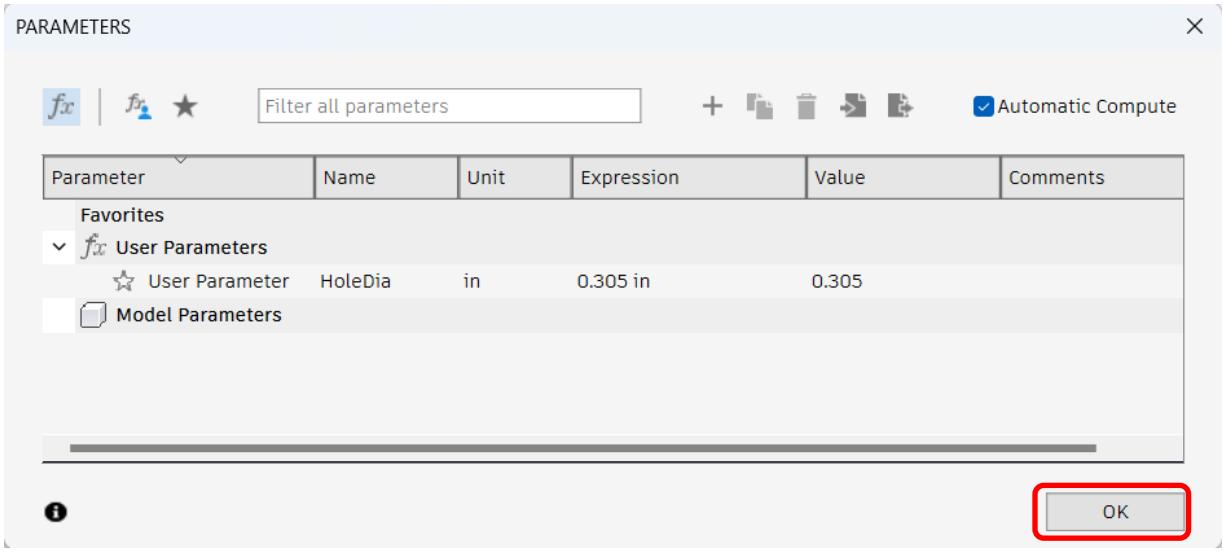
- in the **PARAMETERS** window that just opened, click **+**. If a window about **Parameteric Text** pops up click its **OK** button.



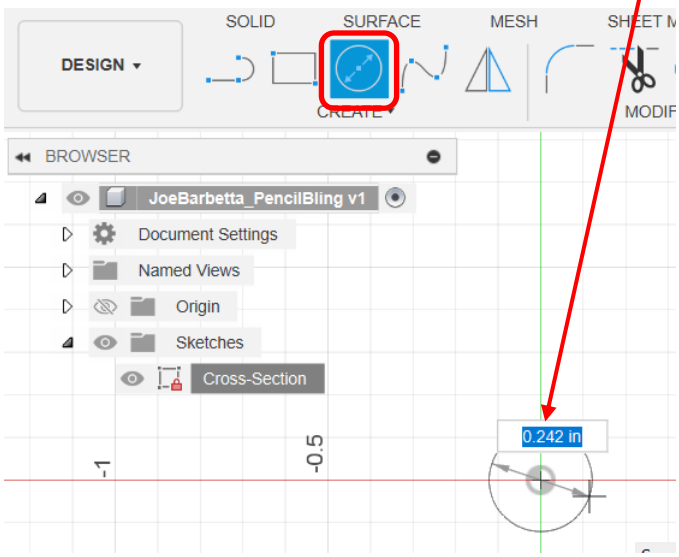
- In the pop-up window set the Name to **Hole** and Expression to **0.305**. Make sure that the Unit is **in** for inches and click **OK**.



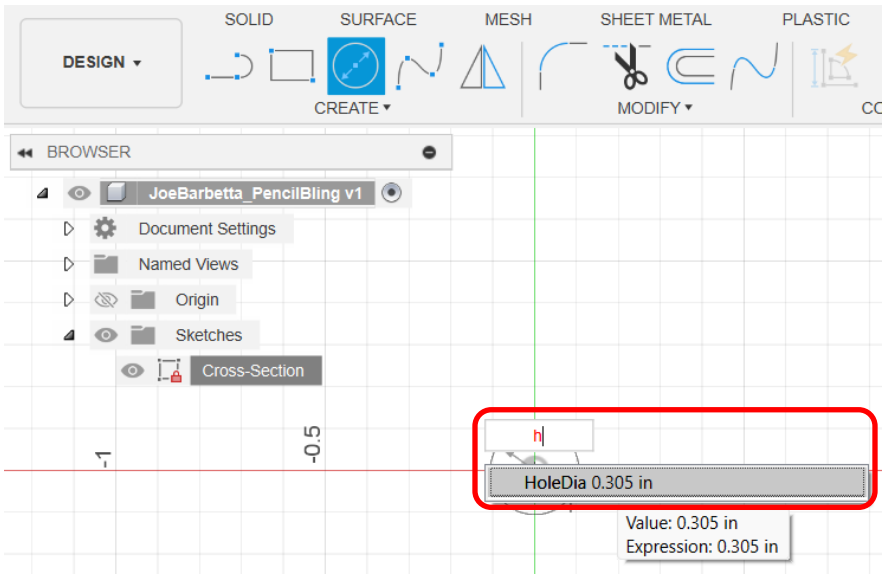
- click **OK** on the Parameters window



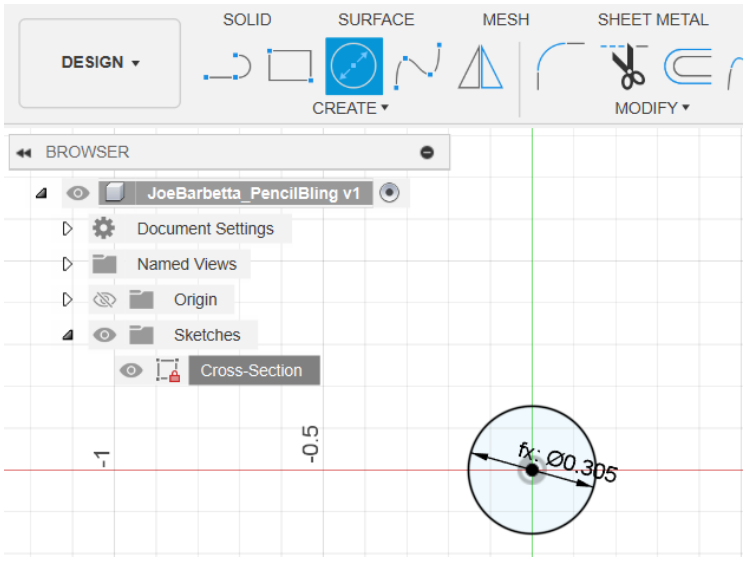
- select the **Center Diameter Circle** tool, click on the **origin**, entend the circle out, and type **h**, which is the first letter of the parameter that we set



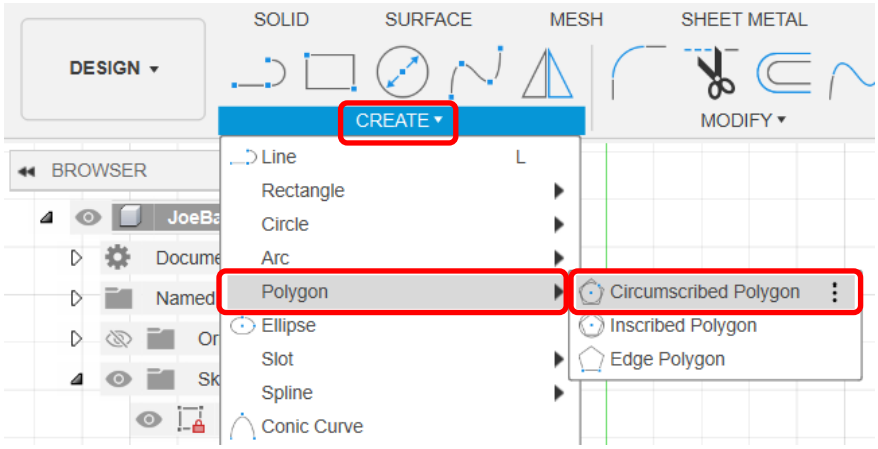
- click on the **HoleDia** option and press the **enter key**



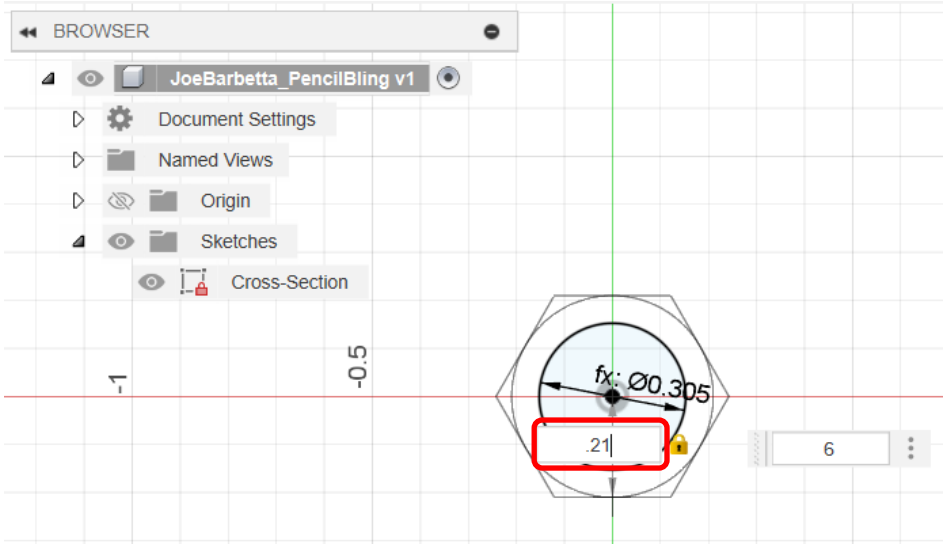
The circle should look like this. The dimension line (with the arrows and value may look different). It is indicating the diameter of the circle. Later on one can use the Parameters window to adjust this value.



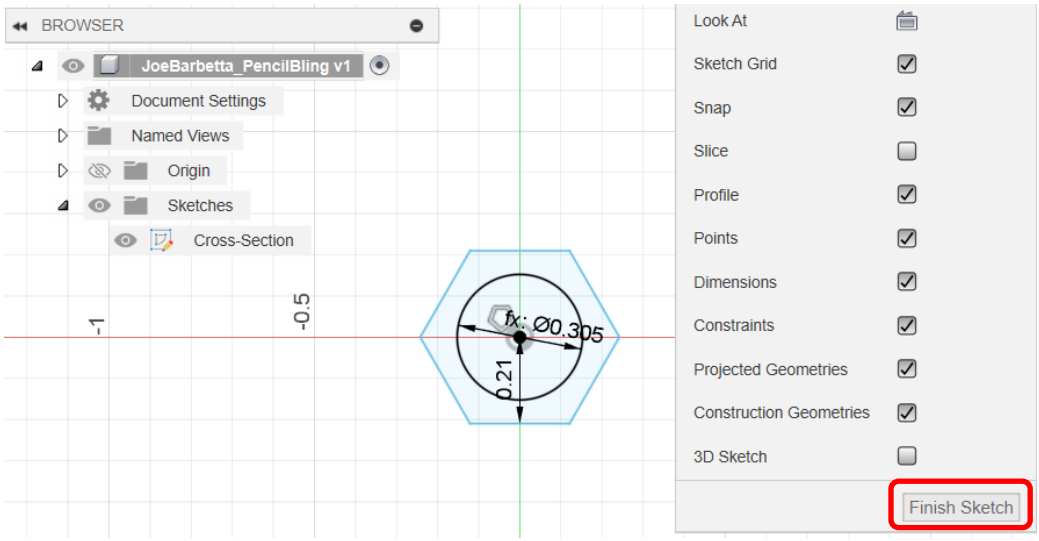
- from the **CREATE** menu select **Polygon** and **Circumscribed Polygon**



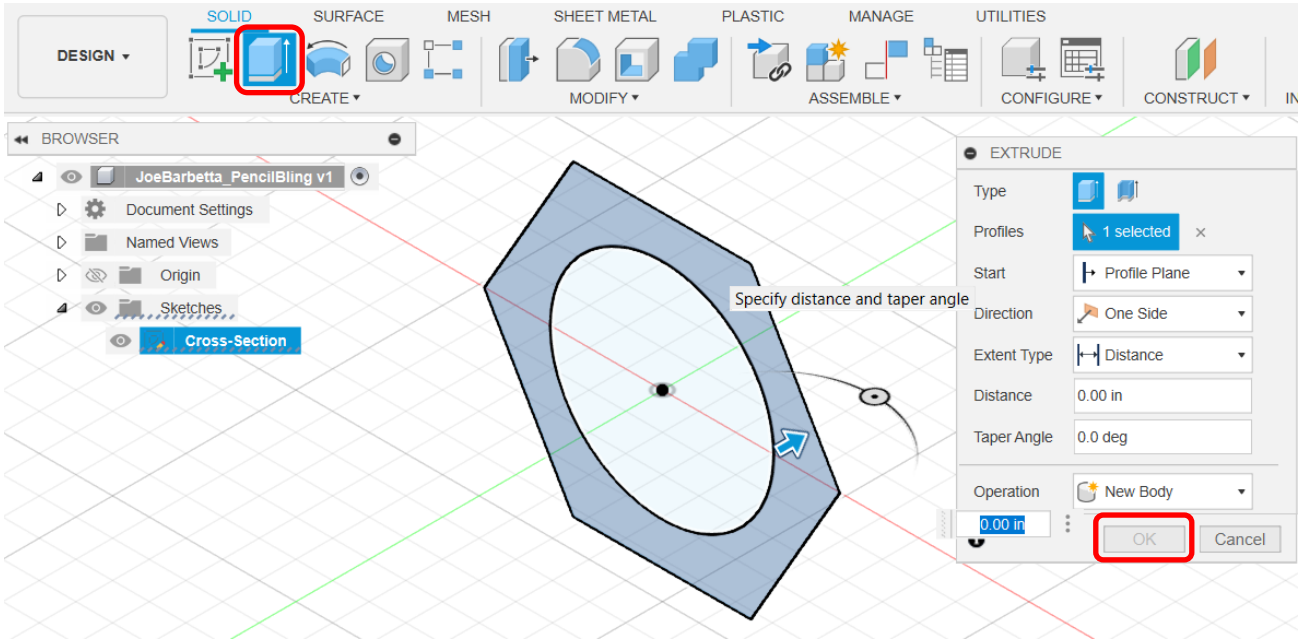
- click on the **origin**, **move the mouse down**, type **0.21**, and press the **Enter key**



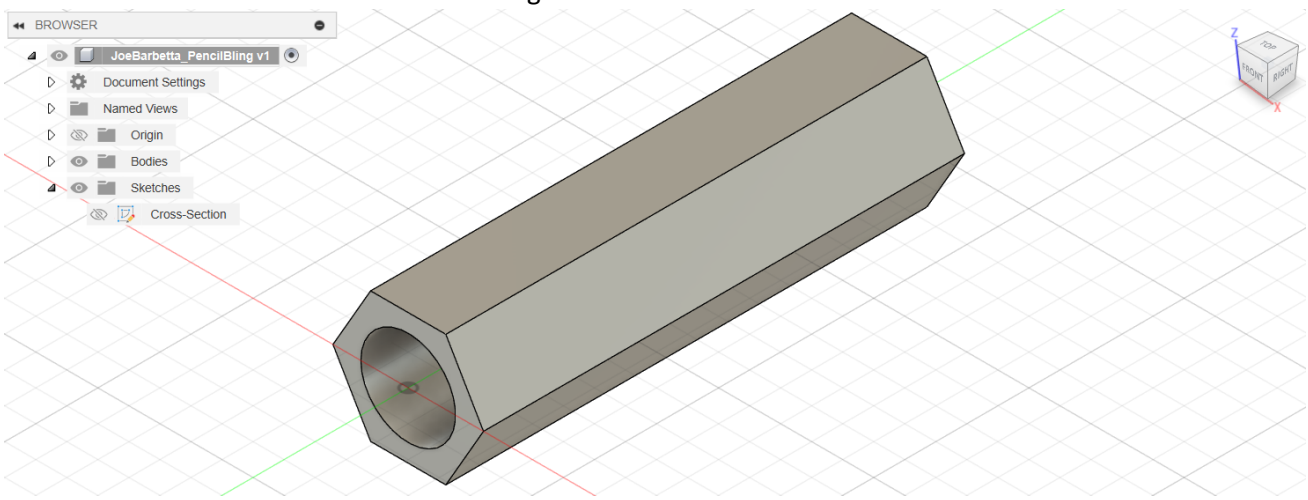
The hexagon should look like that below. As with the circle dimension line the dimension line showing 0.21 may look different as well. Click **Finish Sketch**.



- click on the **Home icon** at the **View Cube**
- select the **Extrude** tool and click on the **region between the circle and the hexagon**
- type **1.55** and click **OK**



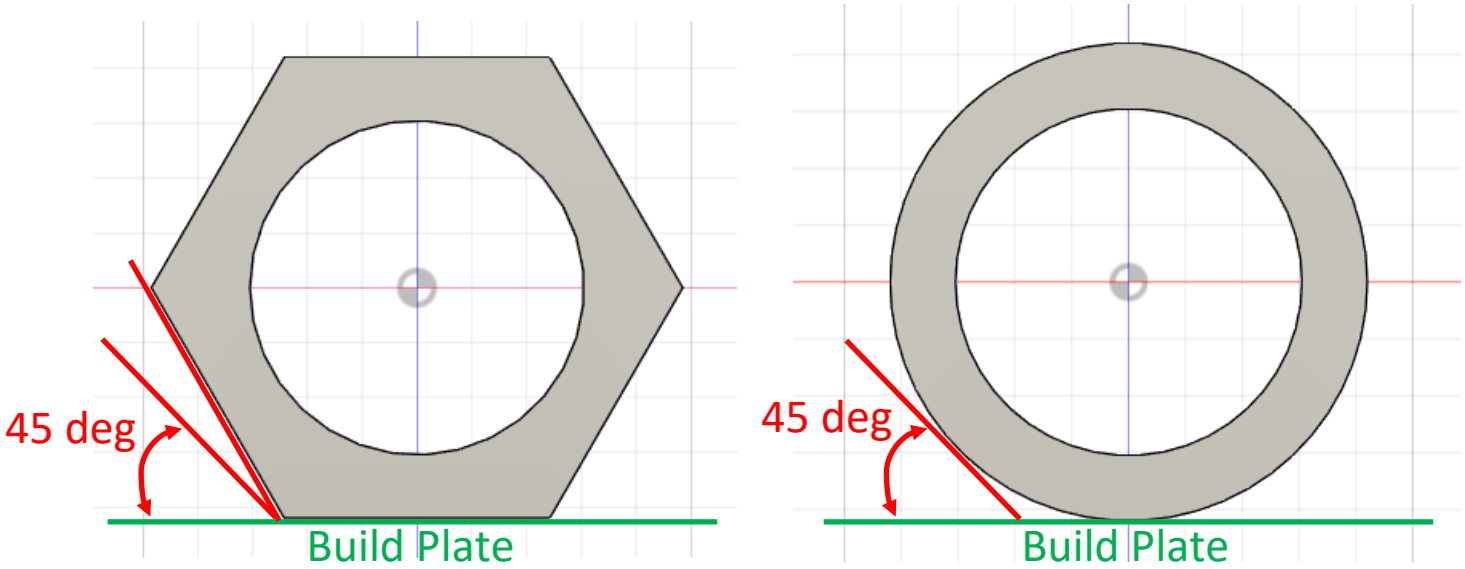
- click on the **Home icon** at the **View Cube** again the the “tube” should look similar to that below



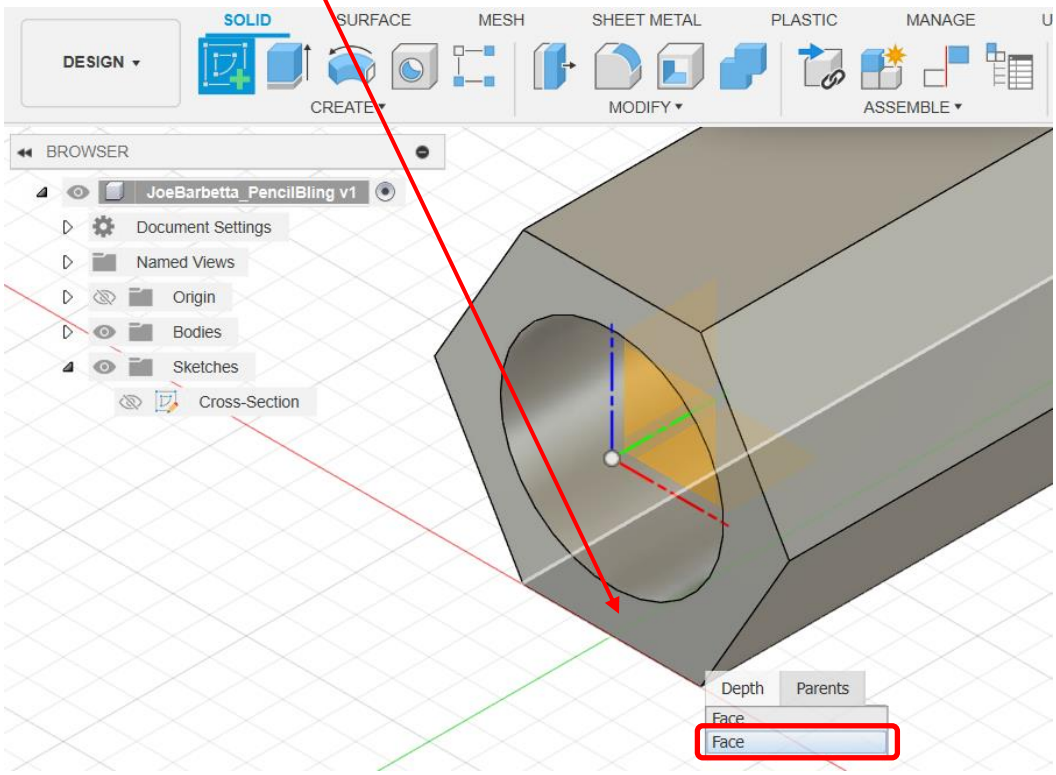
This is Not an instruction step, but explains the tube shape used.

