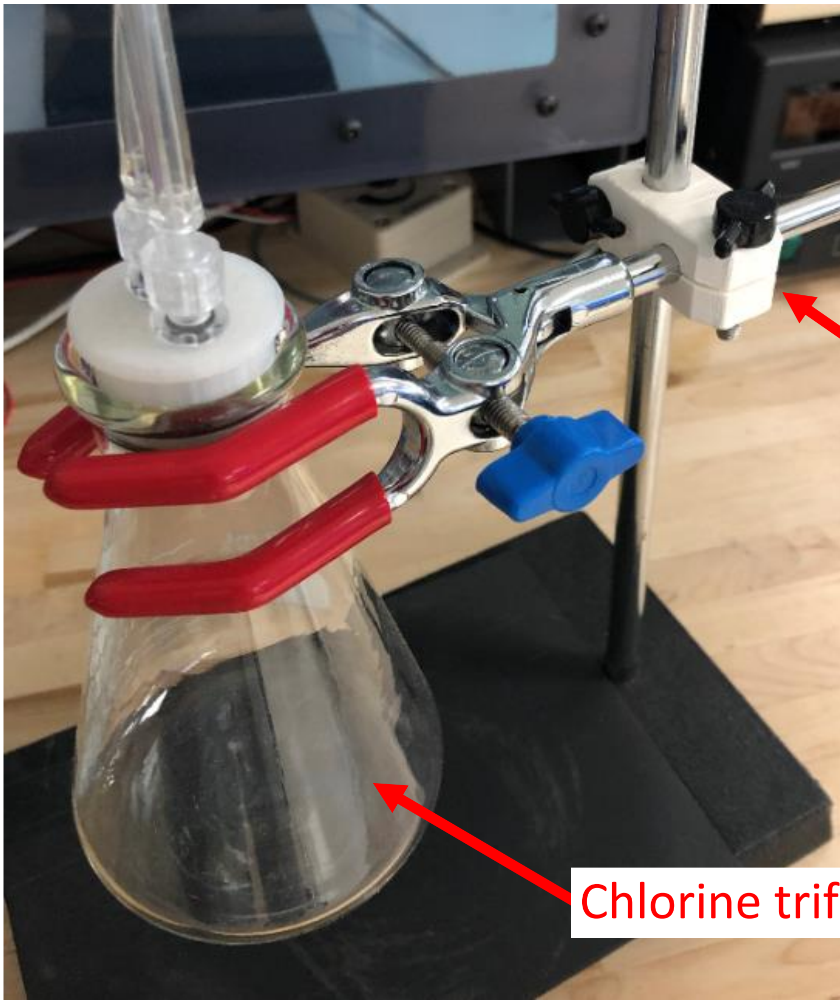
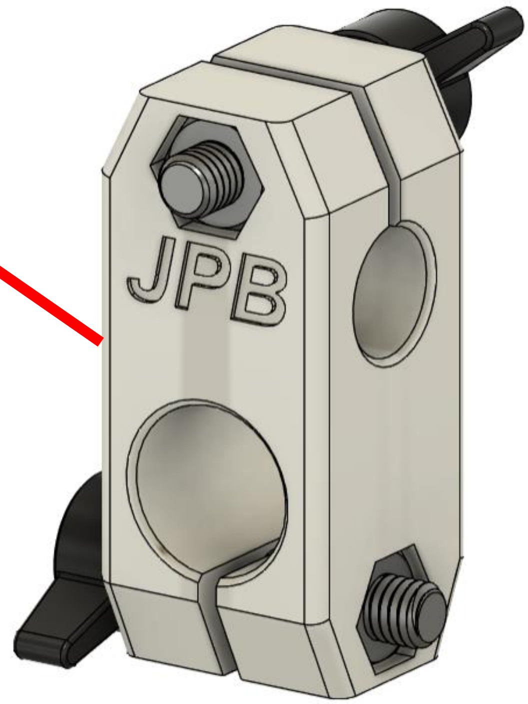
Mr. Barbetta's *"I don't care about 3D Printing. I just want the art credit."* class

Make a laboratory clamp for a mad scientist.





Today's Lesson is Sponsored by Raytheon

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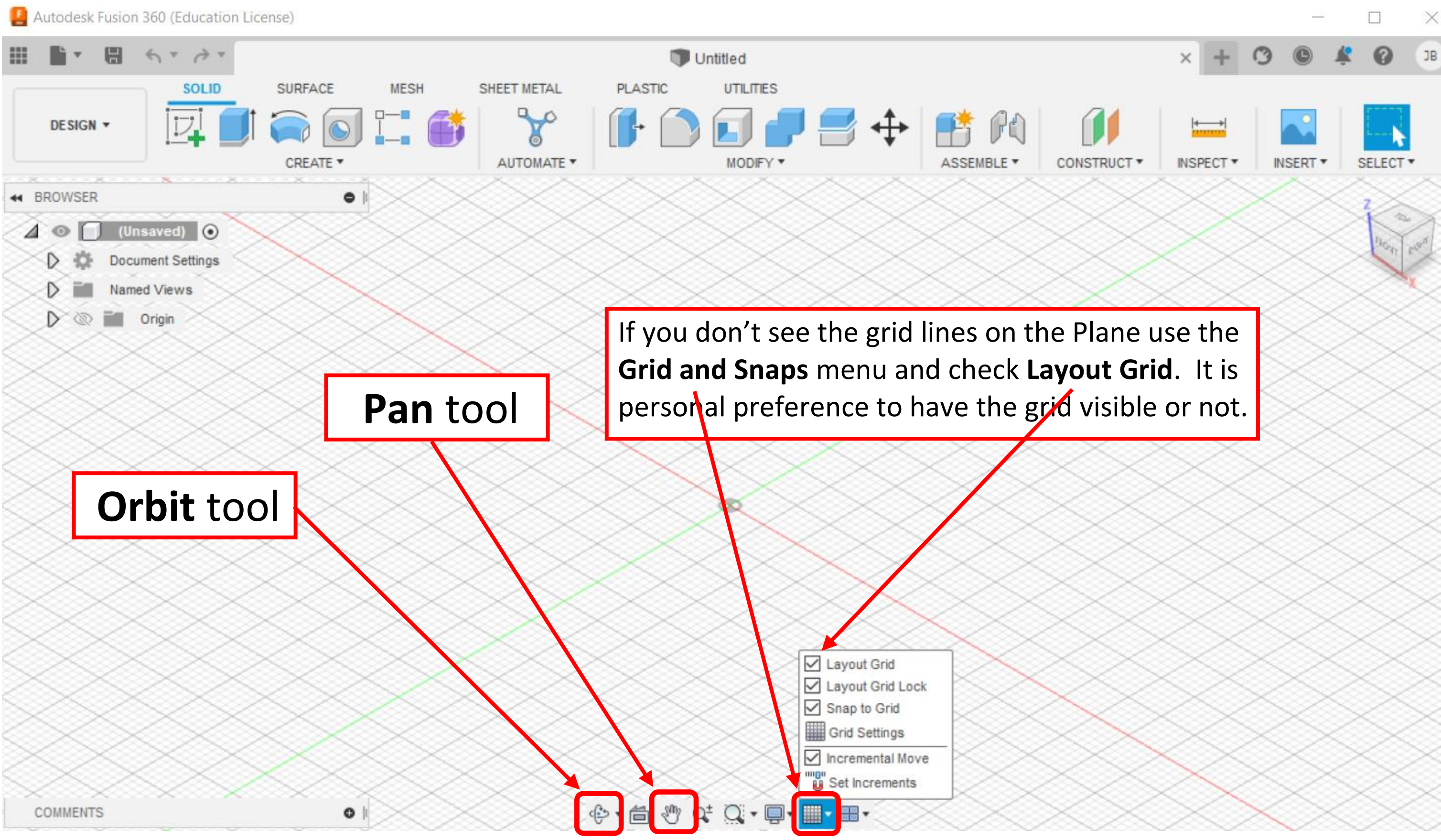
Changing the View of a Design

- 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.

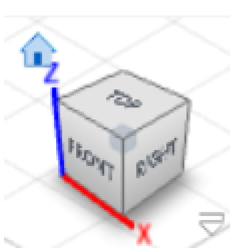


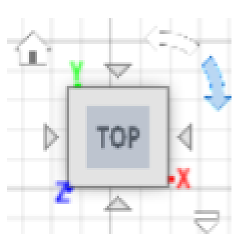
Here is a close-up of the View Cube at the top right of the window.

- 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





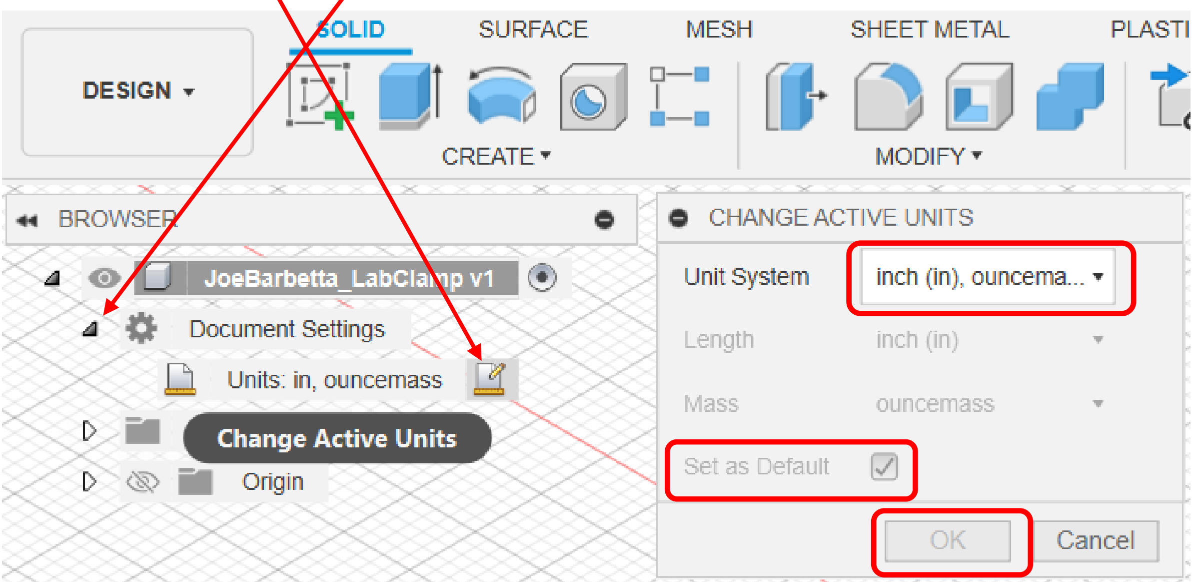
Starting a Design in Fusion (START HERE)

- 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 **_LabClamp** e.g. **JoeBarbetta_LabClamp** (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.