The use of a hexagon shape ensures that the angle at the build plate is greater that 45 degrees. If a circle were used there would be sections at the build plate where the angle will be less that 45 degrees, which would violate the 45 degree rule.

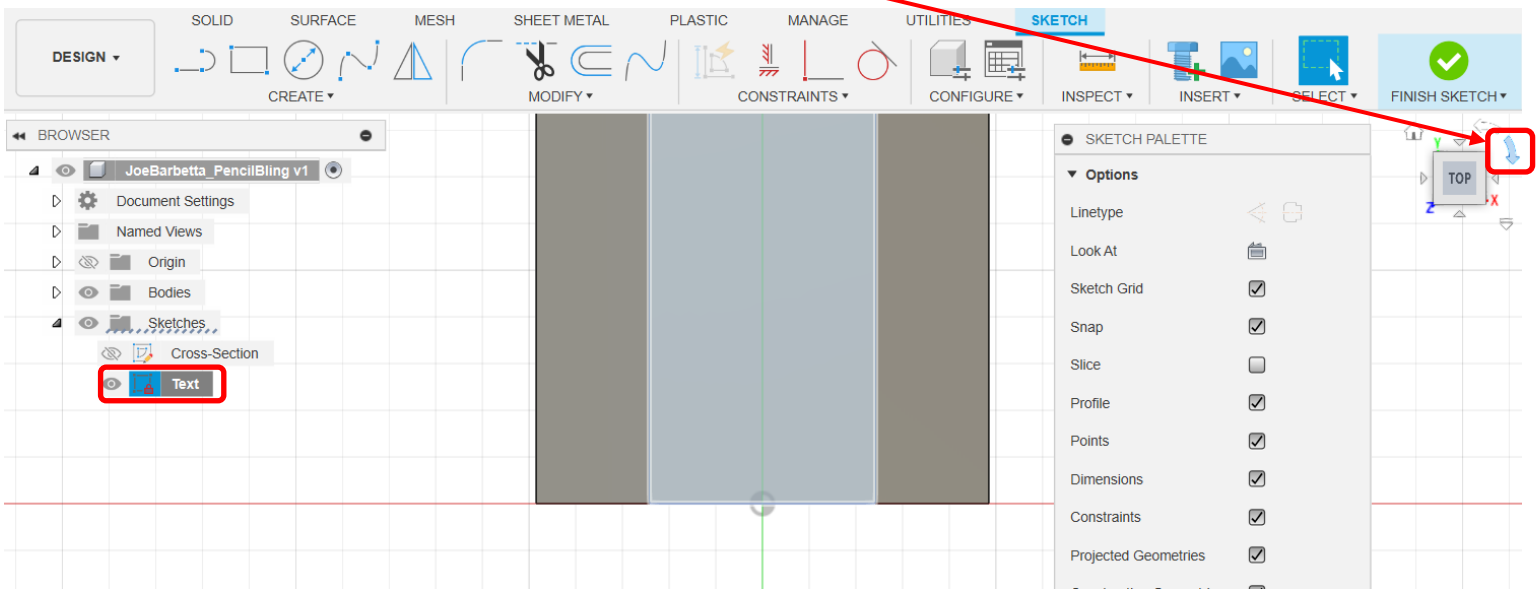
One could argue that the top walls of the inner circle violate the **45 degree rule**. This is true, however, this "bridge" spans a small distance and the distortion will be negligable.



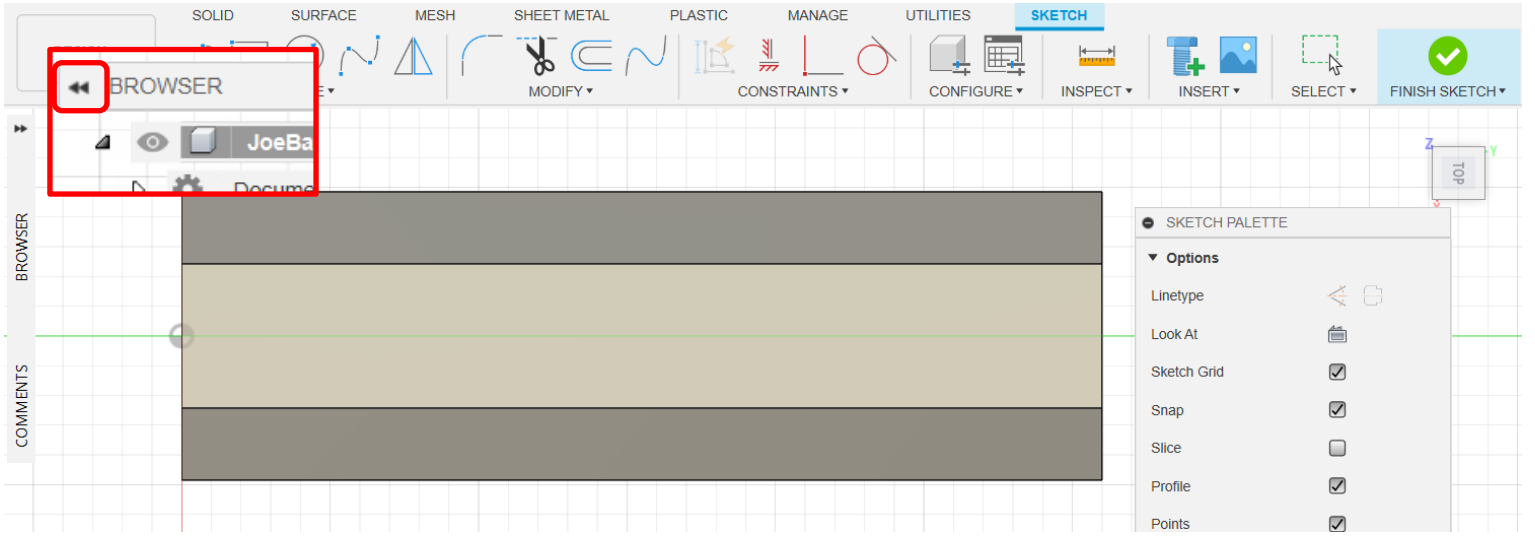
- use the **Mouse wheel to Zoom** into the end as shown
- select the **Create Sketch** tool
- click on the **lower region between the circle and the hexagon** and **hold the mouse button down** until a list of faces shows.
- select the face that **causes the bottom surface to change color**



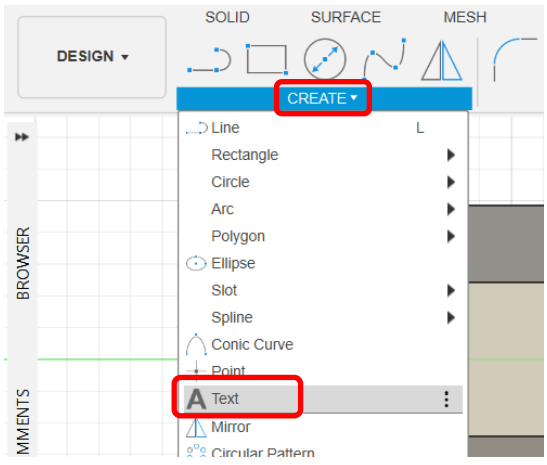
- right-click on the new sketch and select **Rename** to set it name to **Text**
- move the mouse over the **View Cube** and click on the **rotate arrow** that appears



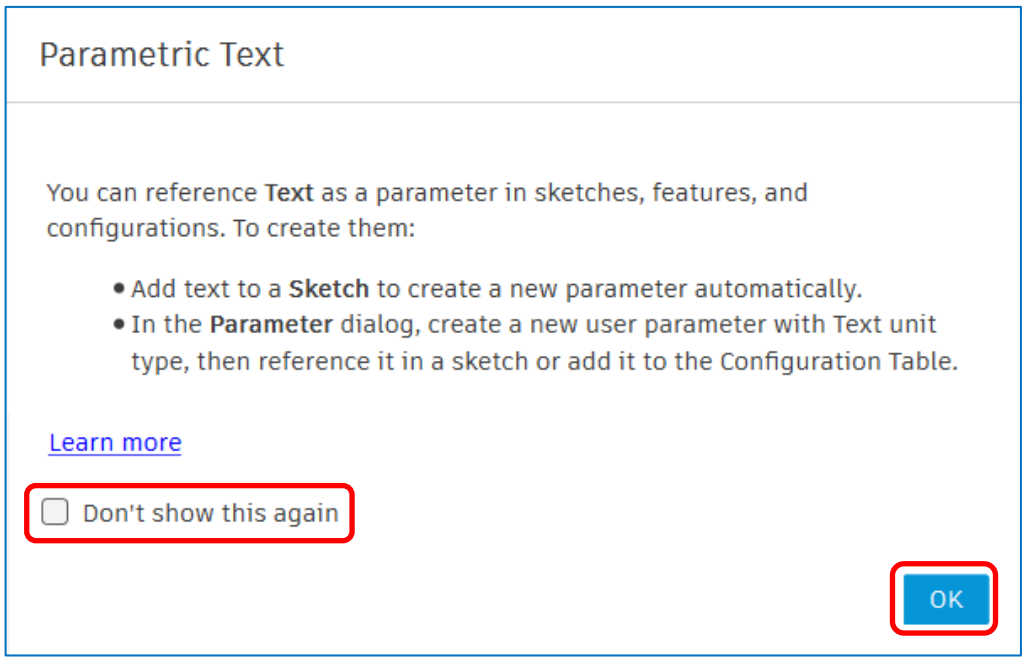
- click on the **left pointing arrows** next to BROWSER to hide this section. It can later be reopened using these arrows.
- Zoom and Pan to achieve a view similar to that below. Holding the Mouse Wheel down allows panning.



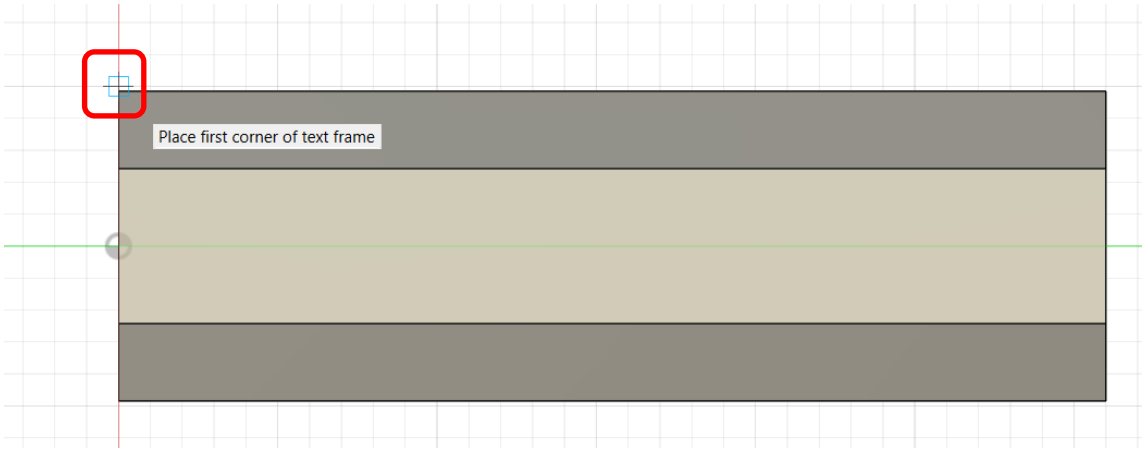
- from the **CREATE** menu select the **Text** tool



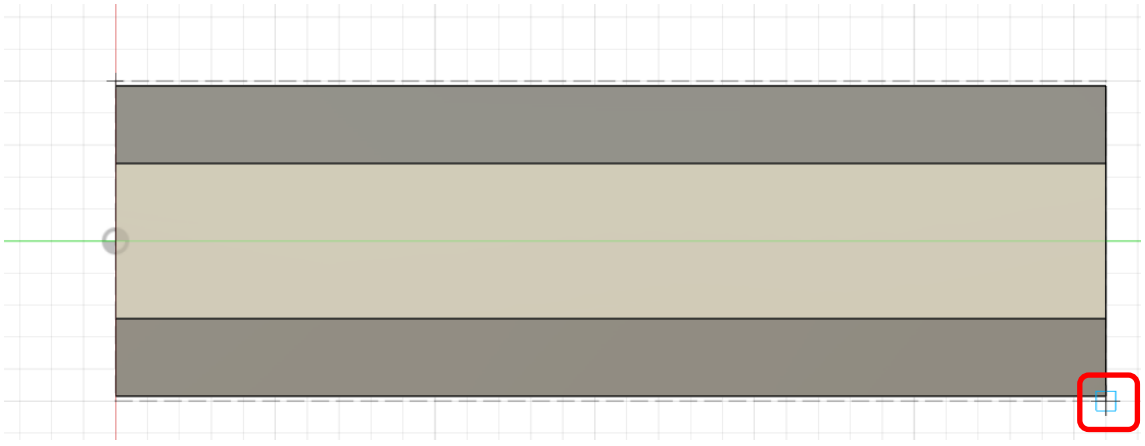
- if this window like this pops up **check the Don't show this again** check box and then click **OK**. It essentially announces a new feature where a Parameter can be used to specify the text. It's a feature that we will Not be using.



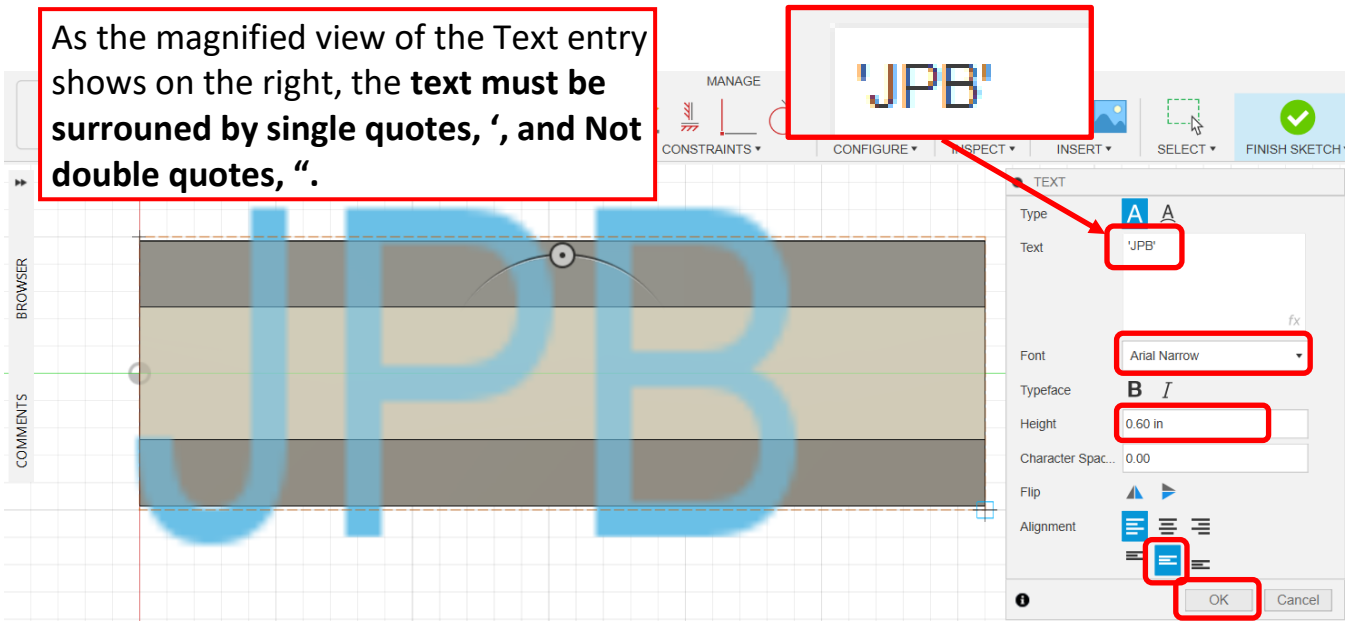
- click at the point shown below to start the text frame rectangle