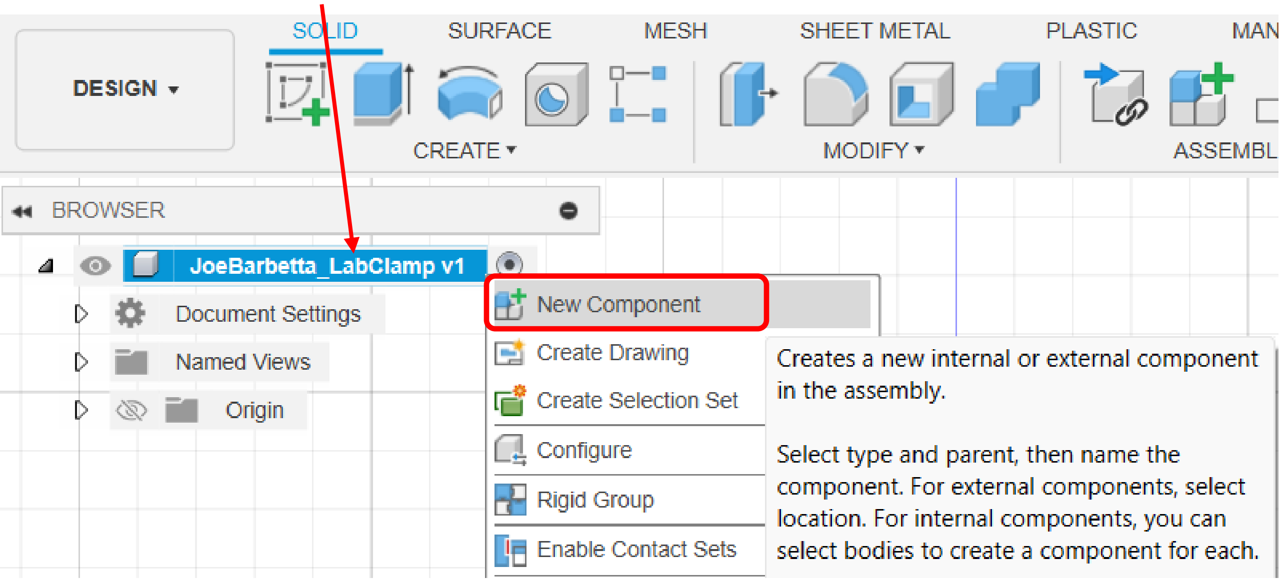


Note that the default units are in mm, which we just changed to inches.

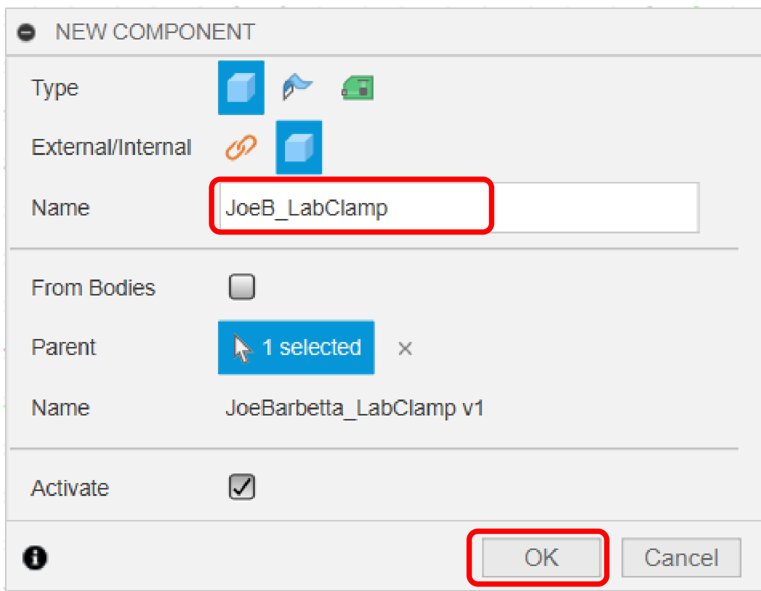
Did you know that the default units have changed over the years? The earliest version used cubits as the default unit.

Creating a Compnent

- right-click on the **Project Name** and select **New Component**

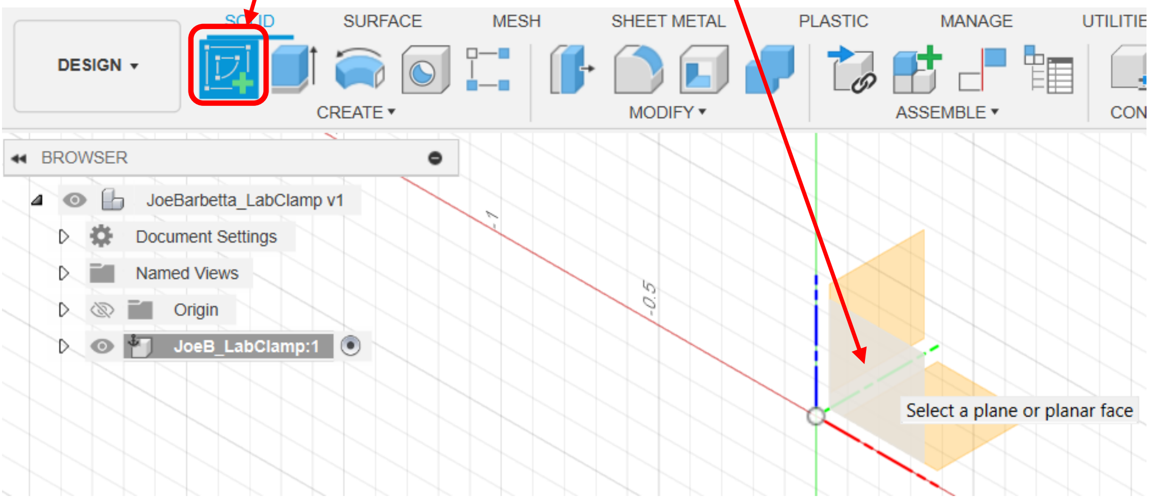


- in the **Name** box enter your **first name followed by your last name initial and _LabClamp** (note the underscore), **e.g. JoeB_LabClamp**
- click **OK**