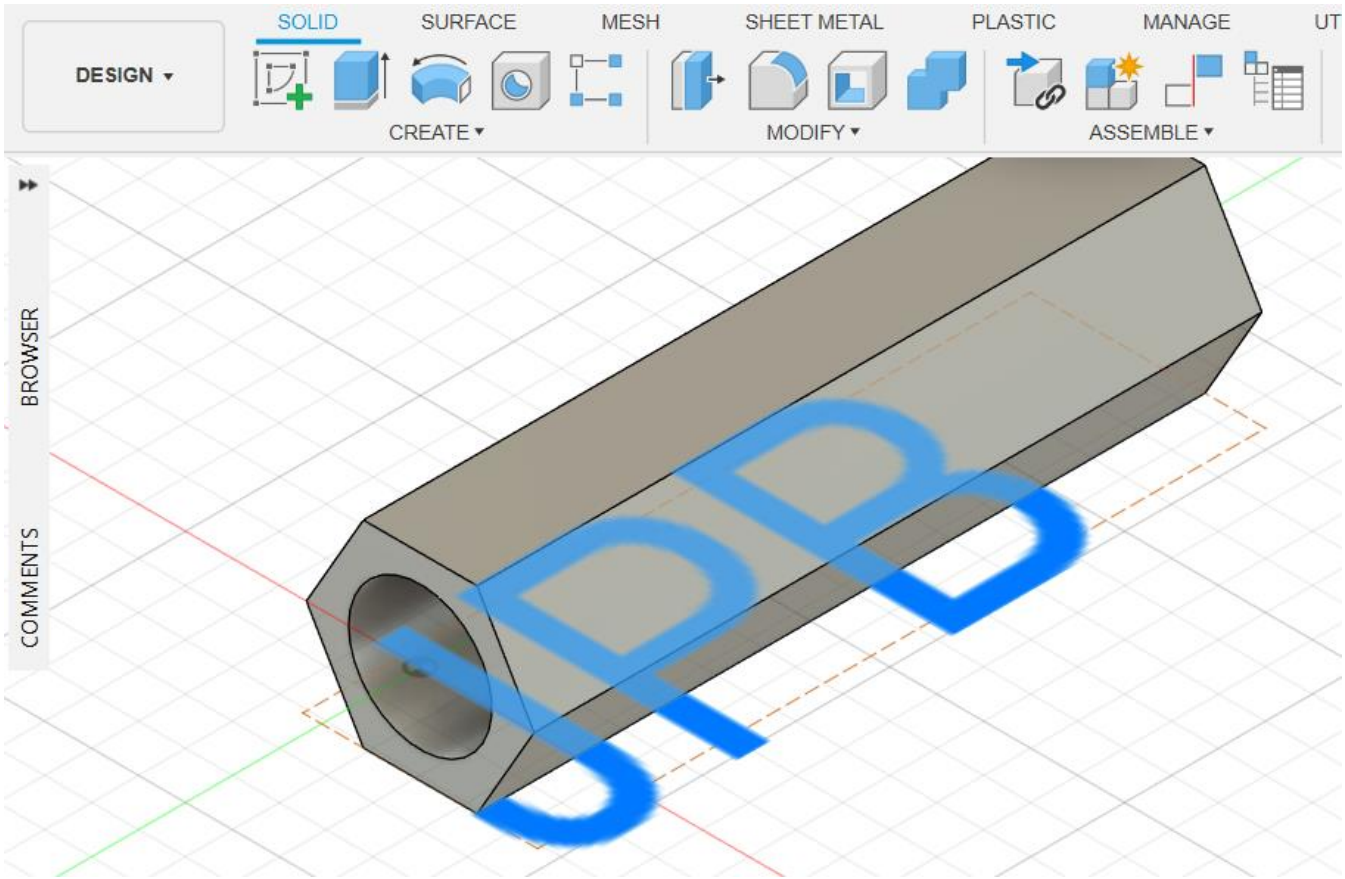
- click at the lower right point to finish creating the text frame



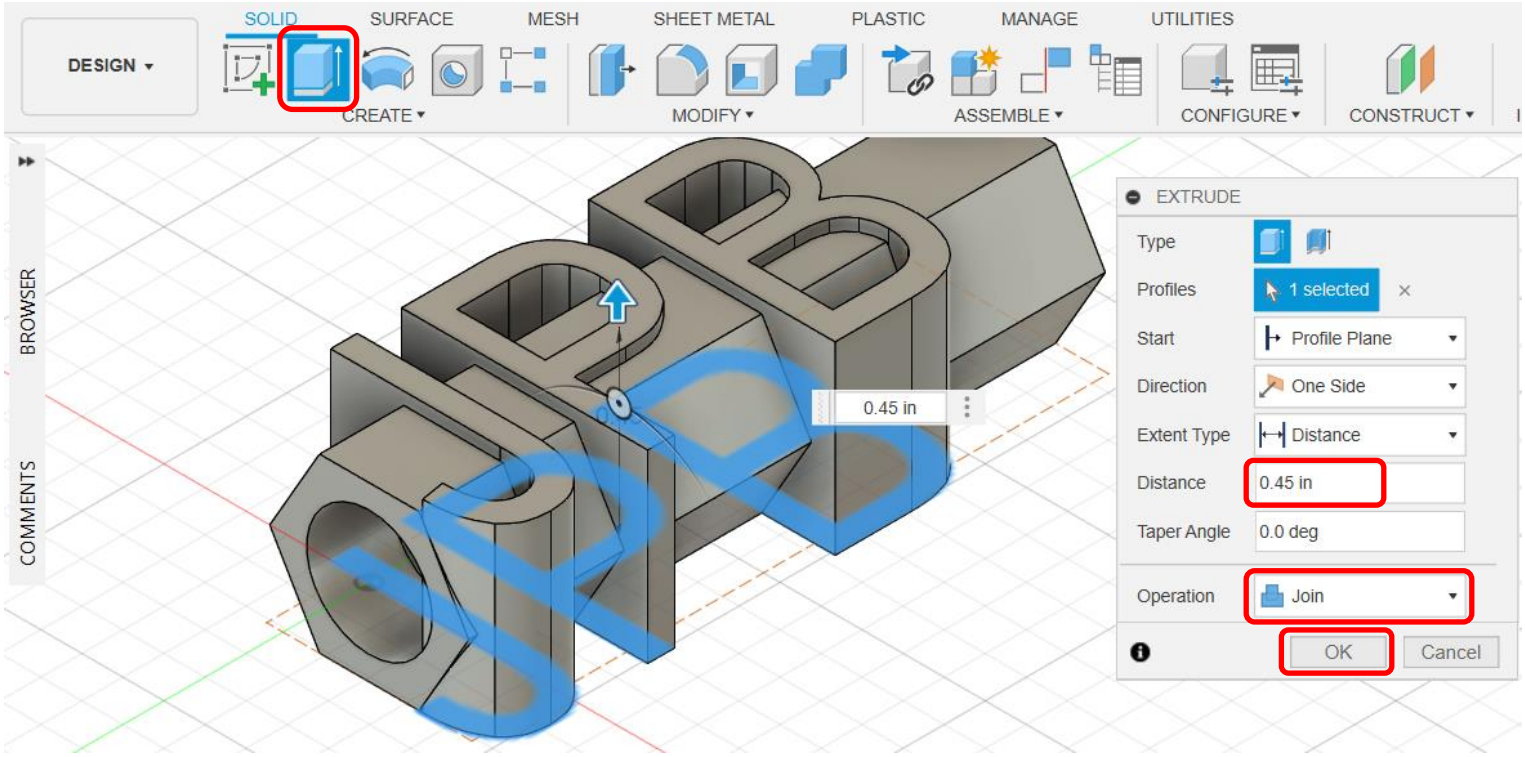
- enter your three initials in the **Text** box
- set the **Font** to **Arial Narrow**
- set the **Height** to **0.60**. There should be some space between the last letter and end of the tube. If there is less than that shown reduce the text height.
- click on the **lower middle Alignment option**
- click **OK** when done and then **Finish Sketch**



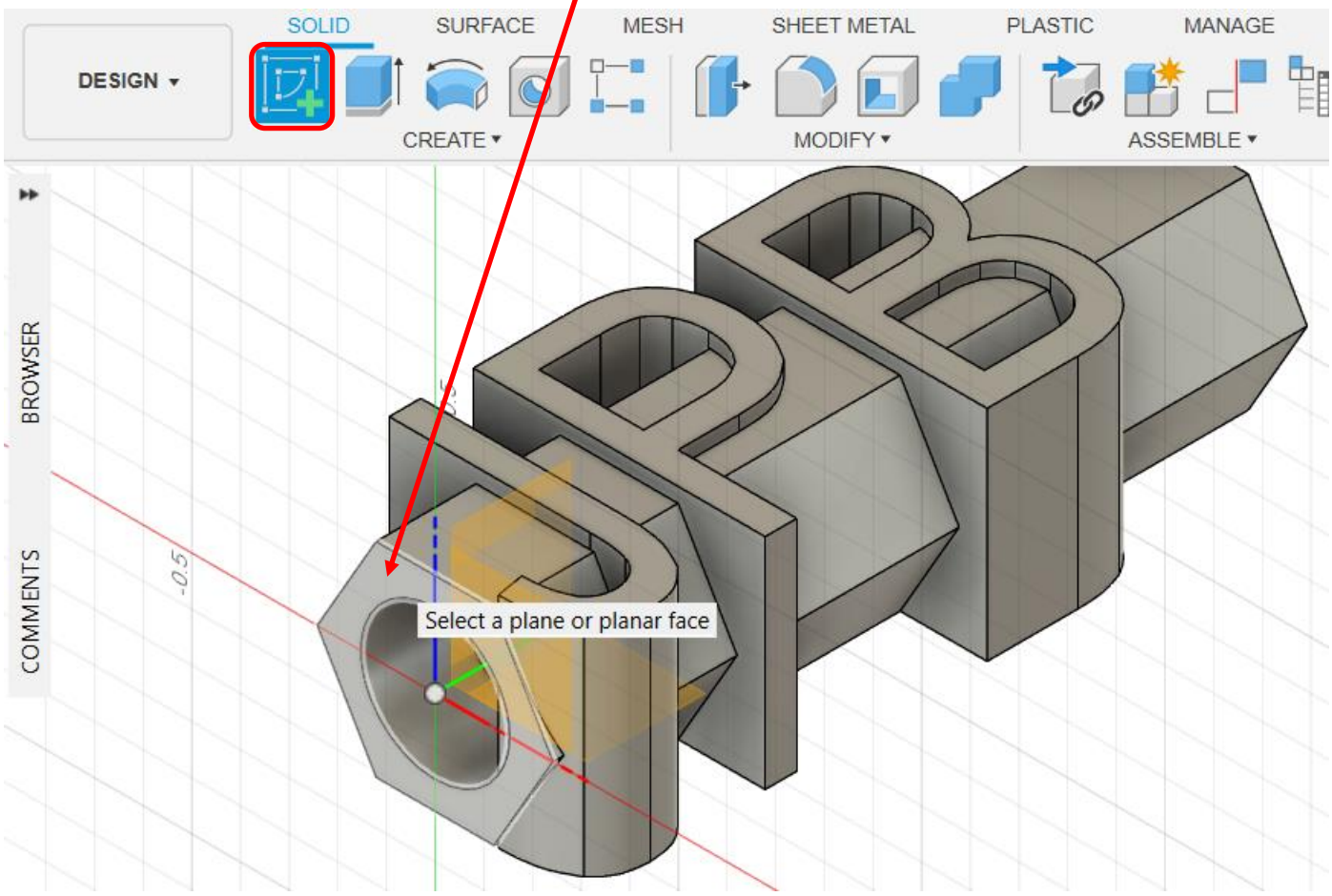
- use the **Home** icon at the **View Cube** and adjust the view similar to that below
- **click on a letter**, which will cause it to turn dark blue



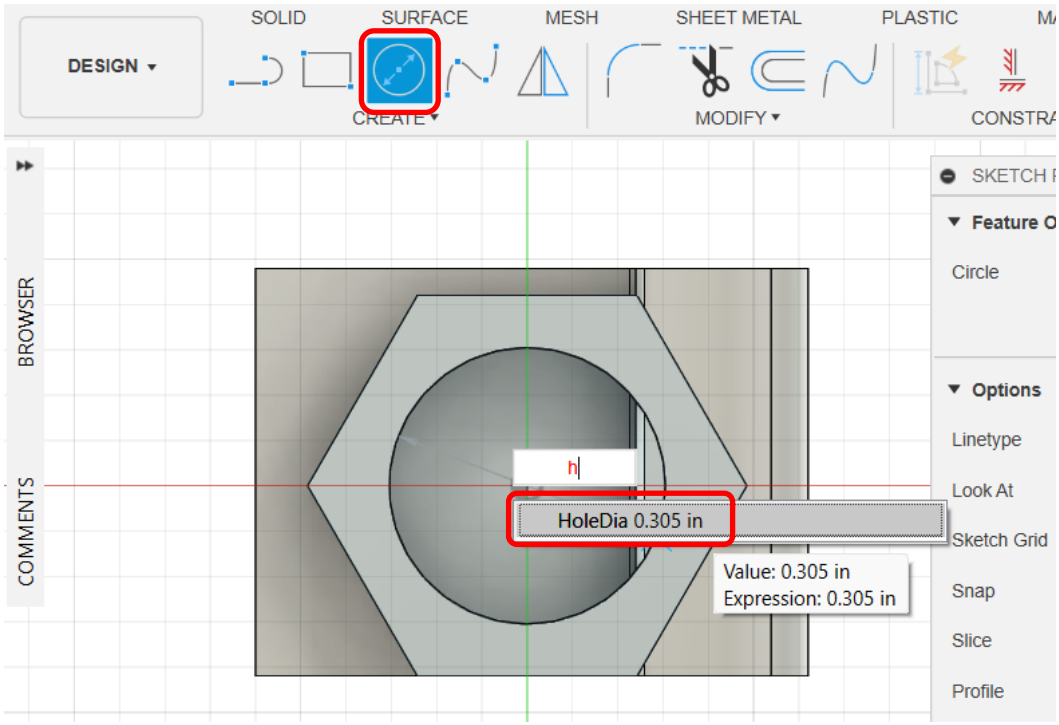
- select the **Extrude** tool
- for **Distance** enter **0.45** and change the **Operation** to **Join** and click **OK**



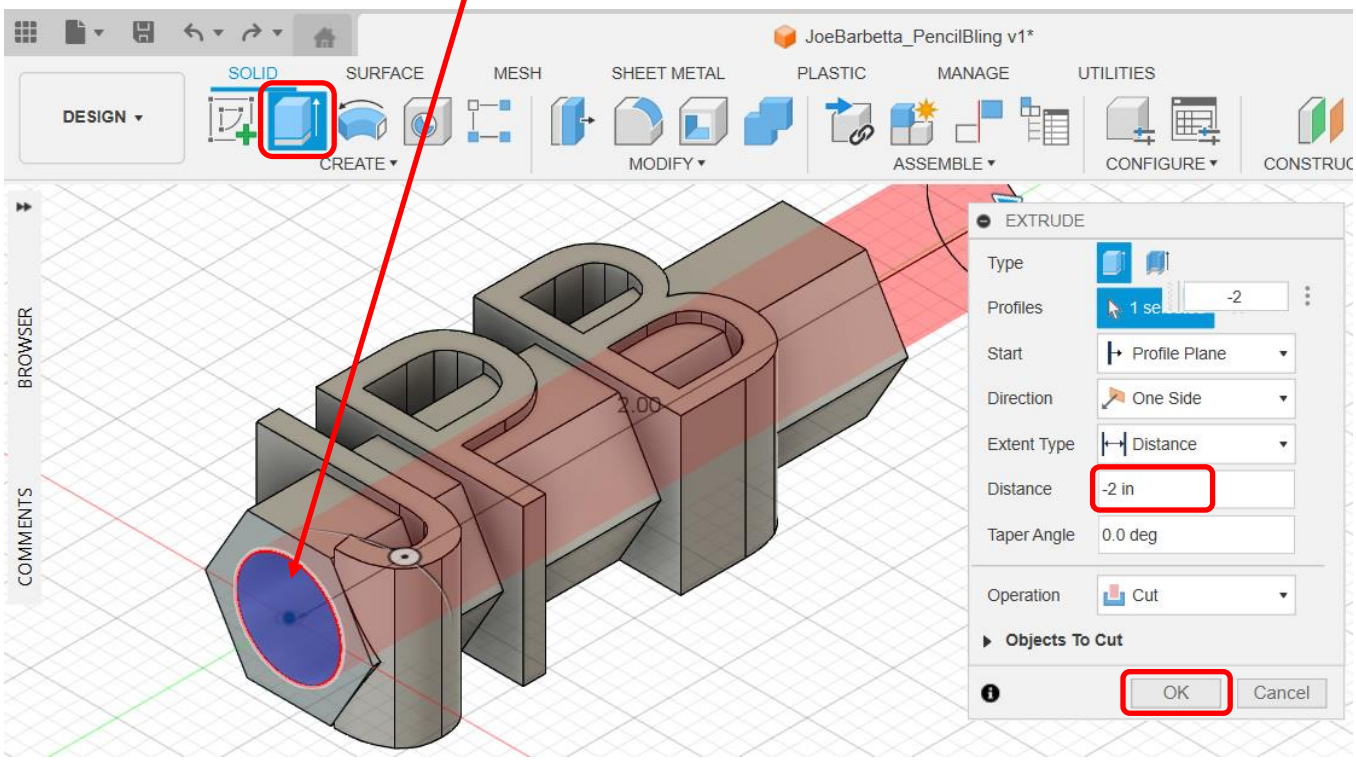
- select the **Create Sketch** tool and click on the **end face** of the tube



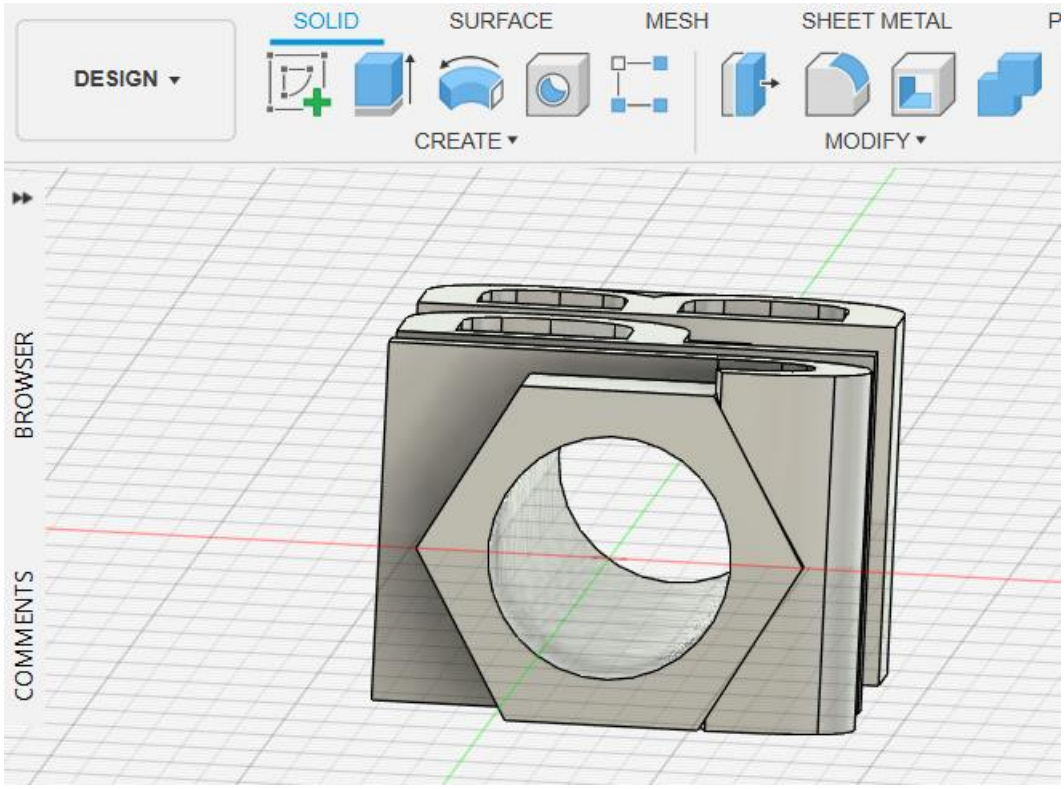
- as you notice that you can no longer see through the tube yell **“What the sigma! How will this go on a pencil?”**
When the letters were extruded they ended up adding material in the hole. Perhaps we could have waited to make that inner hole, but we will just do it again.
- select the **Circle** tool, type **h**, and select **HoleDia** option, and press the **Enter key**
- click **Finish Sketch**



- return to the Home view
- select the **Extrude** tool, click on the **end circle**, enter **-1.5** (note the minus sign) and click **OK**
Note that we used **-2** instead of **-1.55** (the extrude value we used earlier). It doesn’t matter. As long as it is longer it will complete the hole.



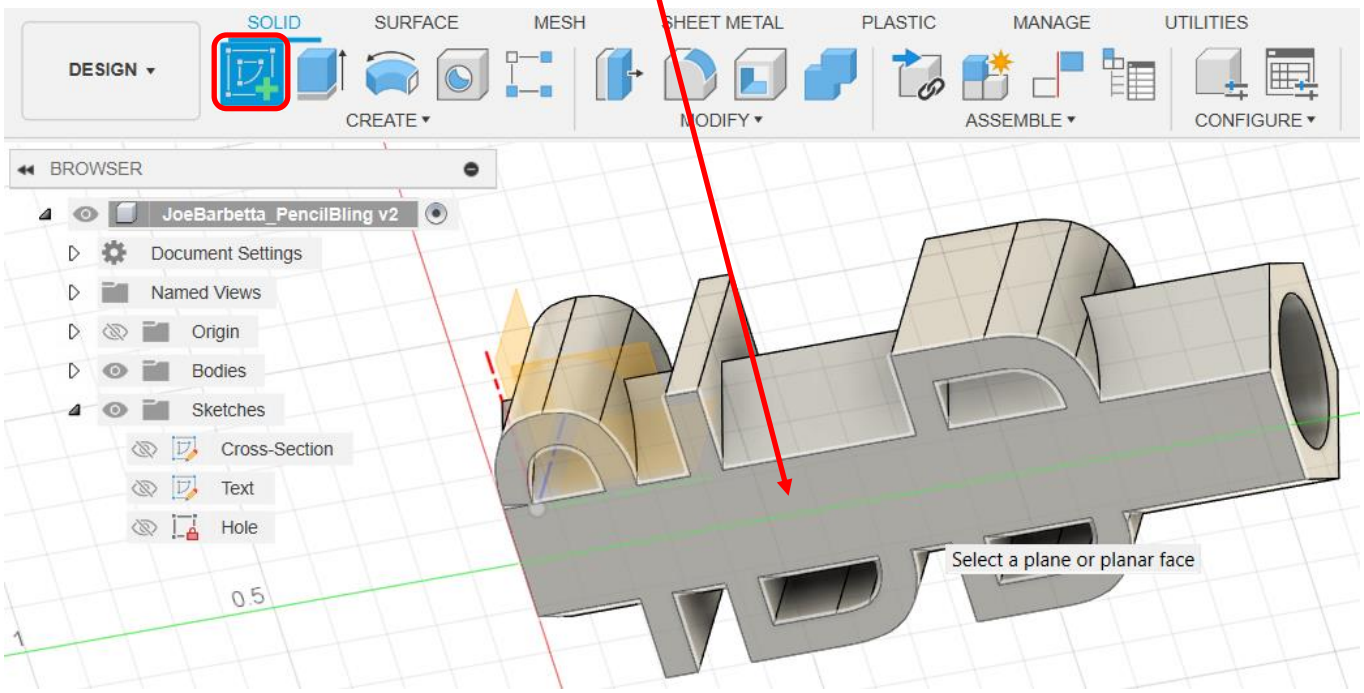
- adjust the view to verify that there is a continuous hole



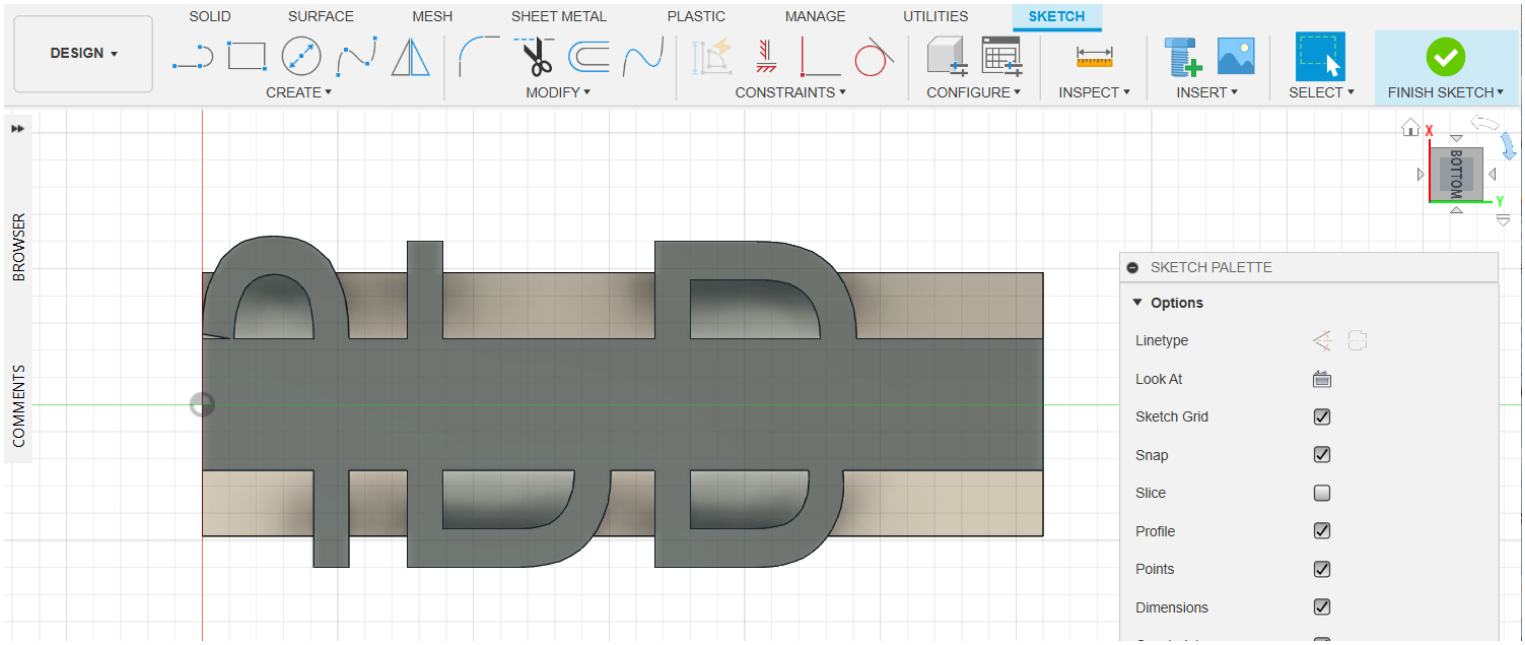
Brim Block

Extra material will be added for a brim block.

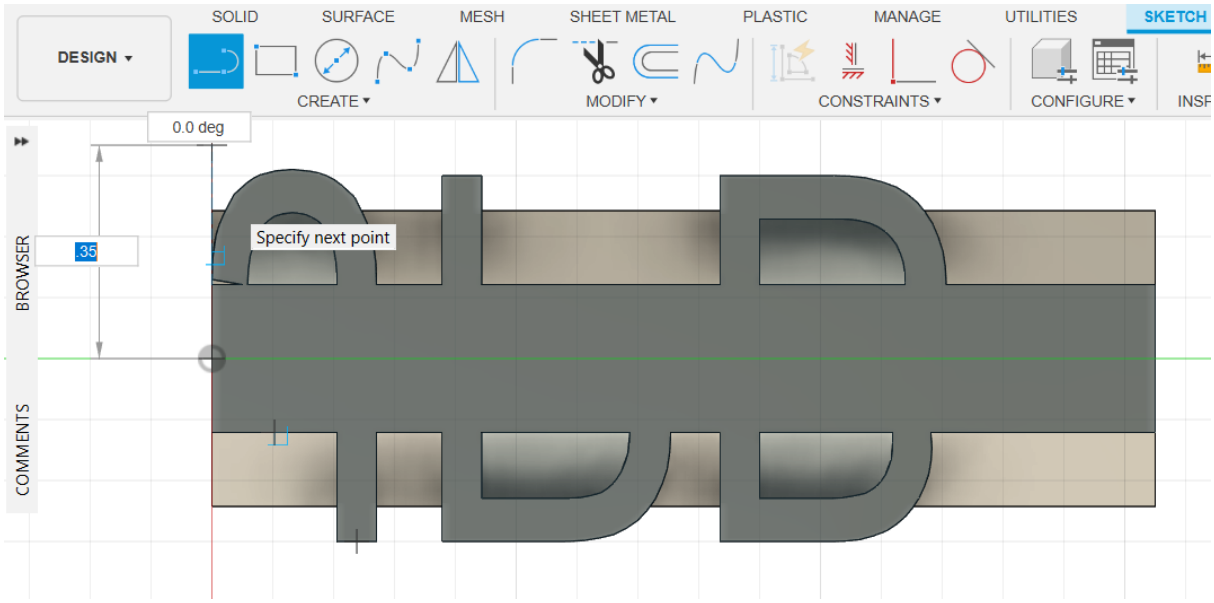
- adjust the view to access the bottom of the pencil bling body
- select the **Create Sketch** tool and click on the **bottom surface**

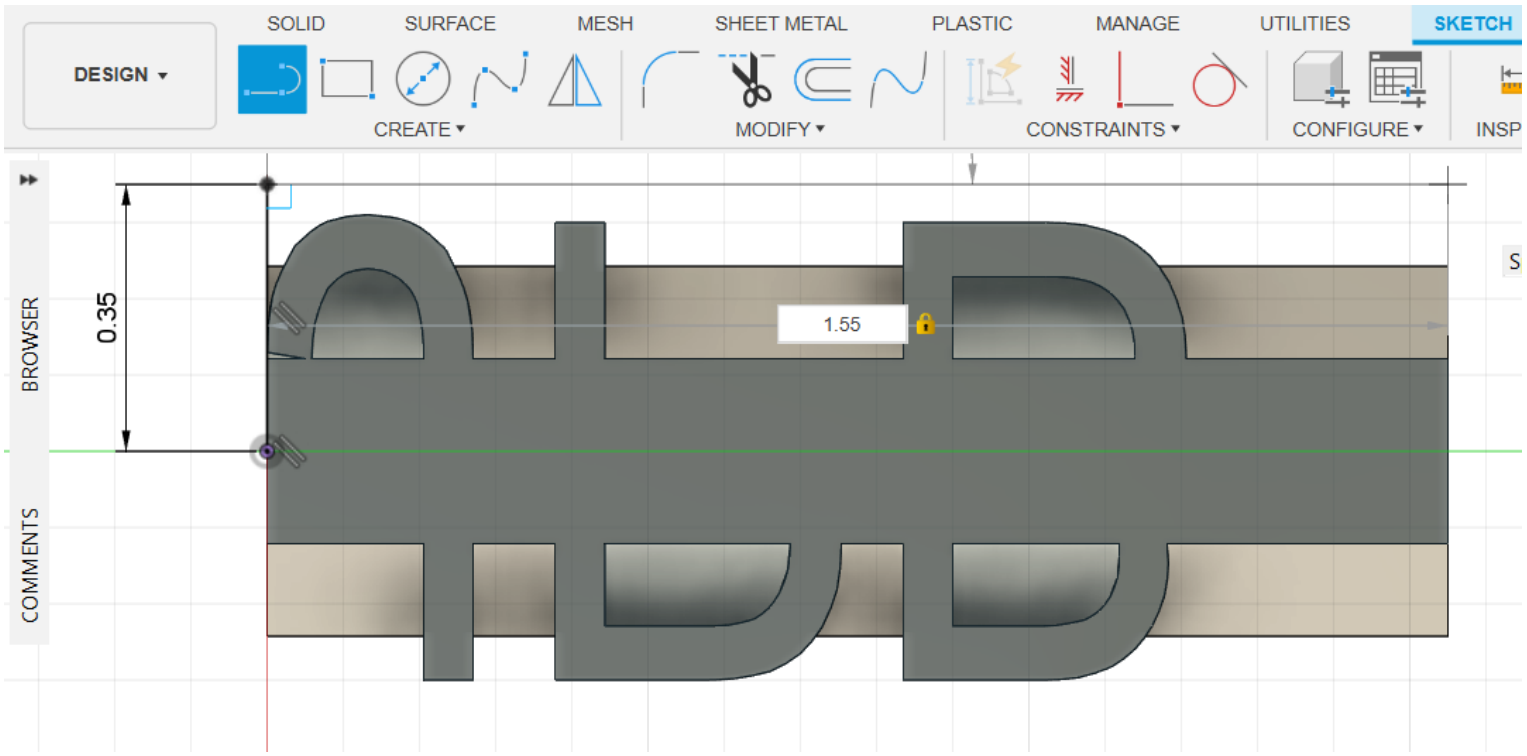


- if needed click on a rotation arrow to achieve a view similar to below



- select the Line tool and click on the center point of one end, extend the line up, type 0.35, and press the Enter Key