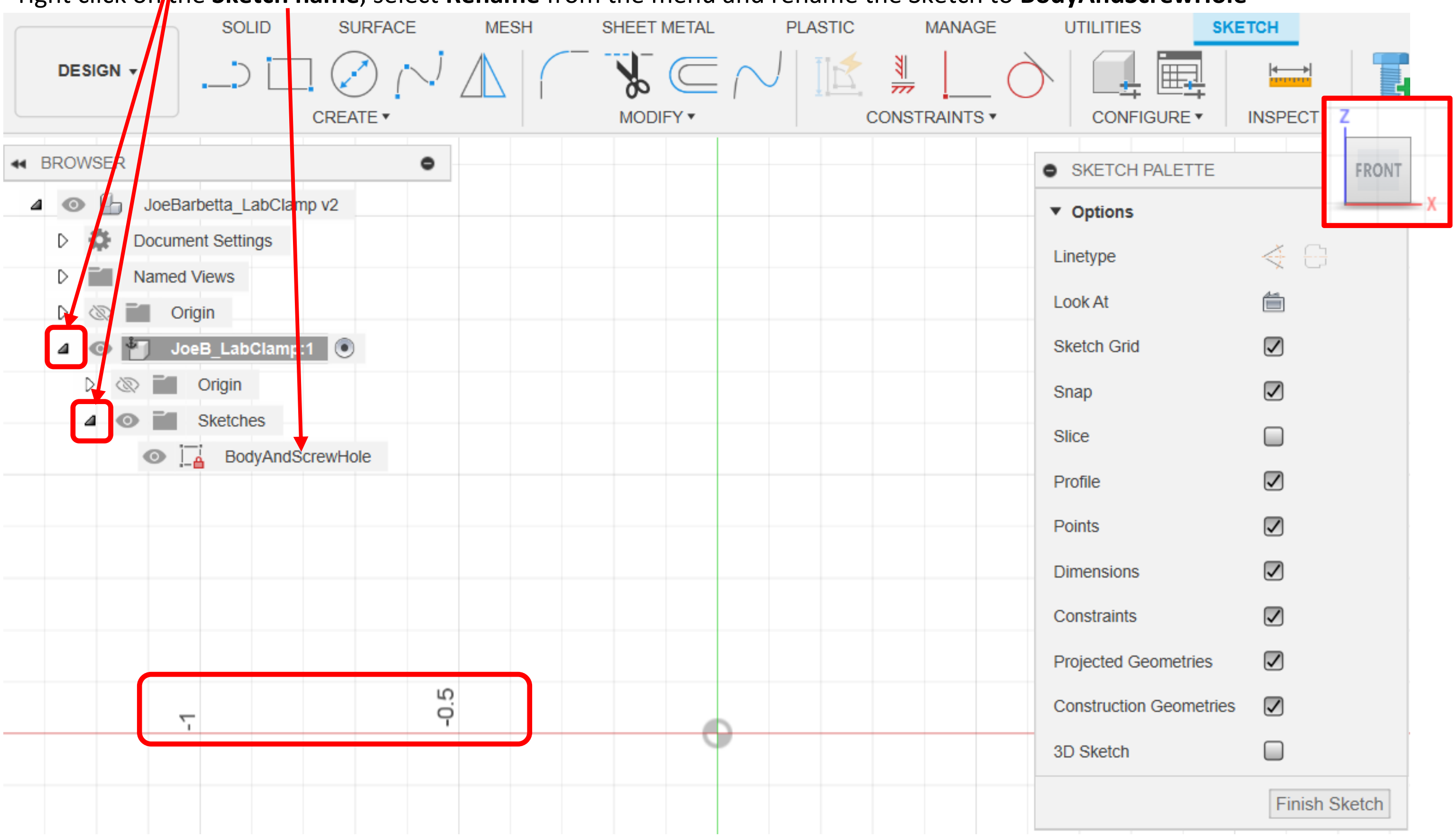


Creating the First Sketch

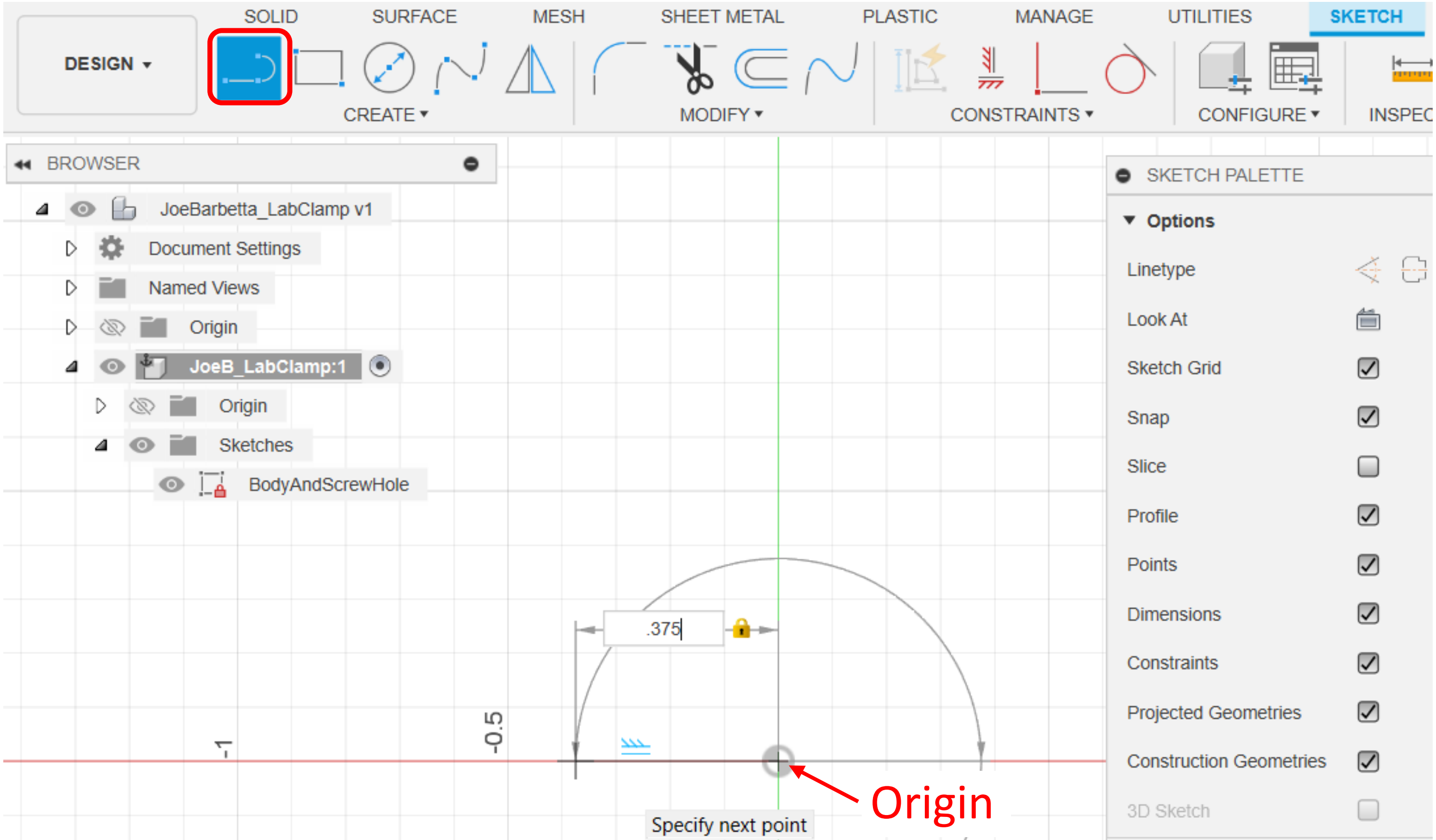
- select the top **Create Sketch** tool and click on the **front rhombus** to select the X-Y Plane.
If a tool can't be found, one can always look in the **CREATE** and **MODIFY** menus for it.



- 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 **arrows** next to the **Component** and **Sketches** folder to open them
- right click on the **Sketch name**, select **Rename** from the menu and rename the Sketch to **BodyAndScrewHole**

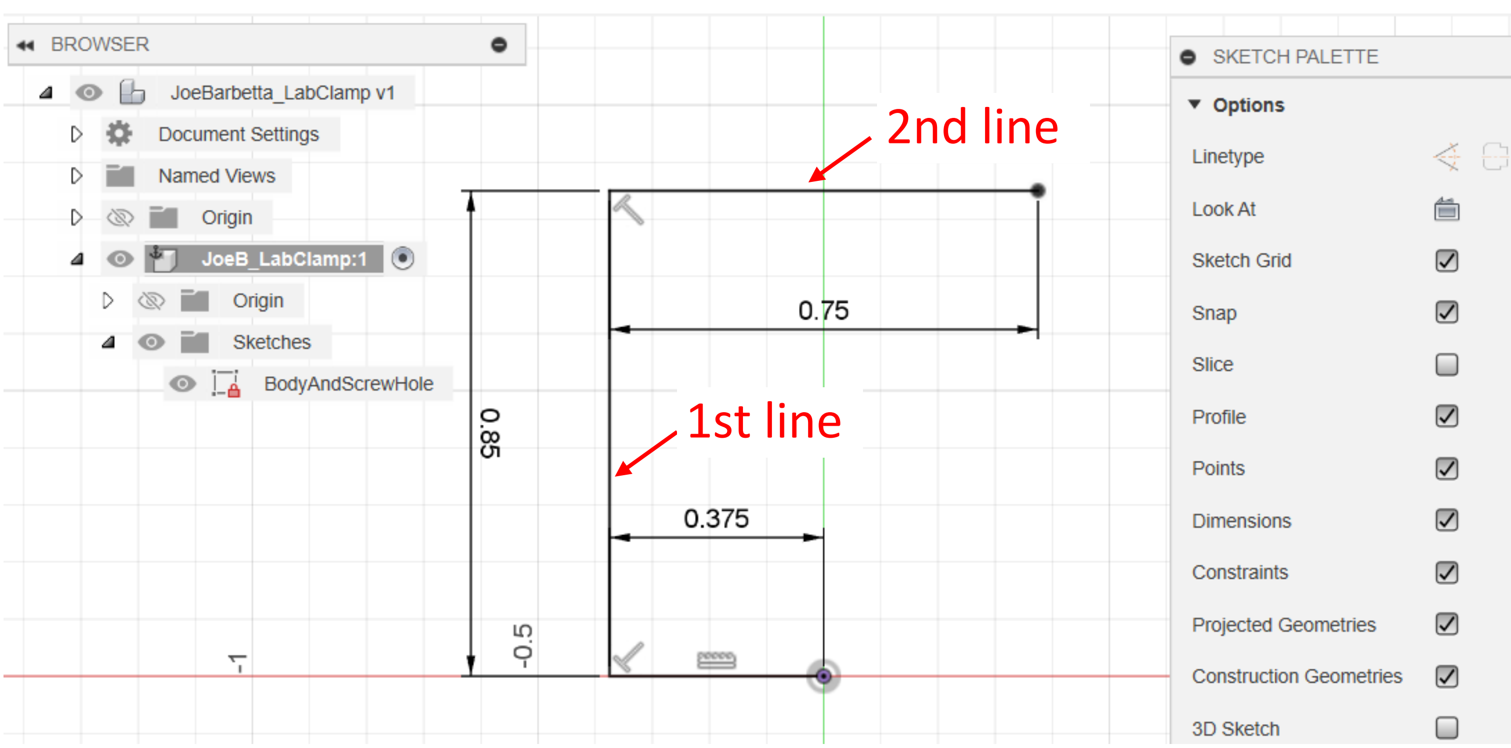


- select the **Line** tool
- click on the **Origin**, extend the line to the left, type **0.375**, and press the **Enter key**