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COMMENTS
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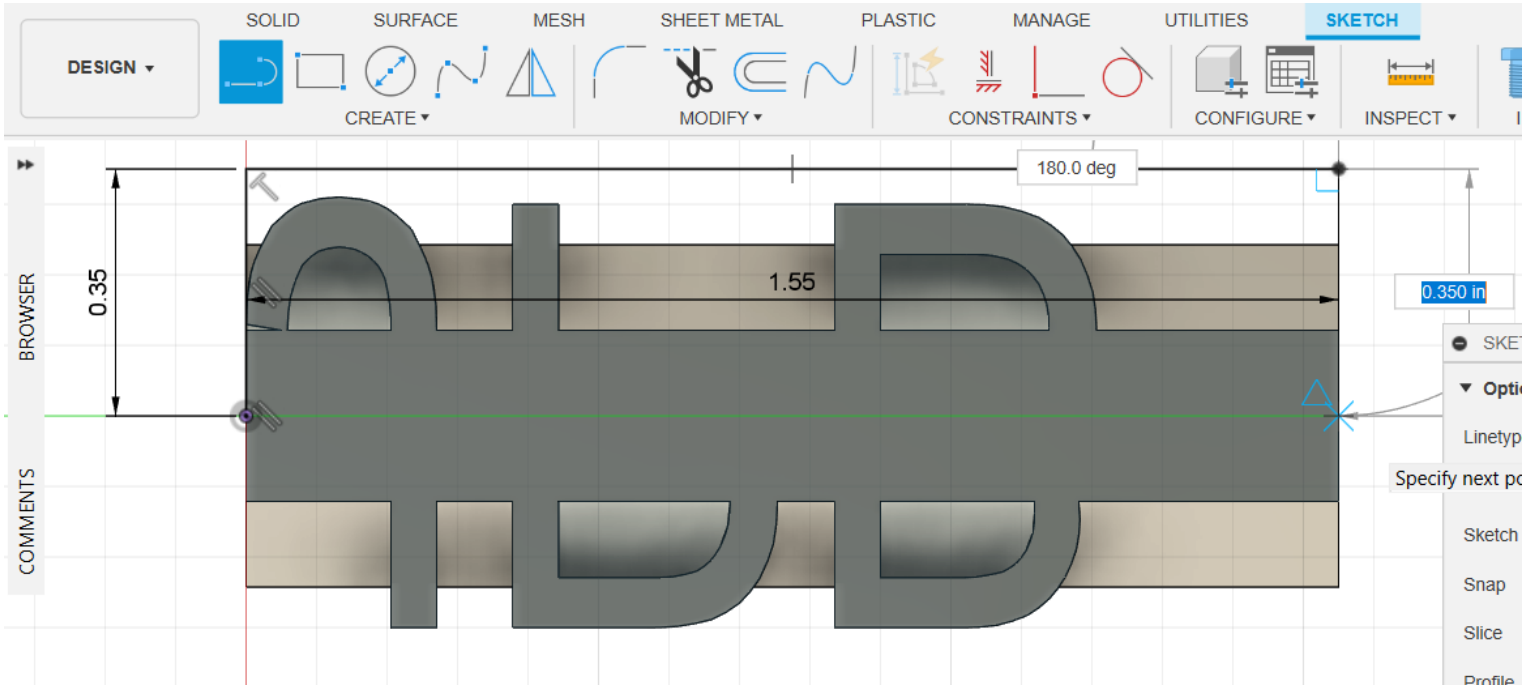
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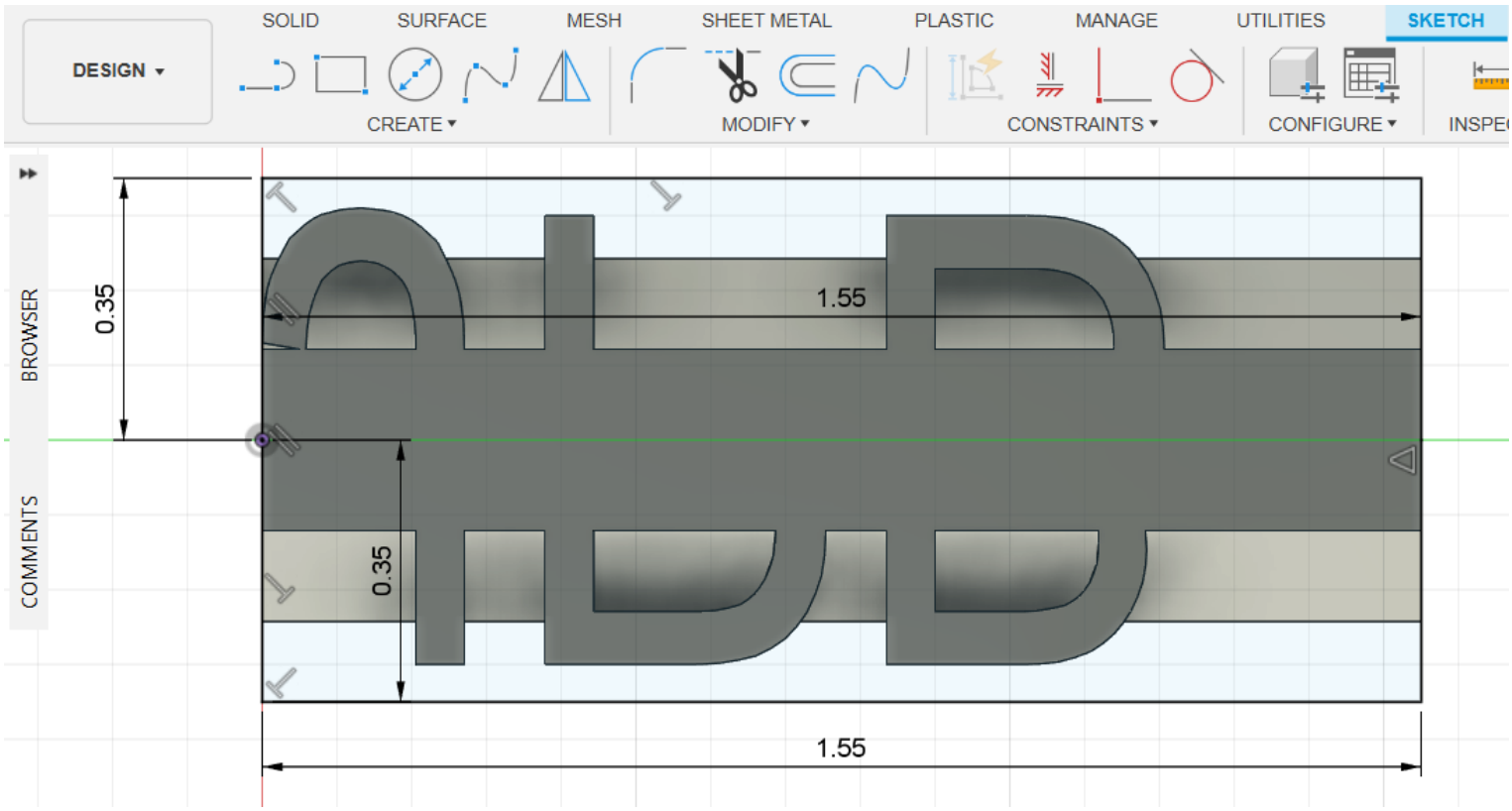
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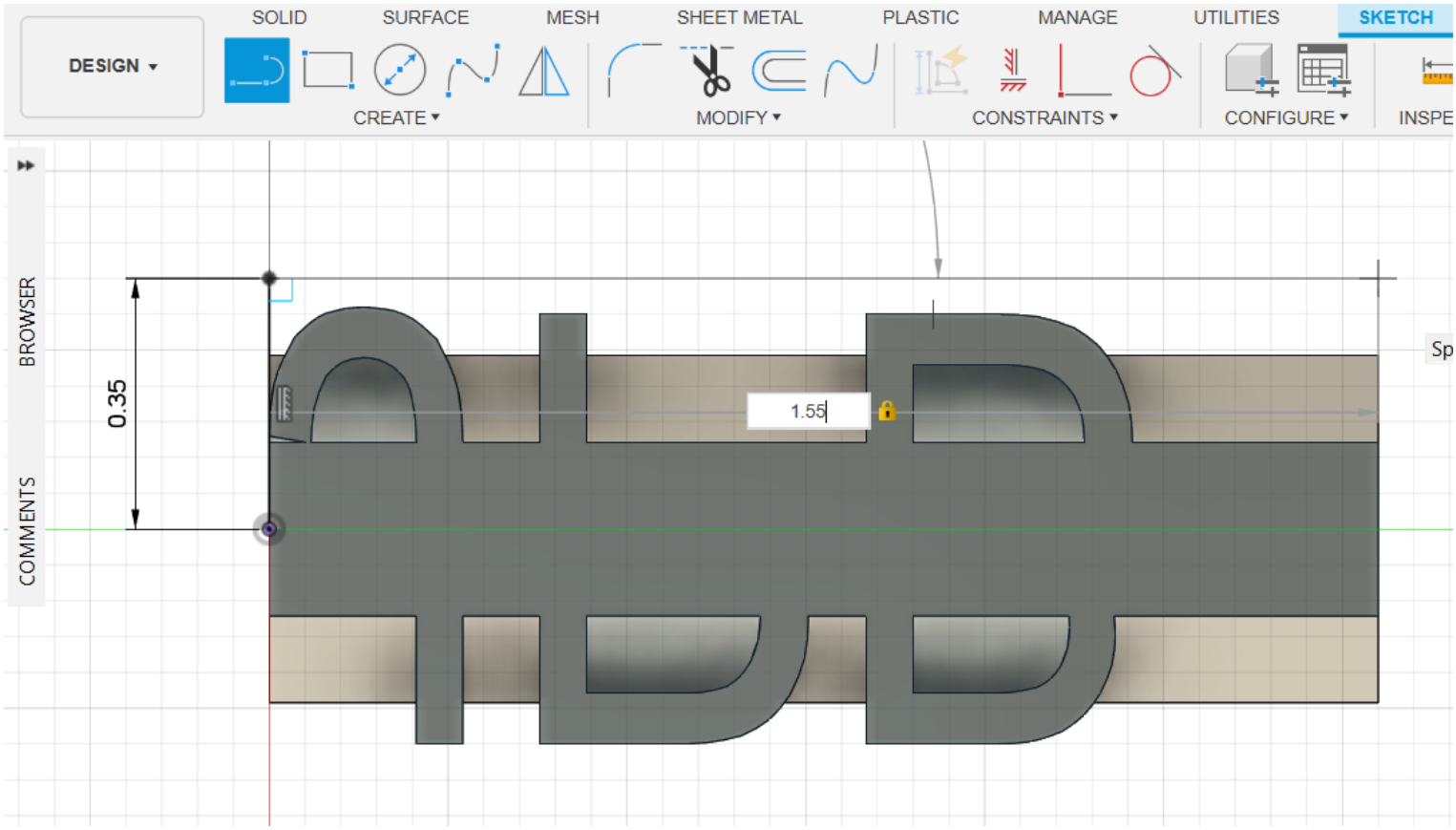
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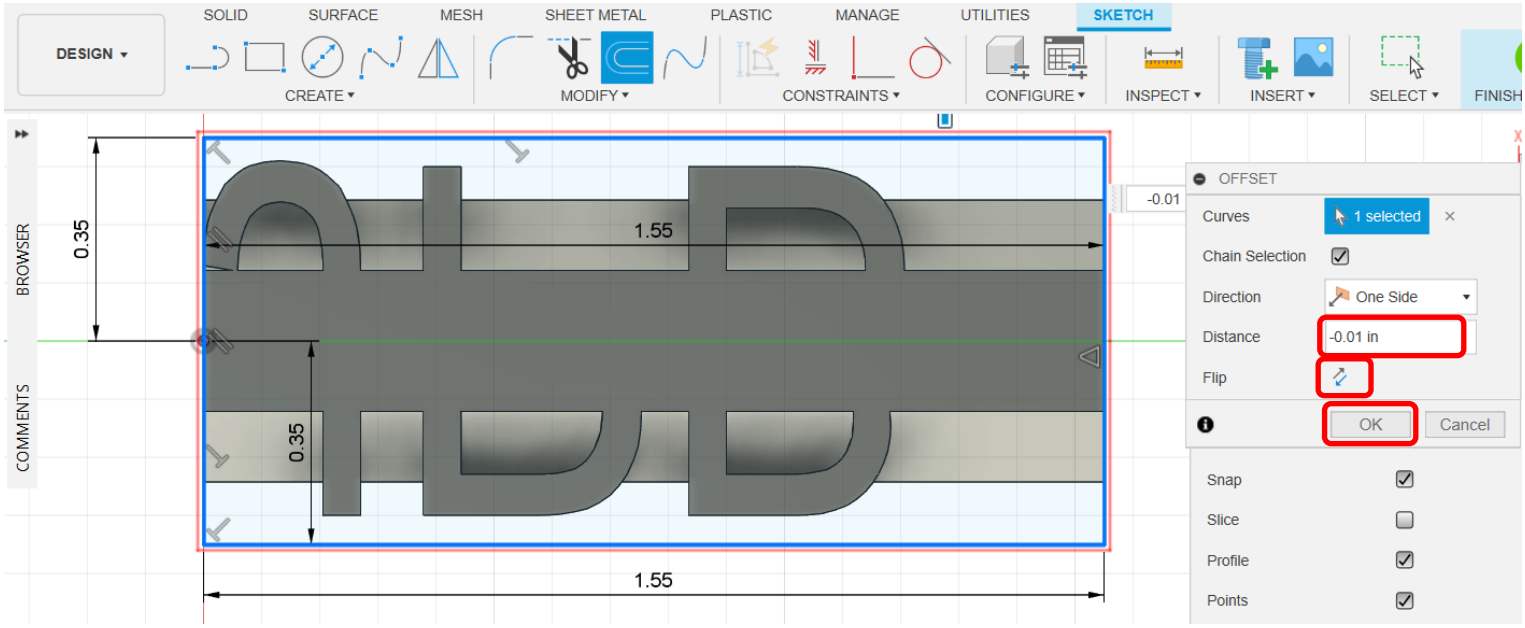
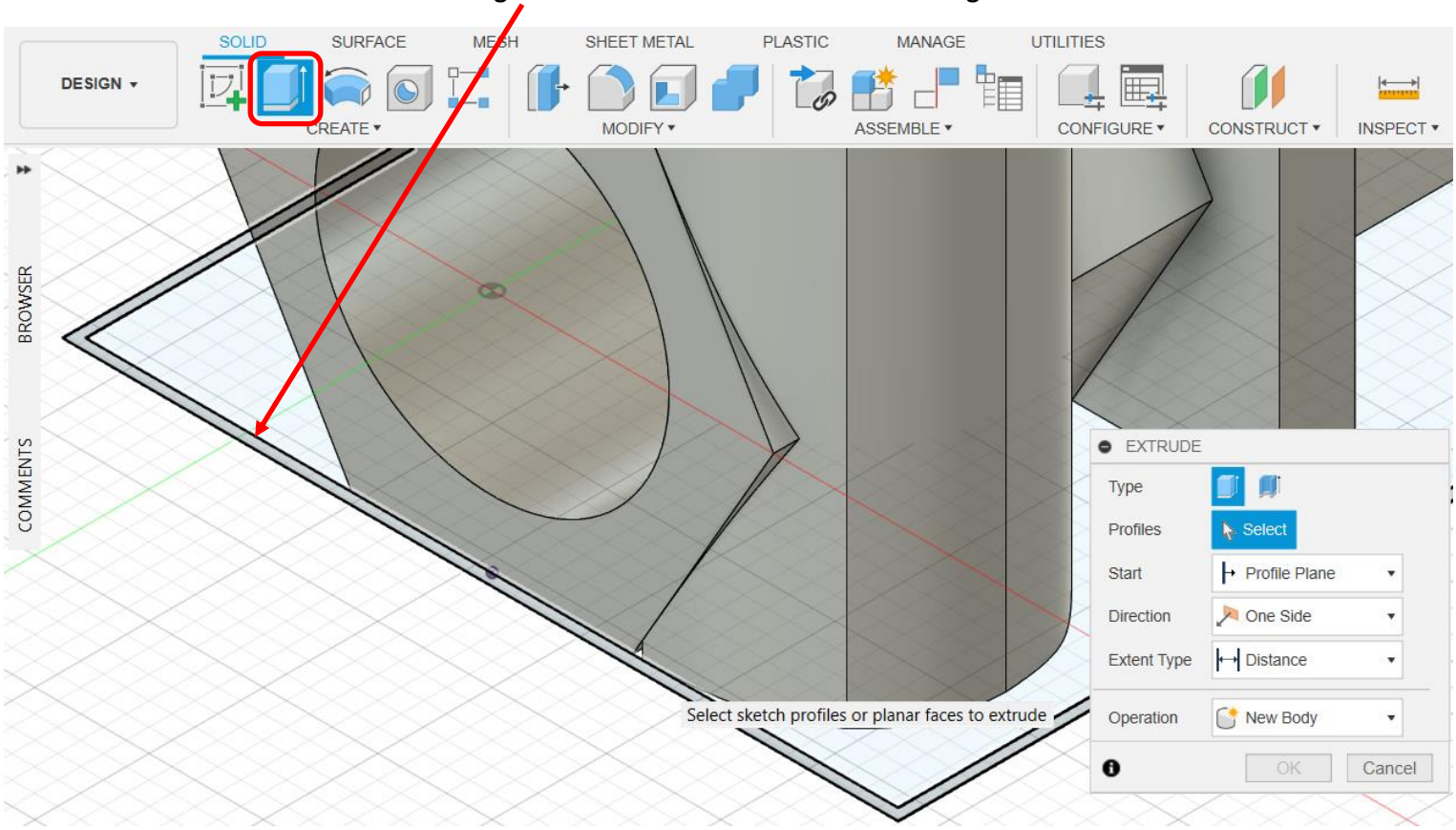
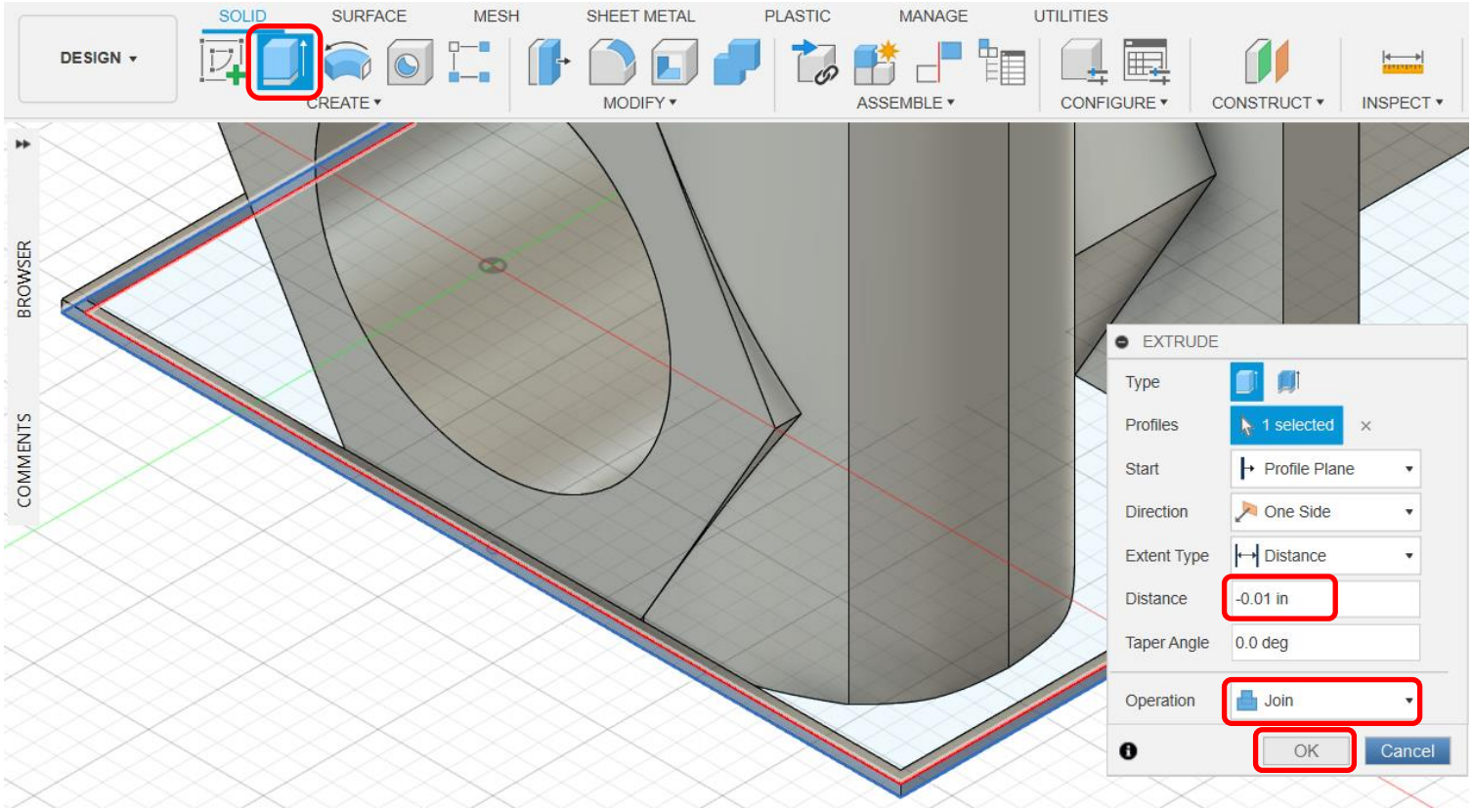
- select the **Offset** tool. If it cannot be seen, select it from the **MODIFY** menu
- click on the rectangle just created and type **0.01**
- if the new red rectangle is inside the blue rectangle, click on the **Flip** icon to cause the red reactangle to be outside the blue
- click OK and then click Finish Sketch



- return to the **Home** view and Zoom into the end to achieve a view similar to this
- select the **Extrude** tool and click on the **region between the inner and outer rectangle**

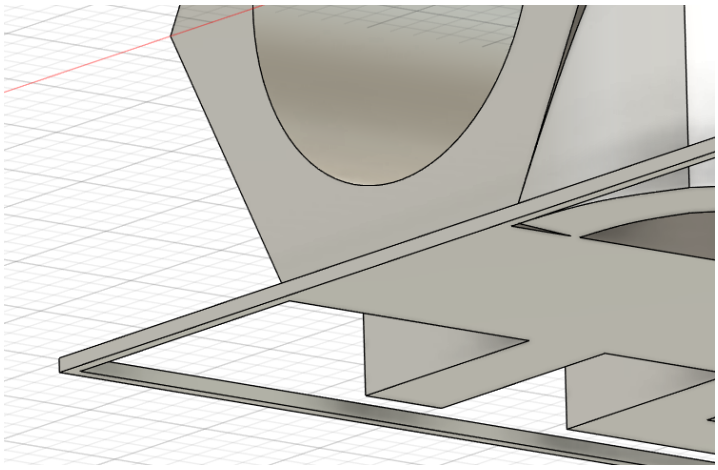


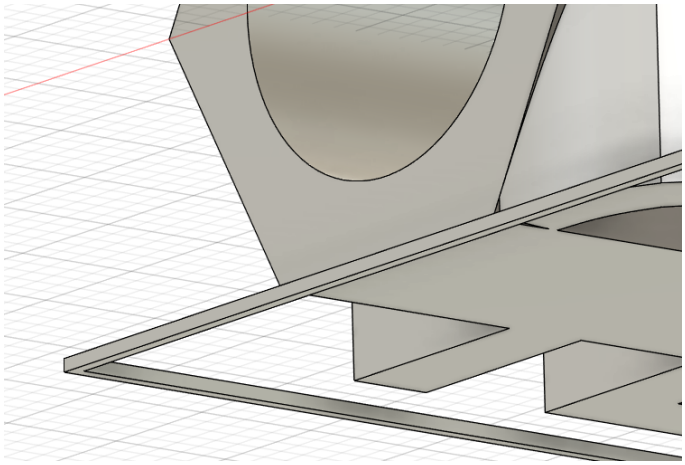
- enter **-0.01** (note the minus sign) for **Distance**, which should cause the region to extend upward slightly
- change the **Operation** to **Join** and click **OK**



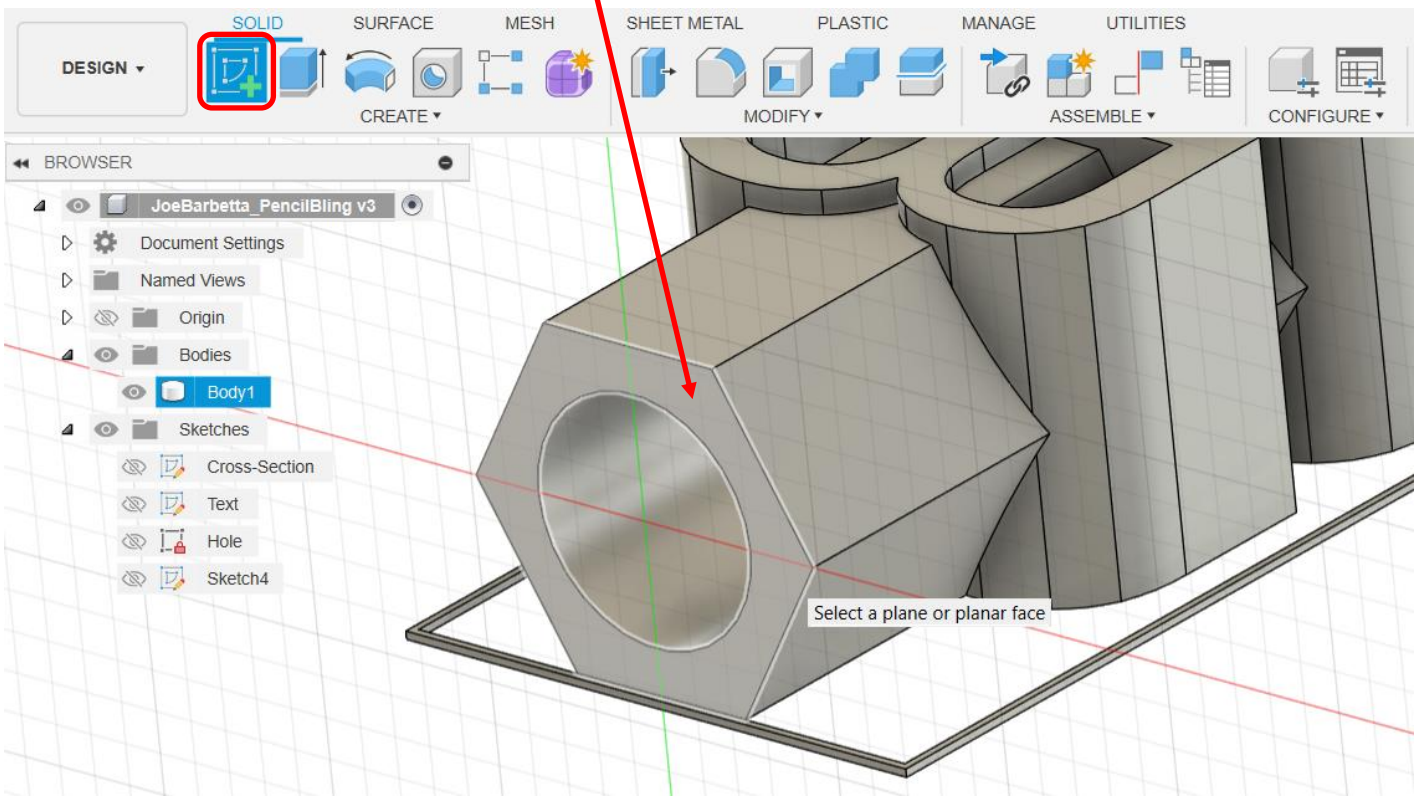
- Zoom into the bottom near the end.

The left picture shows that the brim block was properly extruded upward. The right picture shows a brim block that was extruded in the wrong direction.

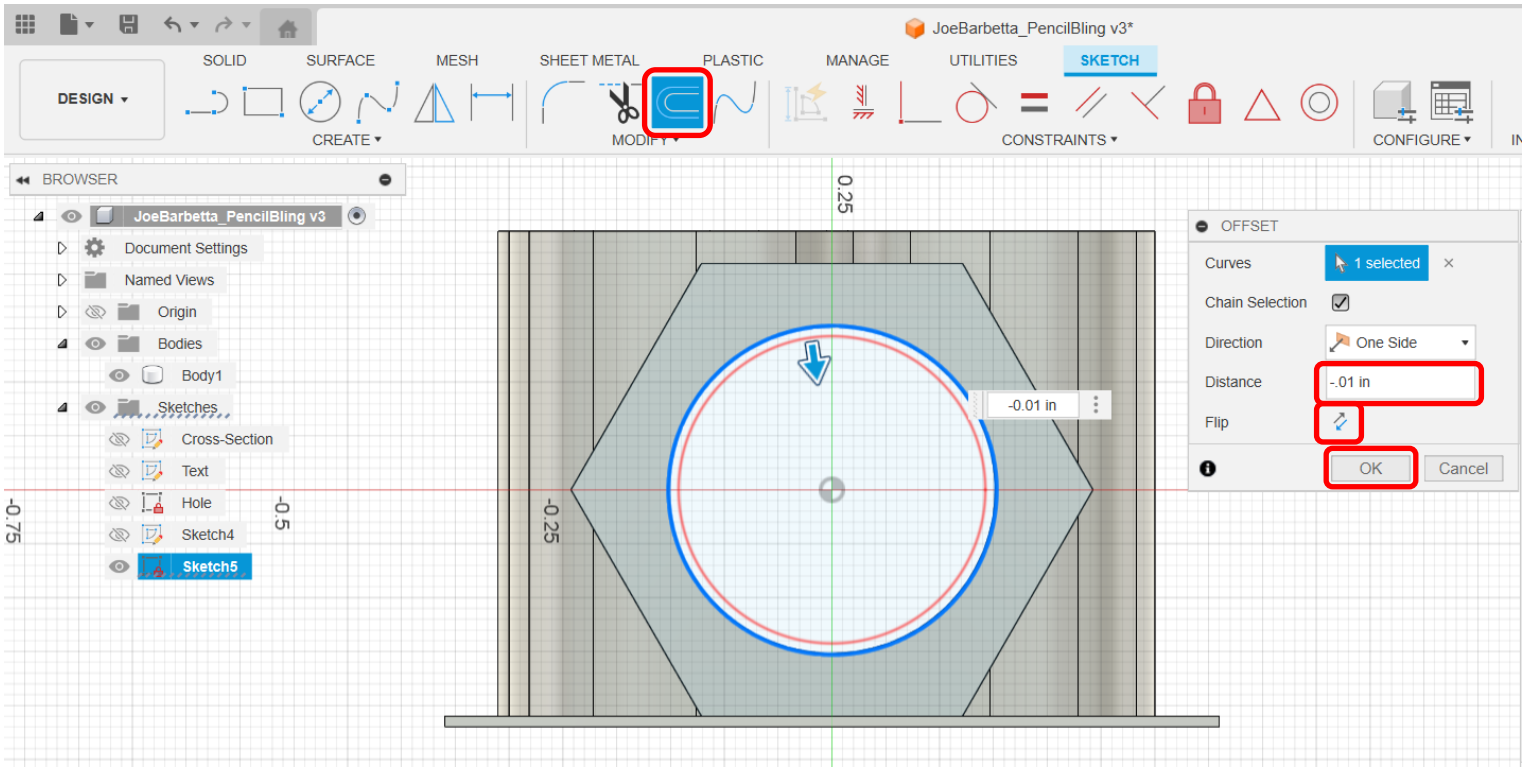


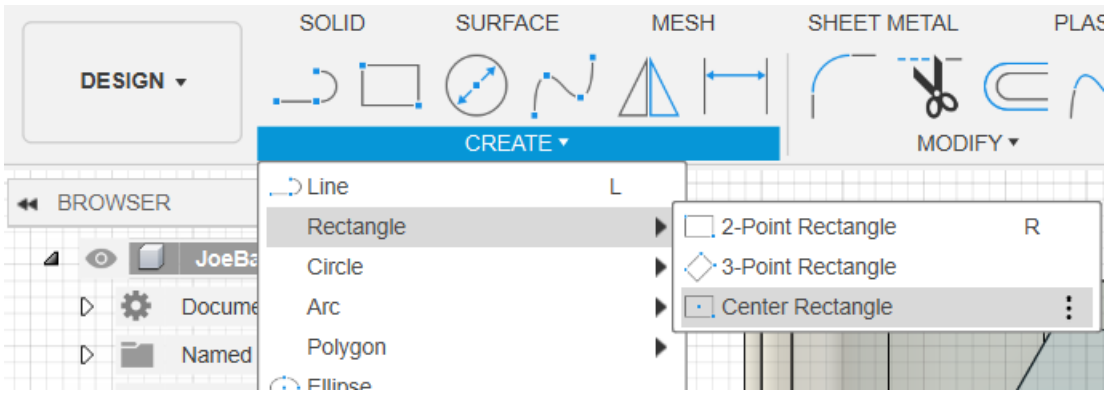


- adjust the view to access the end of the far end as shown
- select the **Create Sketch** tool and click on the **end face**

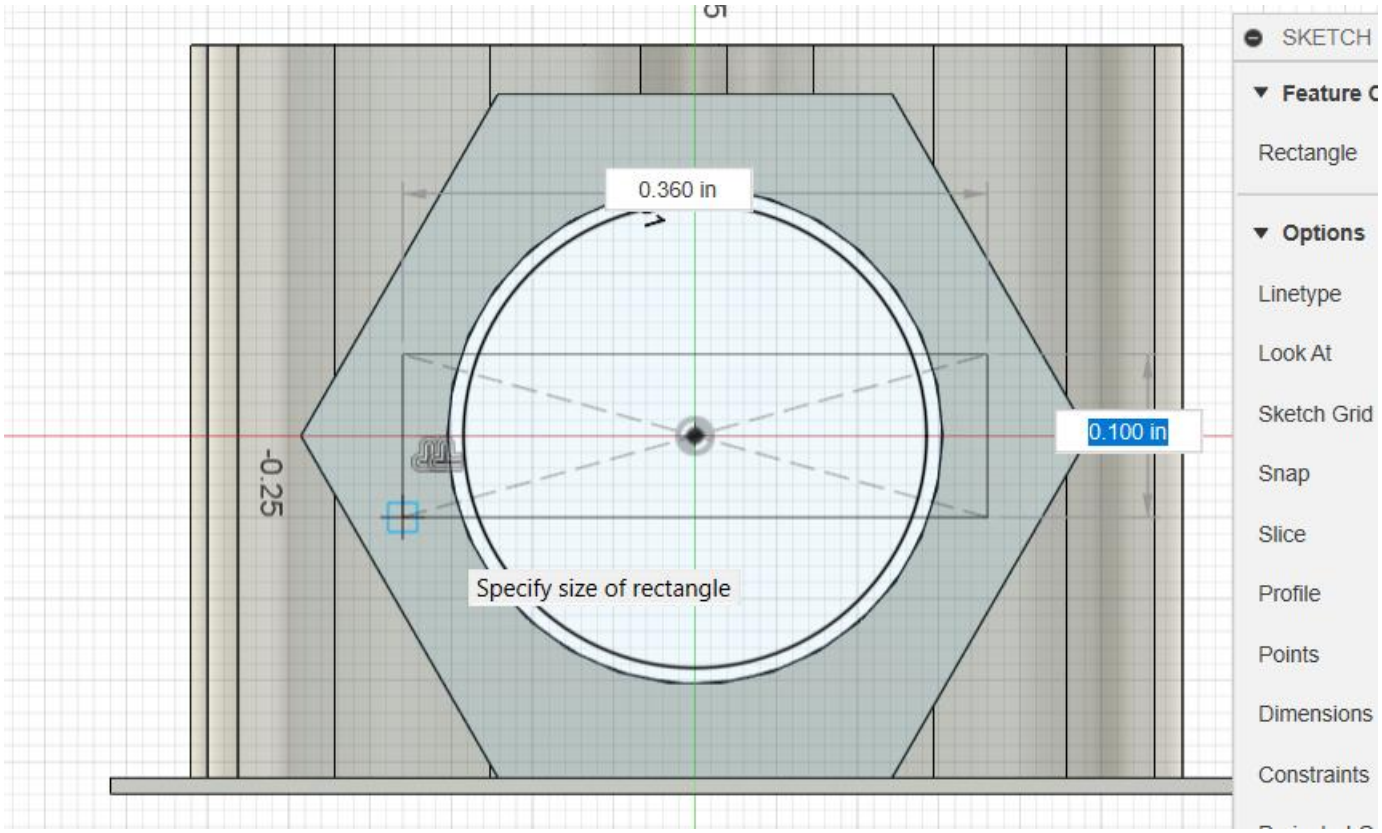


- select the **Offset** tool. If it is not visible, select it from the **MODIFY** menu.
- enter **-0.01** (note the minus sign). The red circle should be inside the blue circle. If not click the **Flip** icon.
- click **OK**

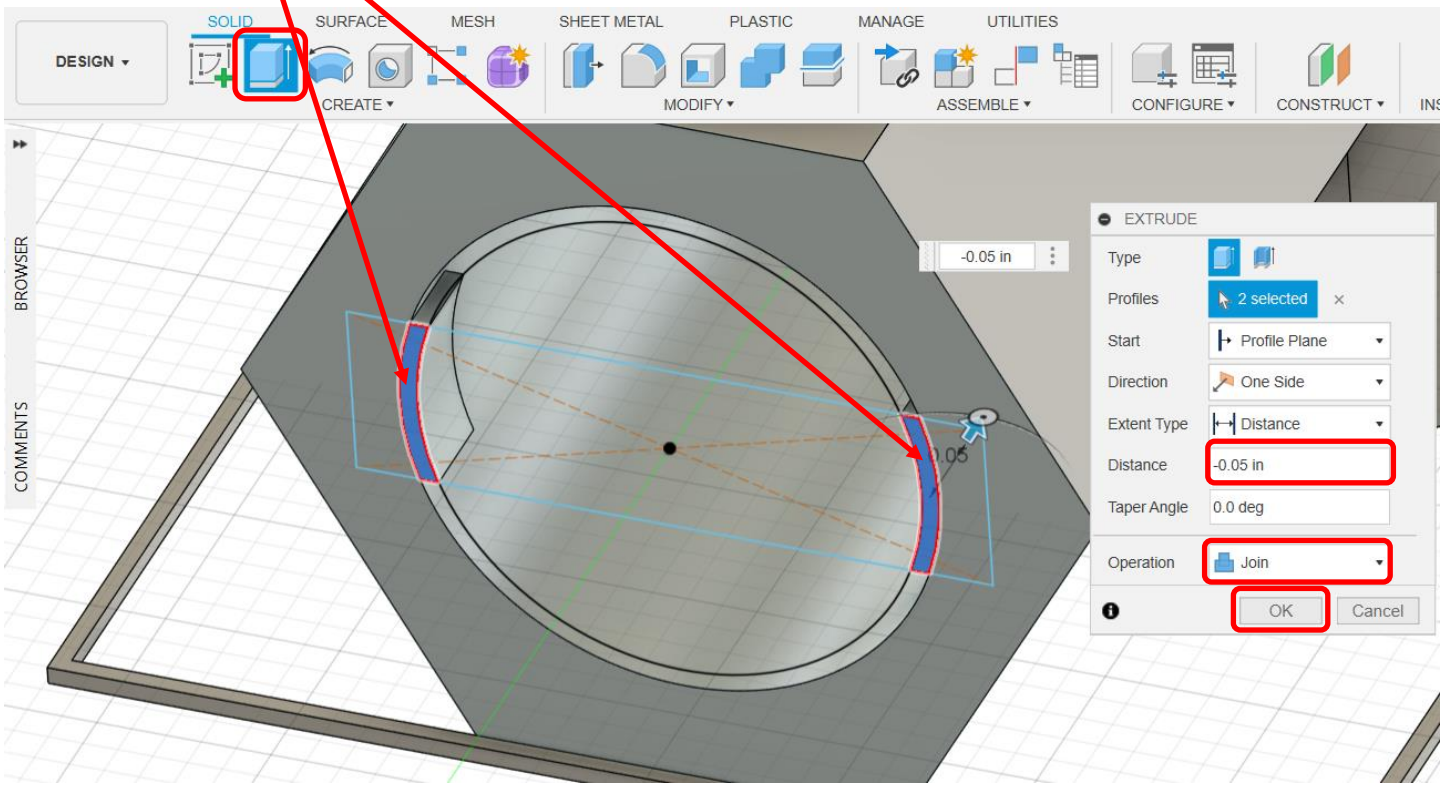




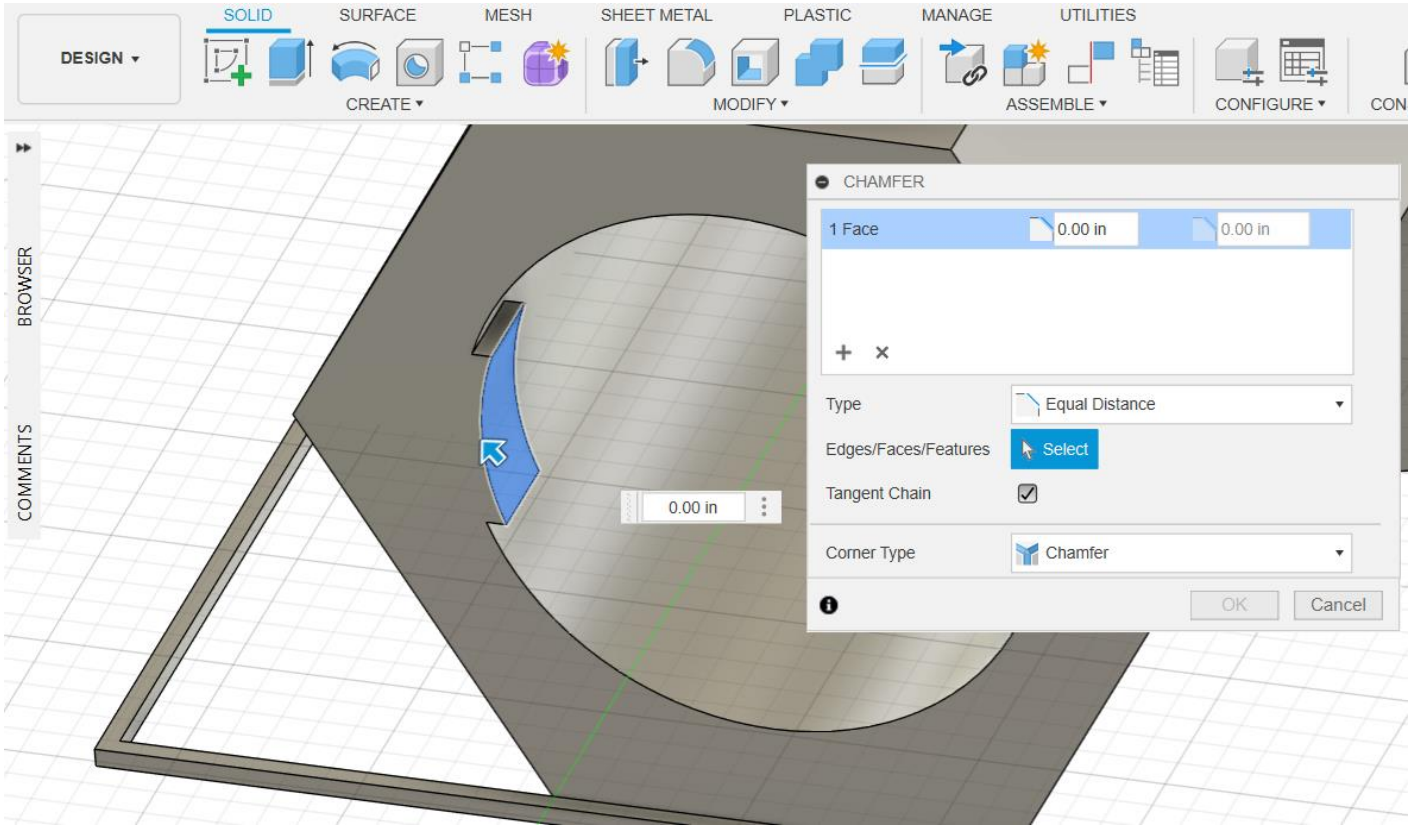
- click Finish Sketch



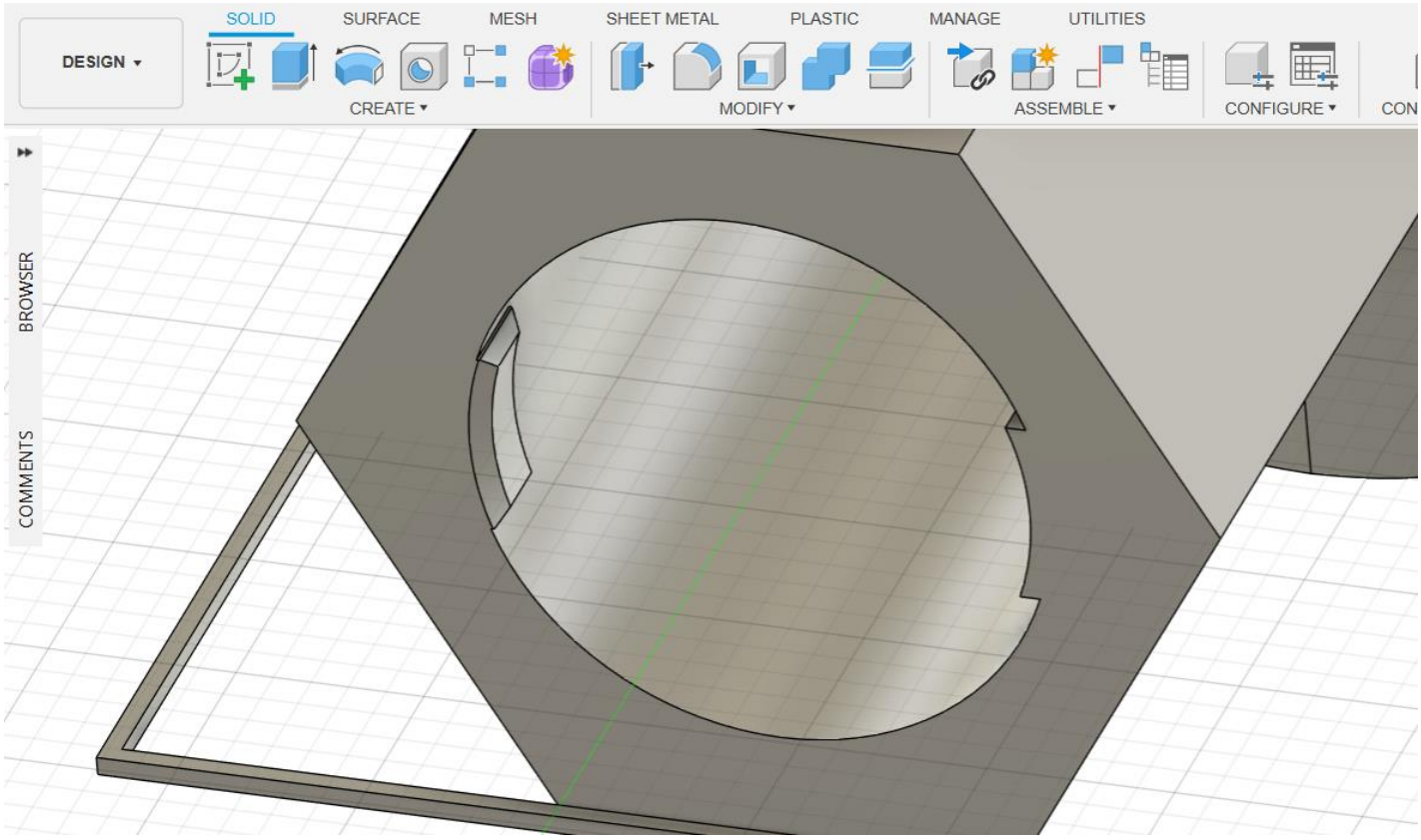
- select the **Extrude** tool
- click on the **two regions** as shown, which will turn them blue
- enter **-0.05** (note the minus sign) and set the **Operation** to **Join**
- click **OK**