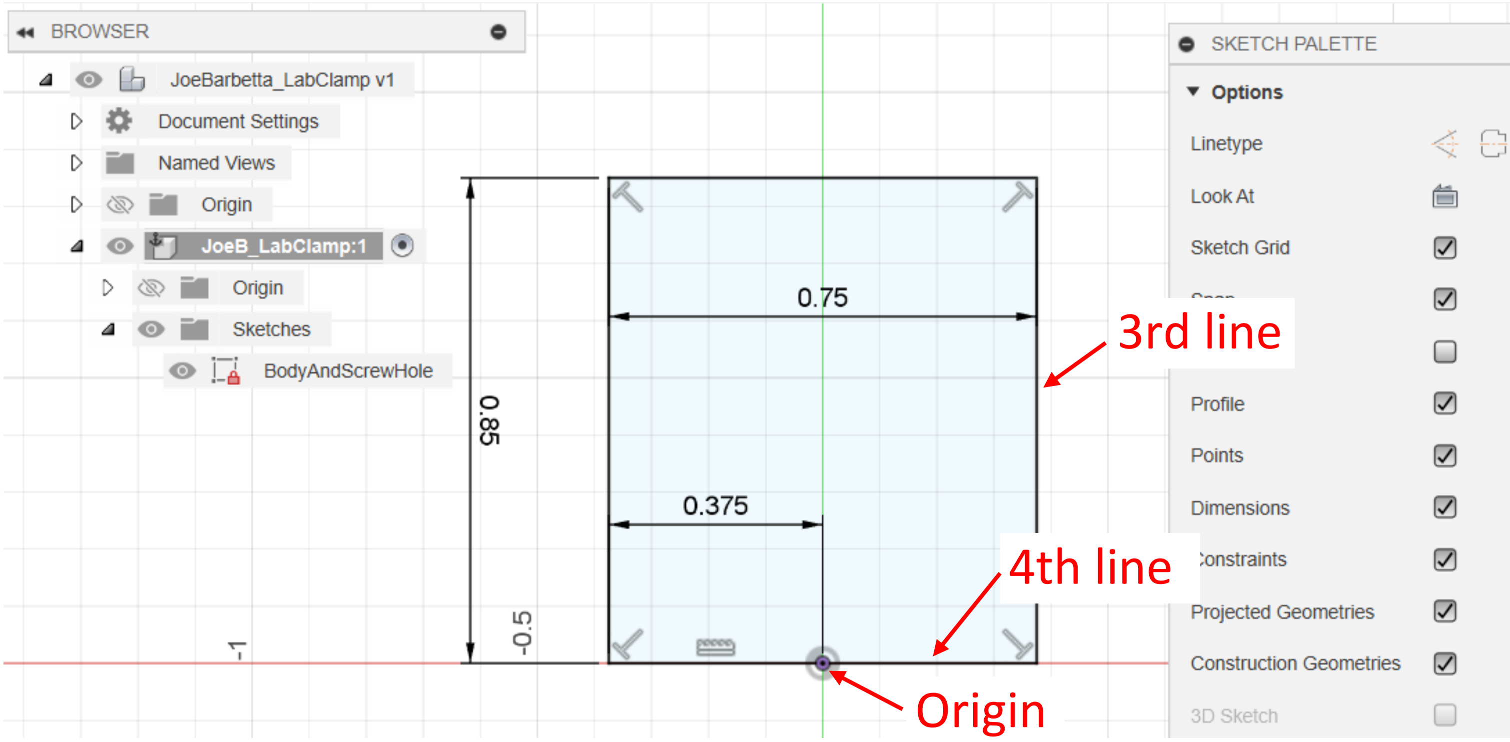


- select the **Line** tool again and create a line **starting from the left end of the line just created** and up by **0.85**
- create another line from the **top of that line** and to the right by **0.75**

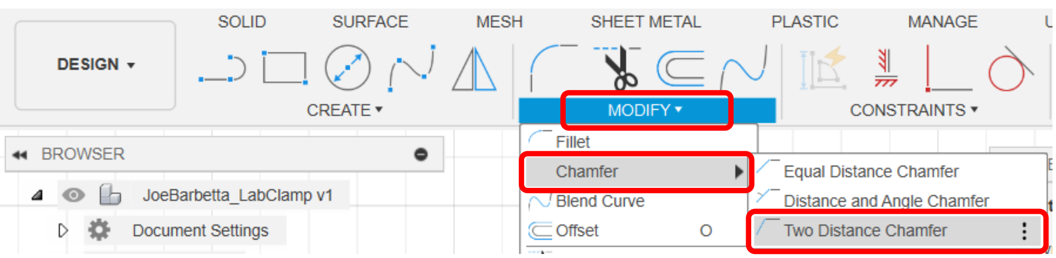
Note that it is OK if the dimension lines (thinner lines with arrows) look different



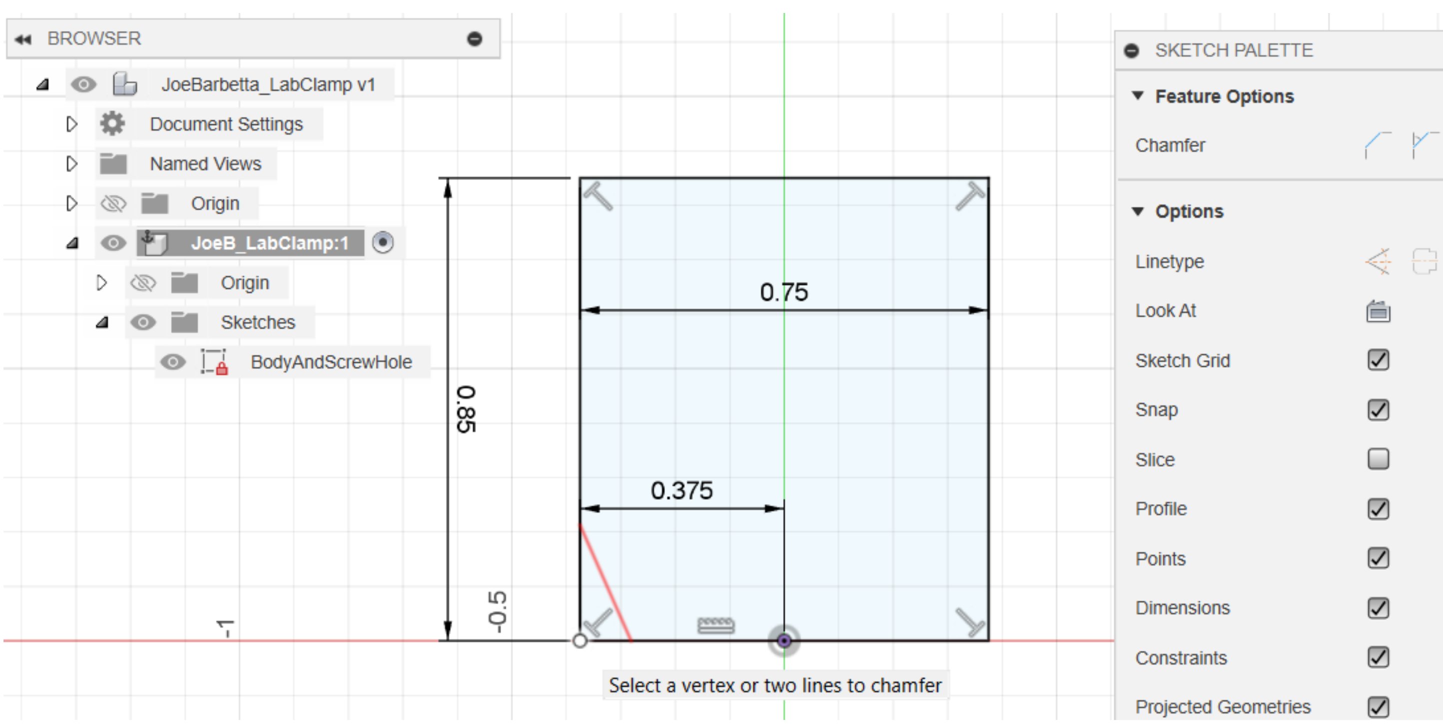
- create a 3rd line **downward from the last point to the red axis**
- create the 4th line from that **last point back to the Origin**. The shape should fill with light blue