- from the **MODIFY** menu select the **Chamfer** tool
- click on the **curved surface**, enter **0.009**, and click **OK**

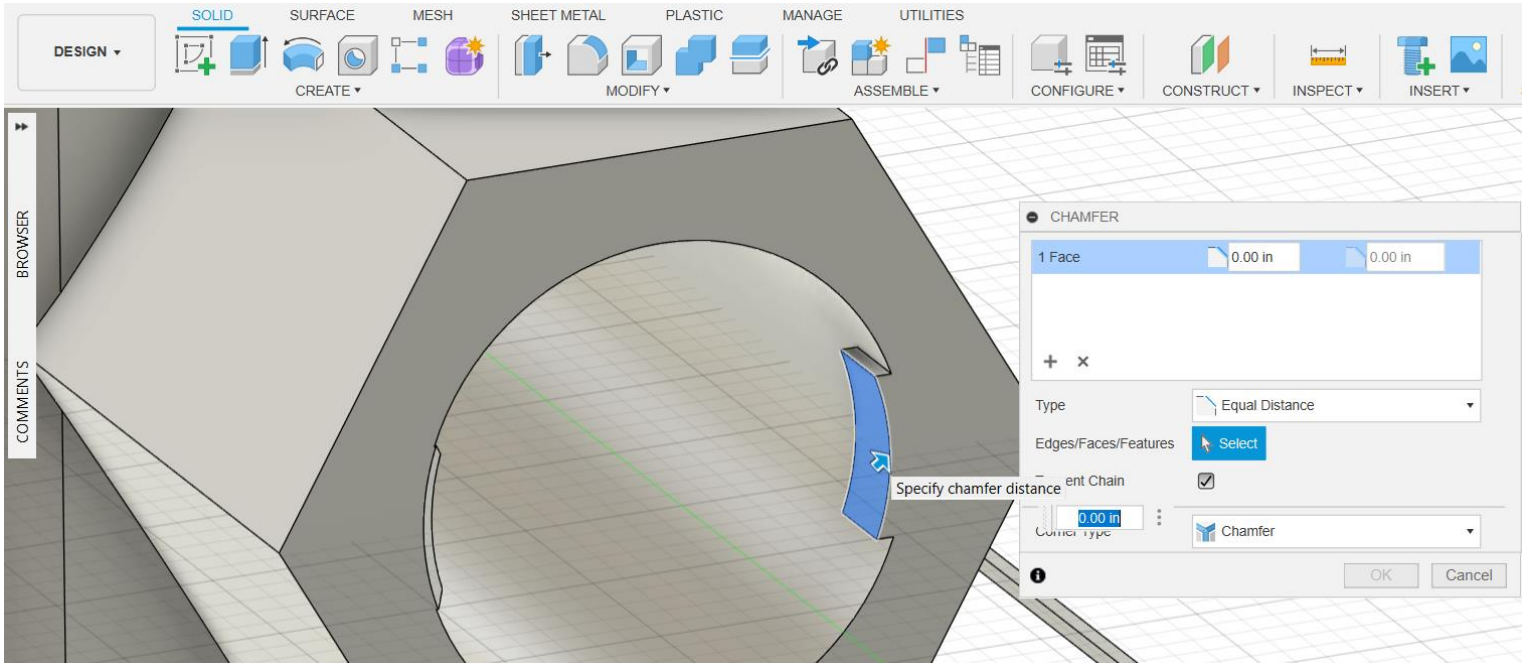


This should be the result of the Chamfer operation.

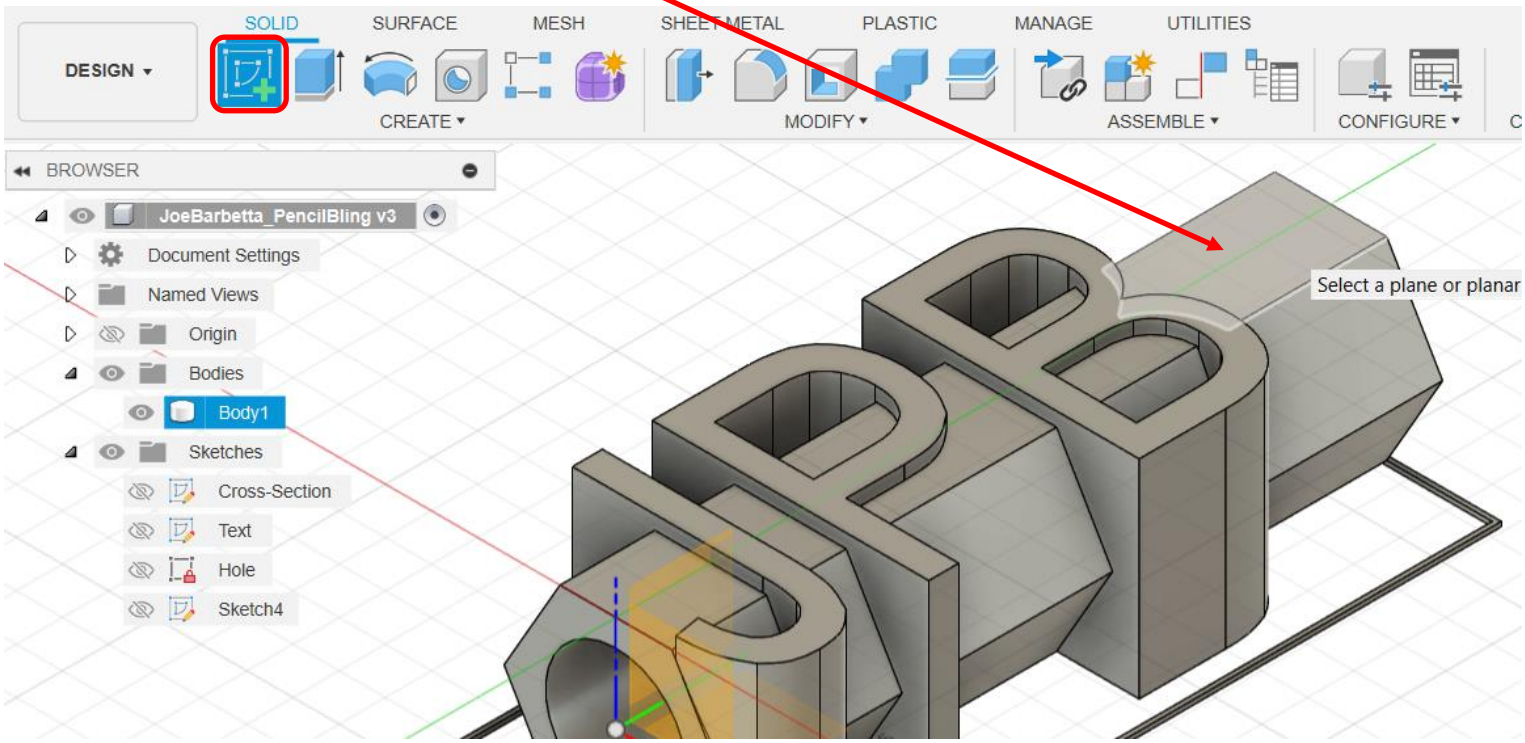


- adjust the view to access the opposite curved surface and **perform the same Chamfer operation**

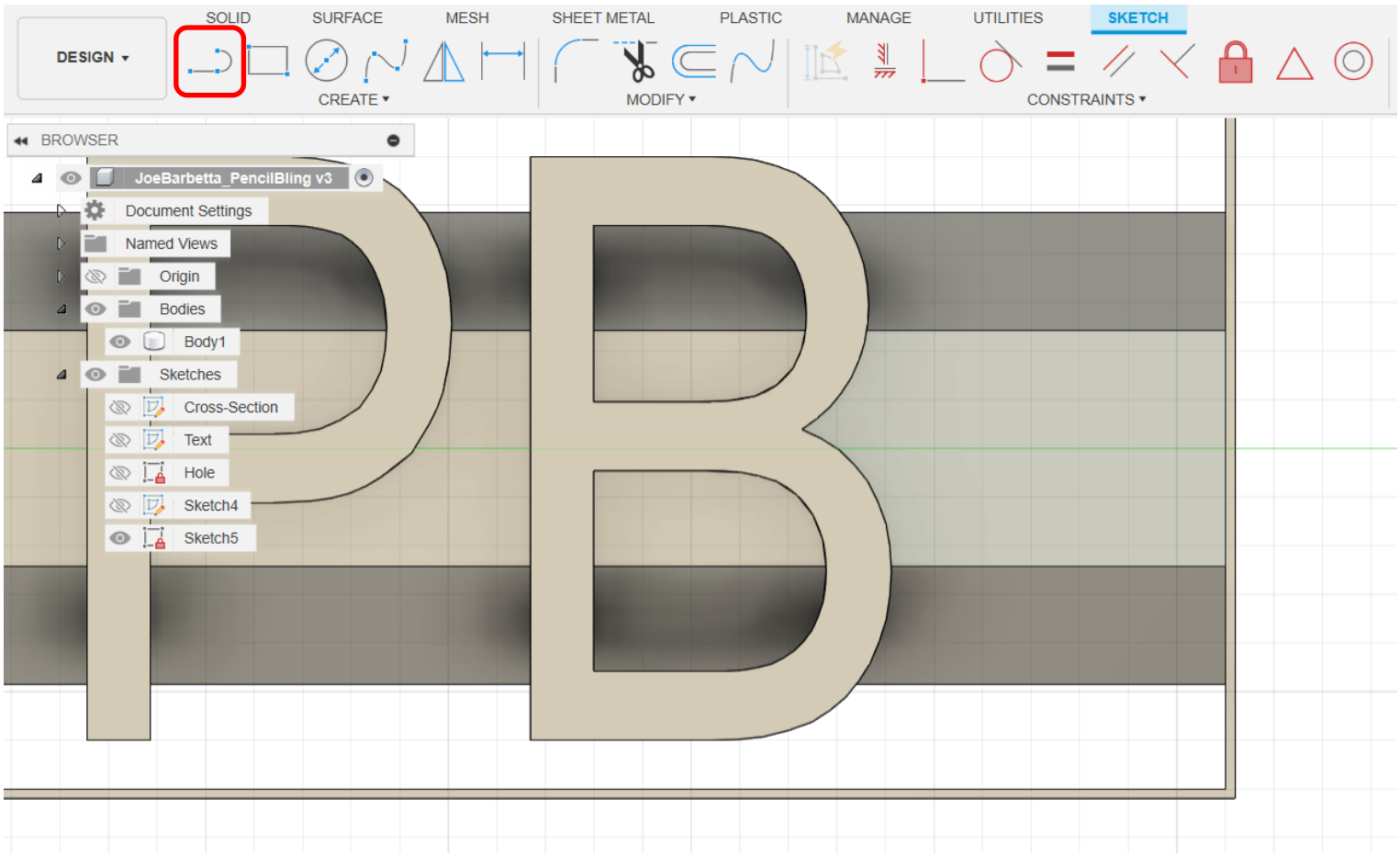
There are now two beveled bumps, which will help hold the bling on the pencil.



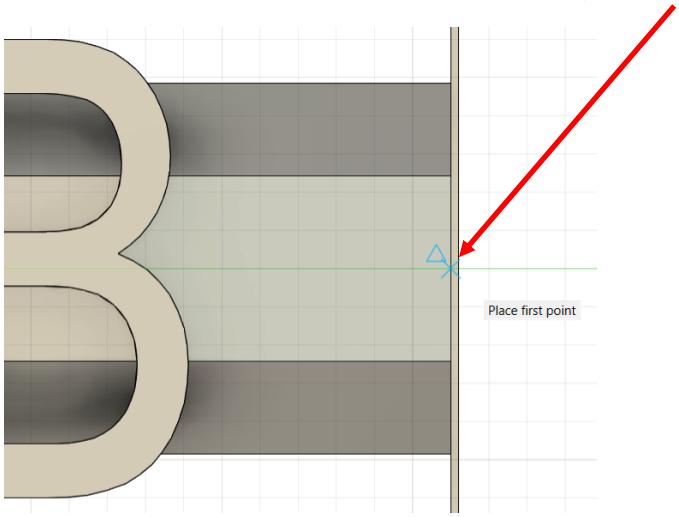
- return to the **Home** view
- select the **Create Sketch** tool and click on the **top surface** of the far end



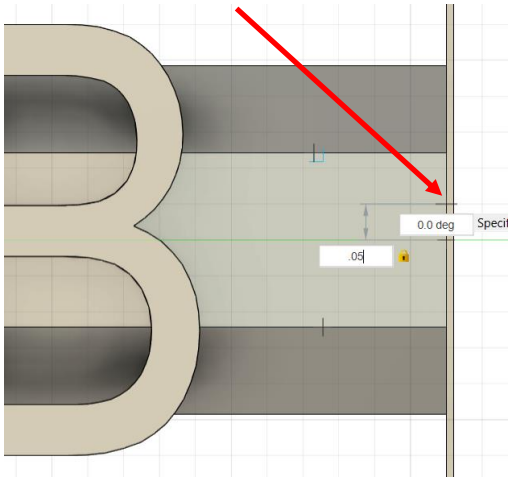
- use the **Rotation Arrows** if needed and **Zoom** into the end as shown
- select the **Line** tool



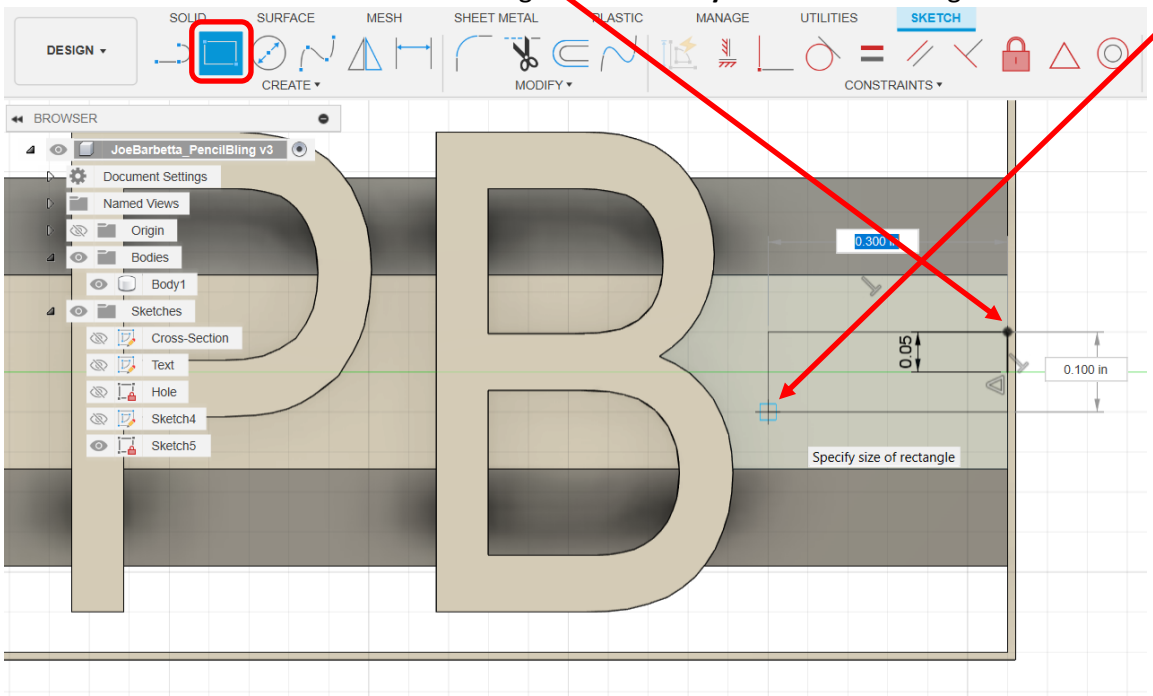
- move the mouse near the end until the **center (blue triangle) shows** and then **click on that point**