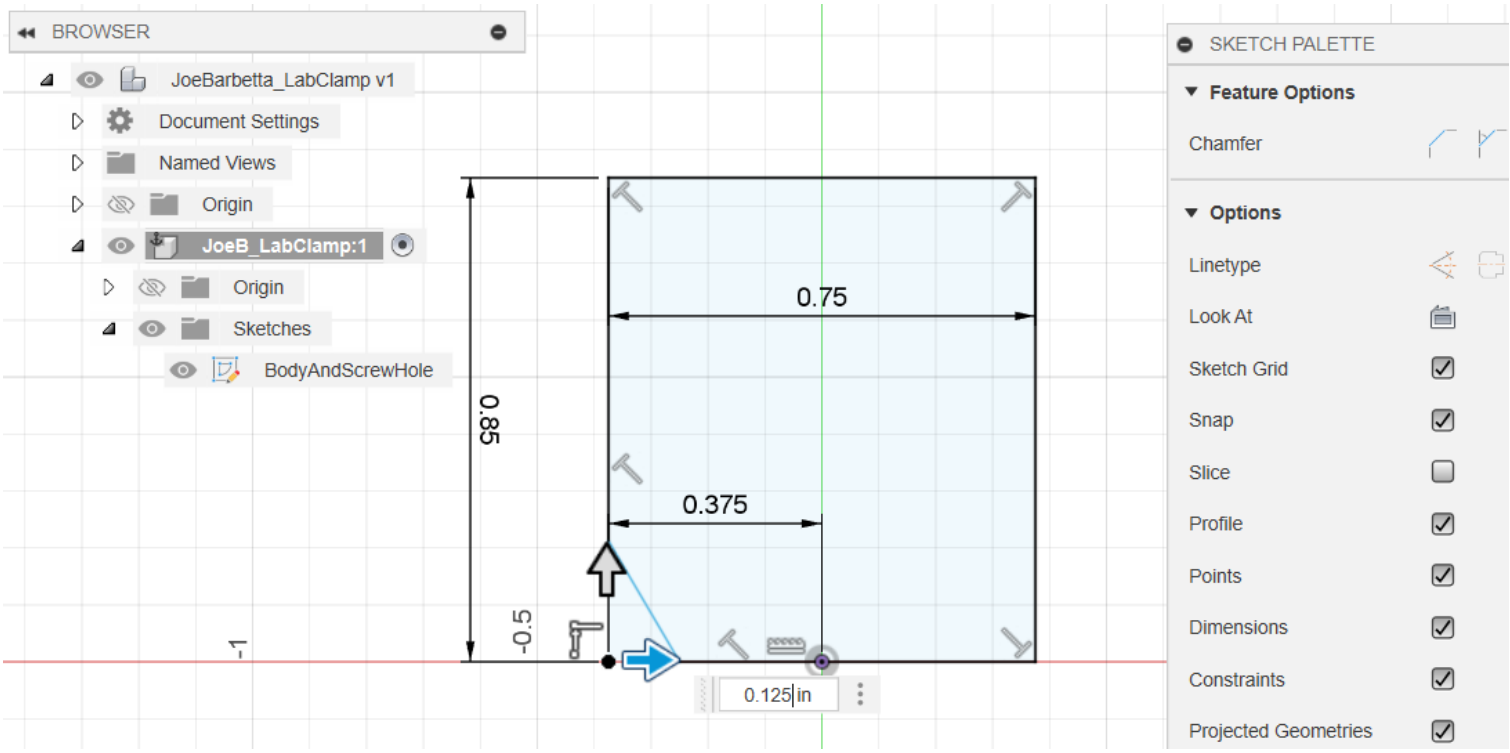
- from the **MODIFY** menu, select **Two Distance Chamfer**