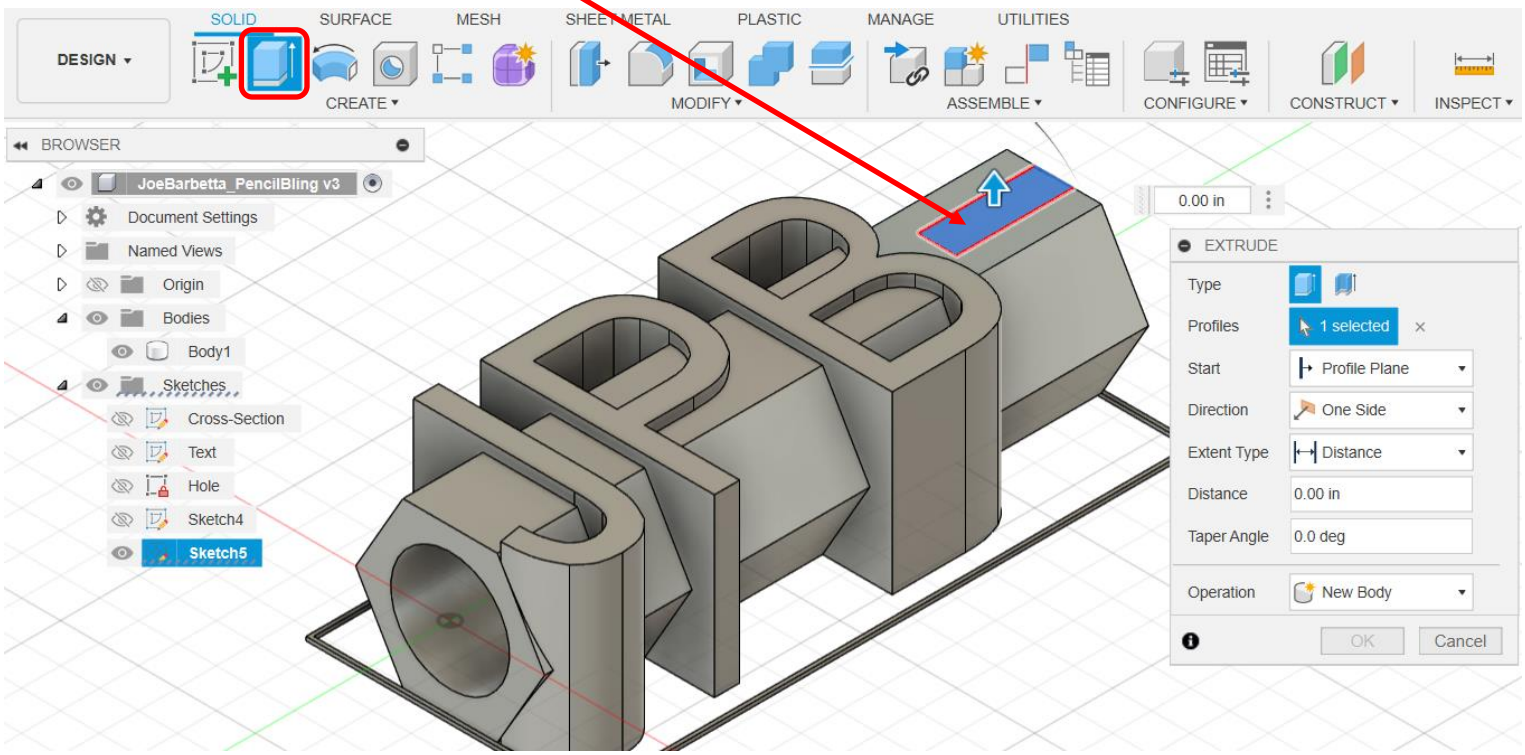
- extend the **mouse up** and enter **0.05**



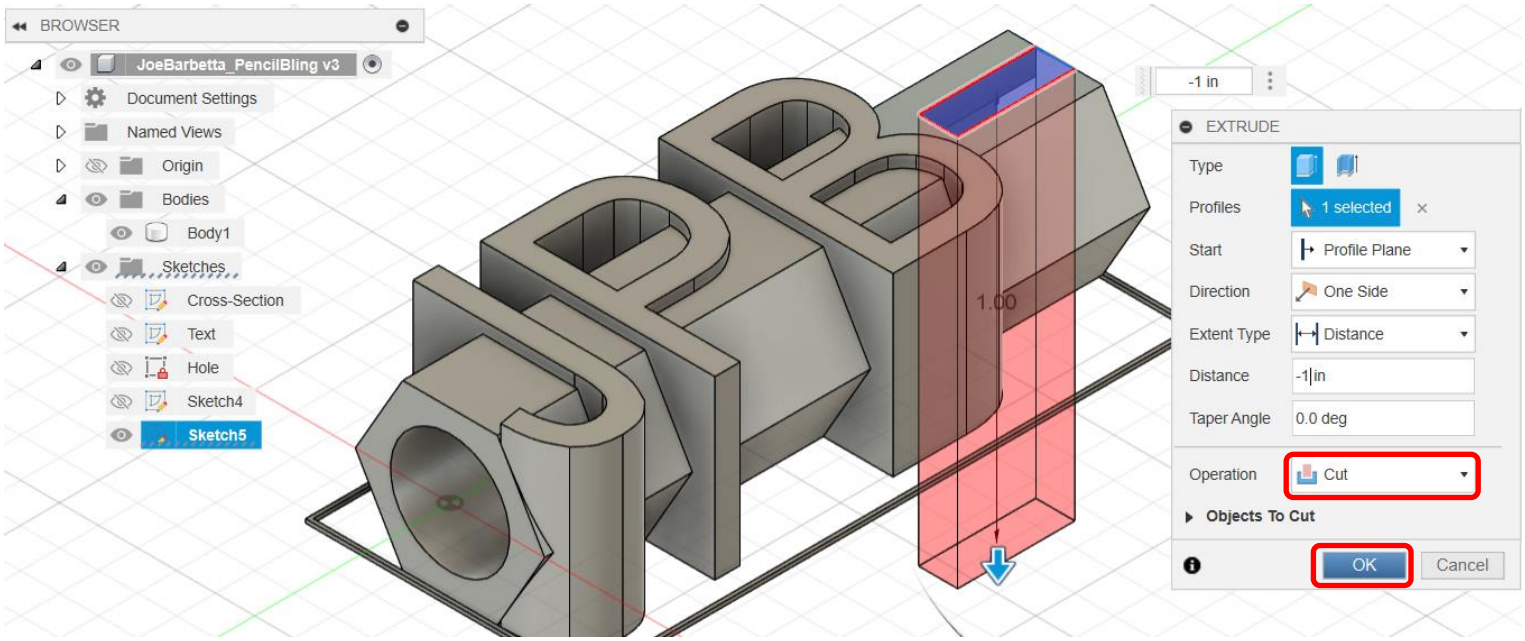
- select the **Rectangle** tool. Click on the **top** of the line just created and extend the rectange **down and to the left**.
- enter **0.3** for the width and **0.1** for the height. The **Tab key** will allow switching between the two dimensions.



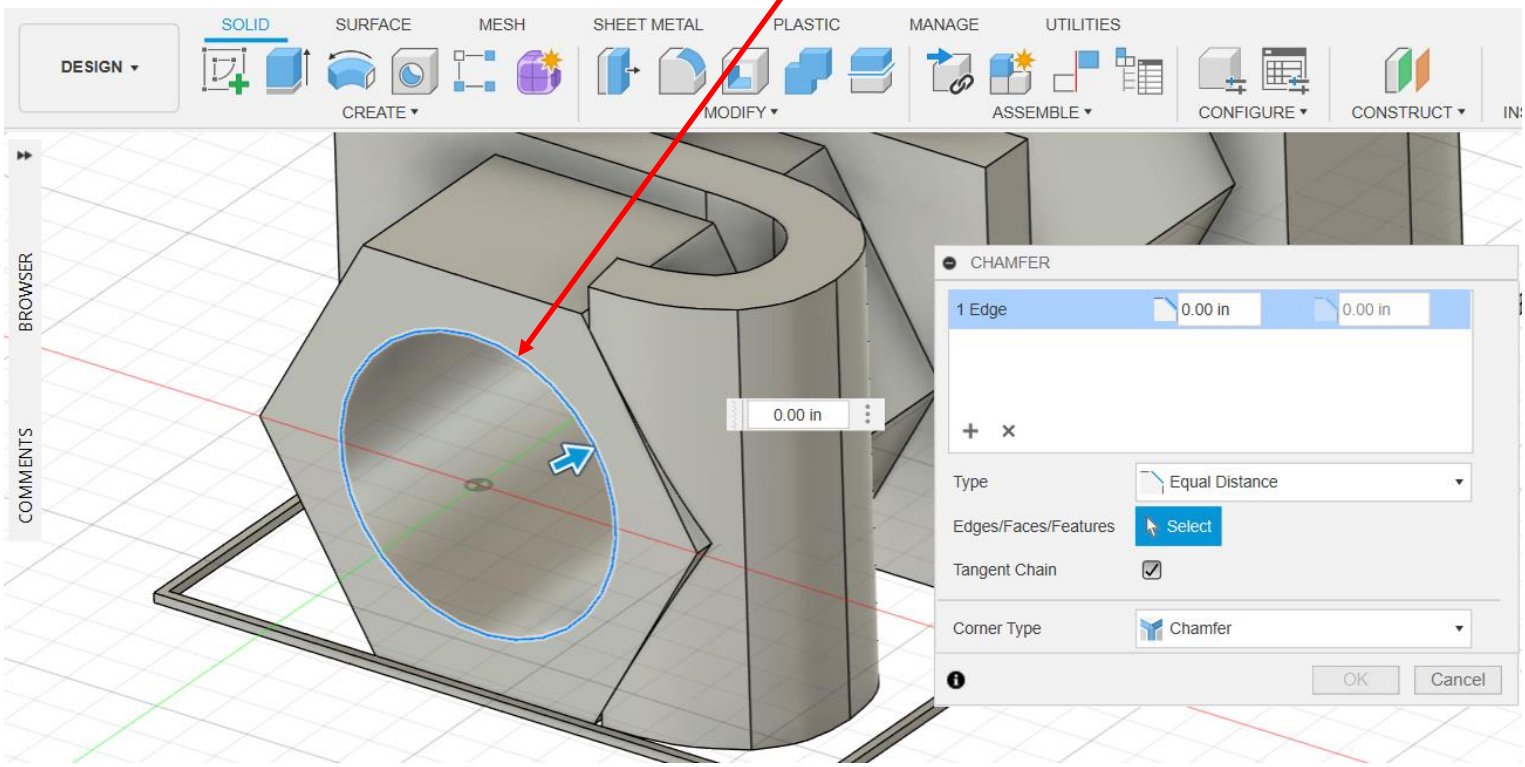
- click **Finish Sketch**
- select the **Extrude** tool and click on the rectangle just created



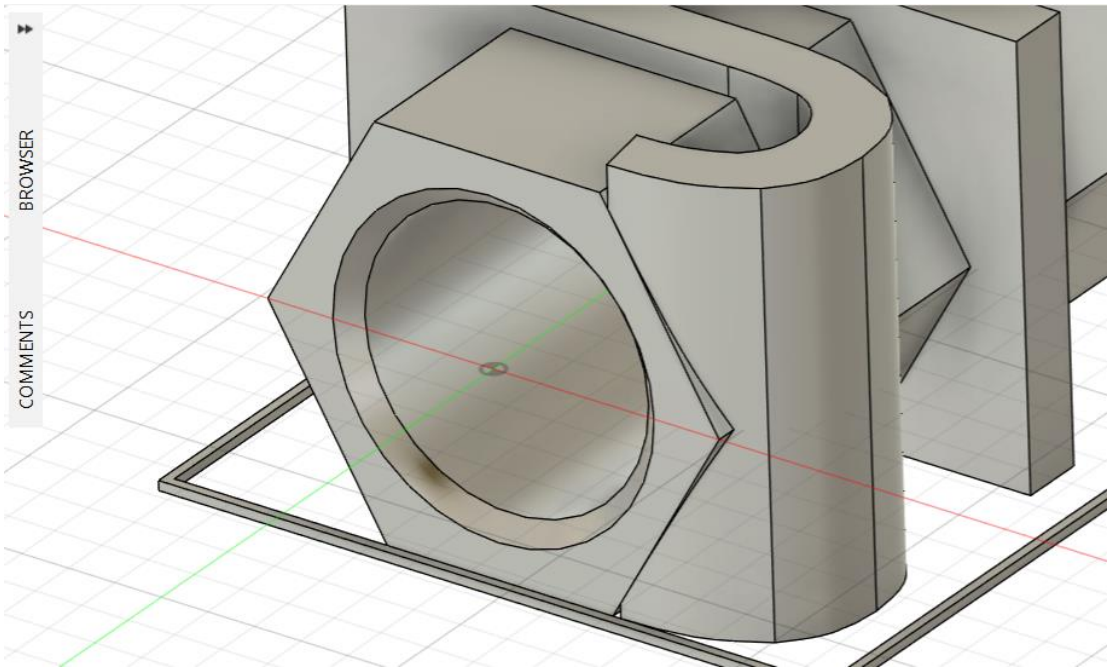
- type **-1** (note the minus sign), ensure the **Operation** is **Cut**, and click **OK**



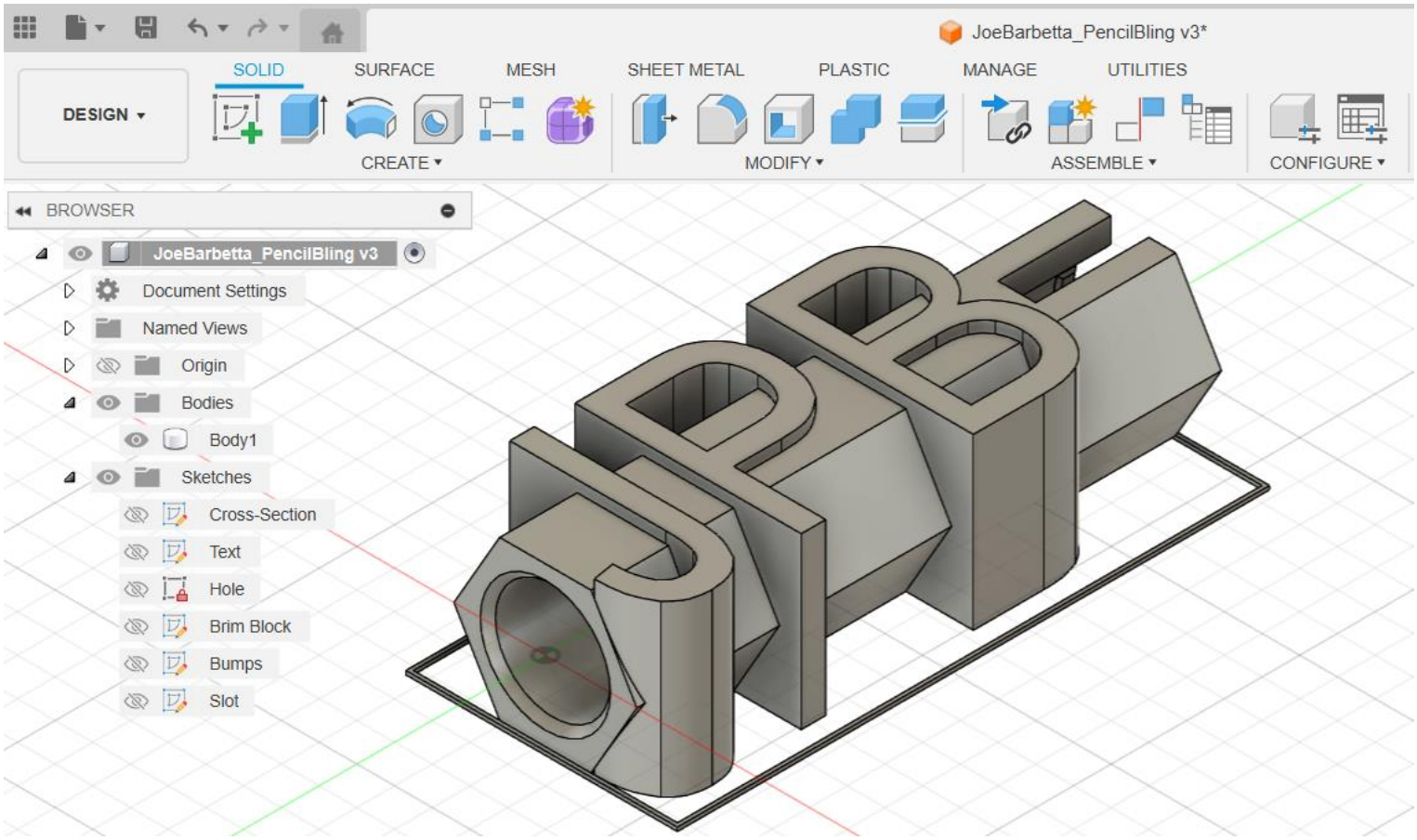
- zoom into the end
- from the **MODIFY** menu select the **Chamfer** tool and click on the **edge** of the hole, which should turn it blue
- type **0.02** and click **OK**



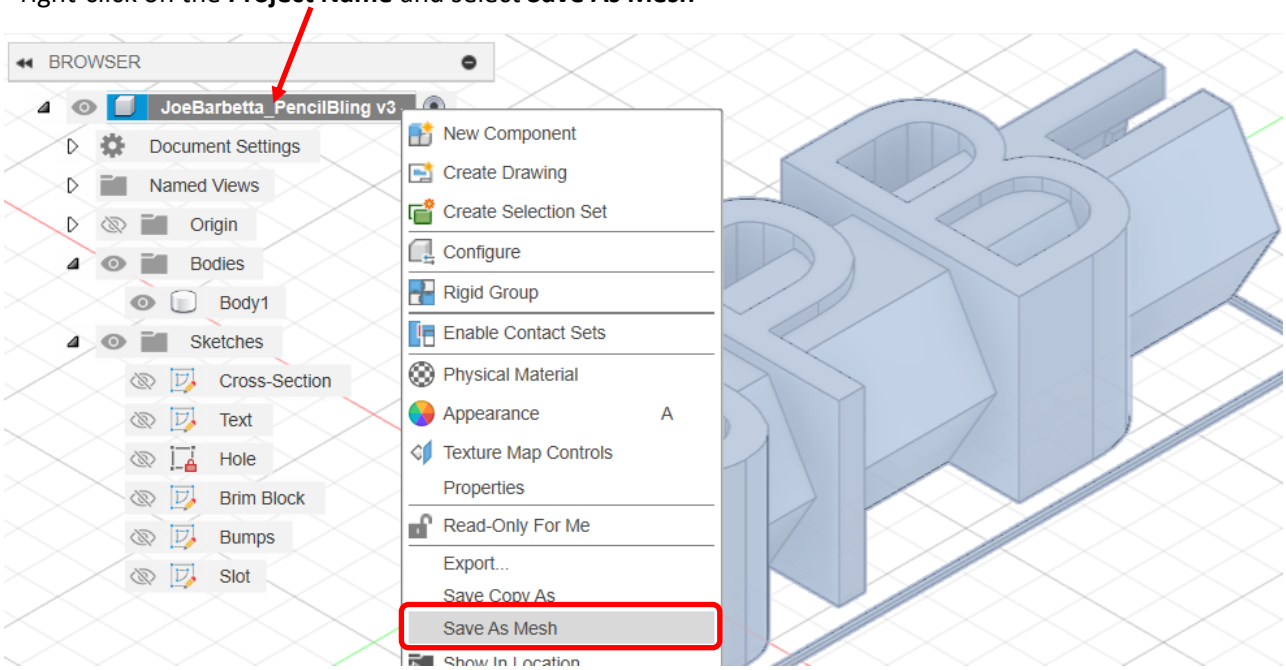
This is how the end should look. The chamfer just added will make it easier to insert a pencil.



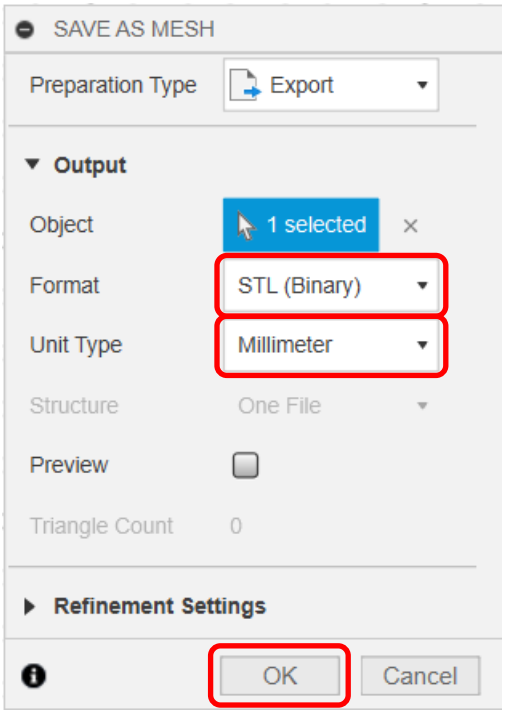
- Admire your pencil bling.
- **Rename the Sketches** that have Not been renamed yet.



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



- ensure the **Format** is set to **STL(Binary)** and **Unit Type** is set to **Millimeter** and click **OK**



- ensure that **Save to my computer** is checked
- the path shown should be the **Downloads** folder, which is the Fusion default. If it is not, one can click the button to the right to select a new location to save the file.
- click **Save**