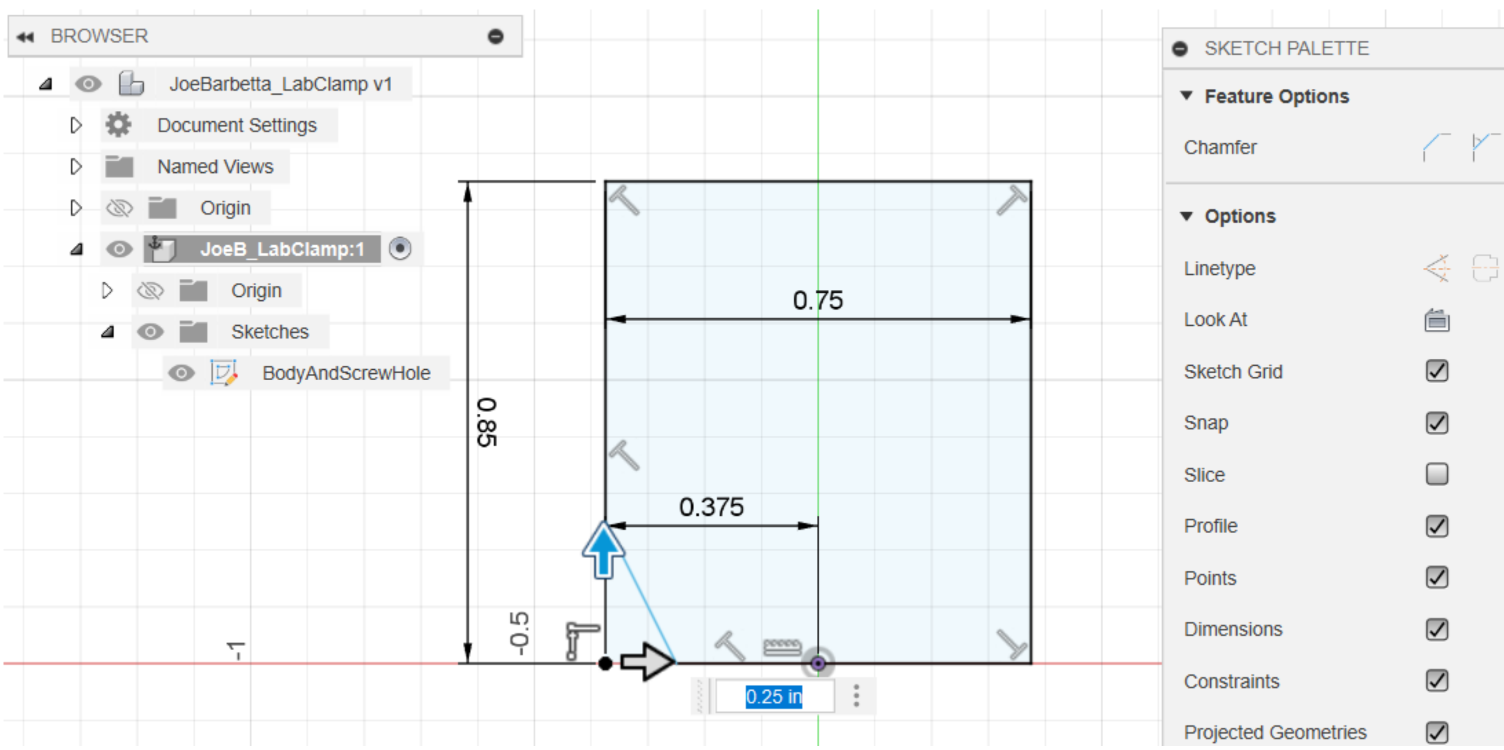
- click on the bottom right corner of the shape. Any warnings that appear can be ignored when creating **Chamfers**.



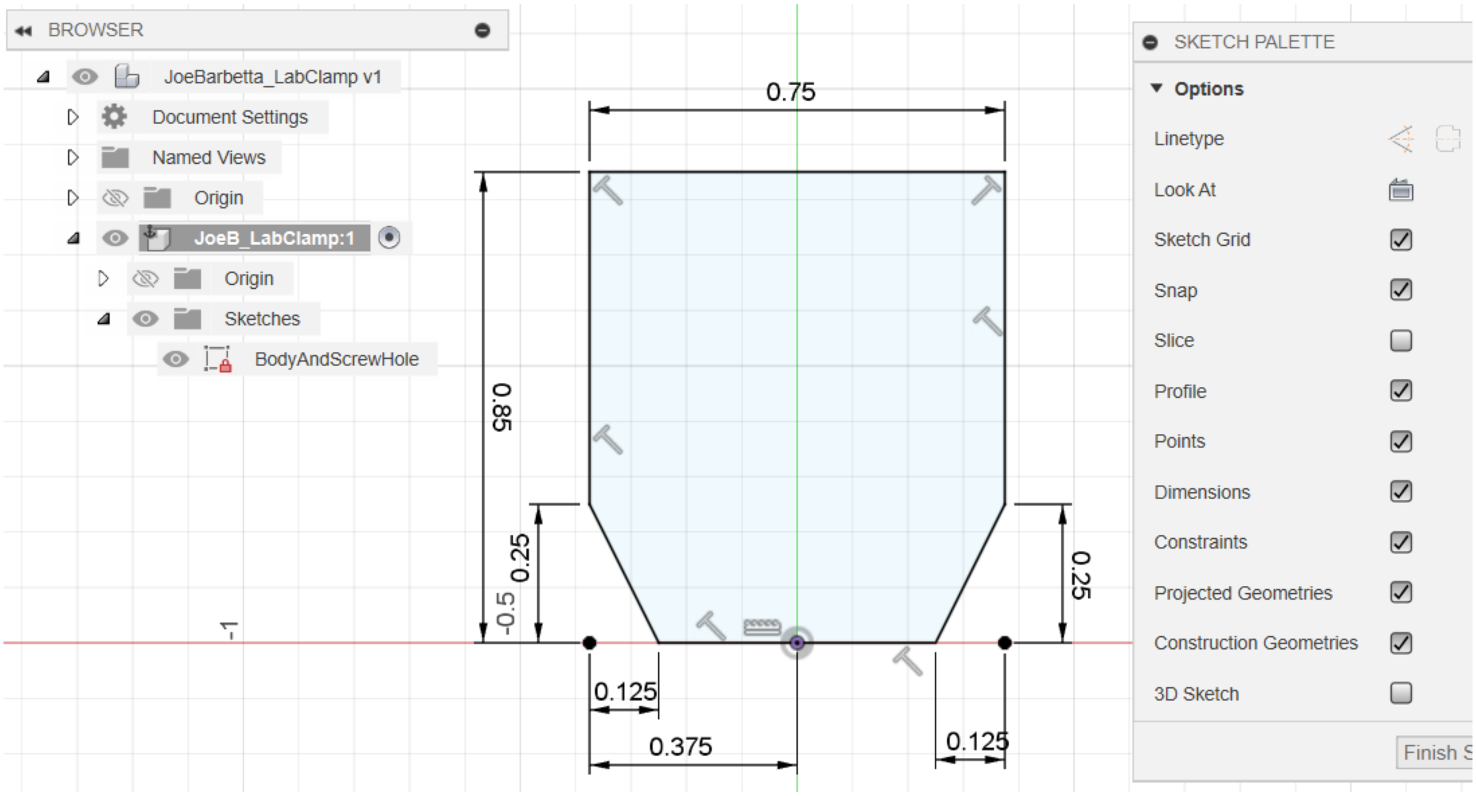
-click the the **horizontal arrow** and enter **0.125**



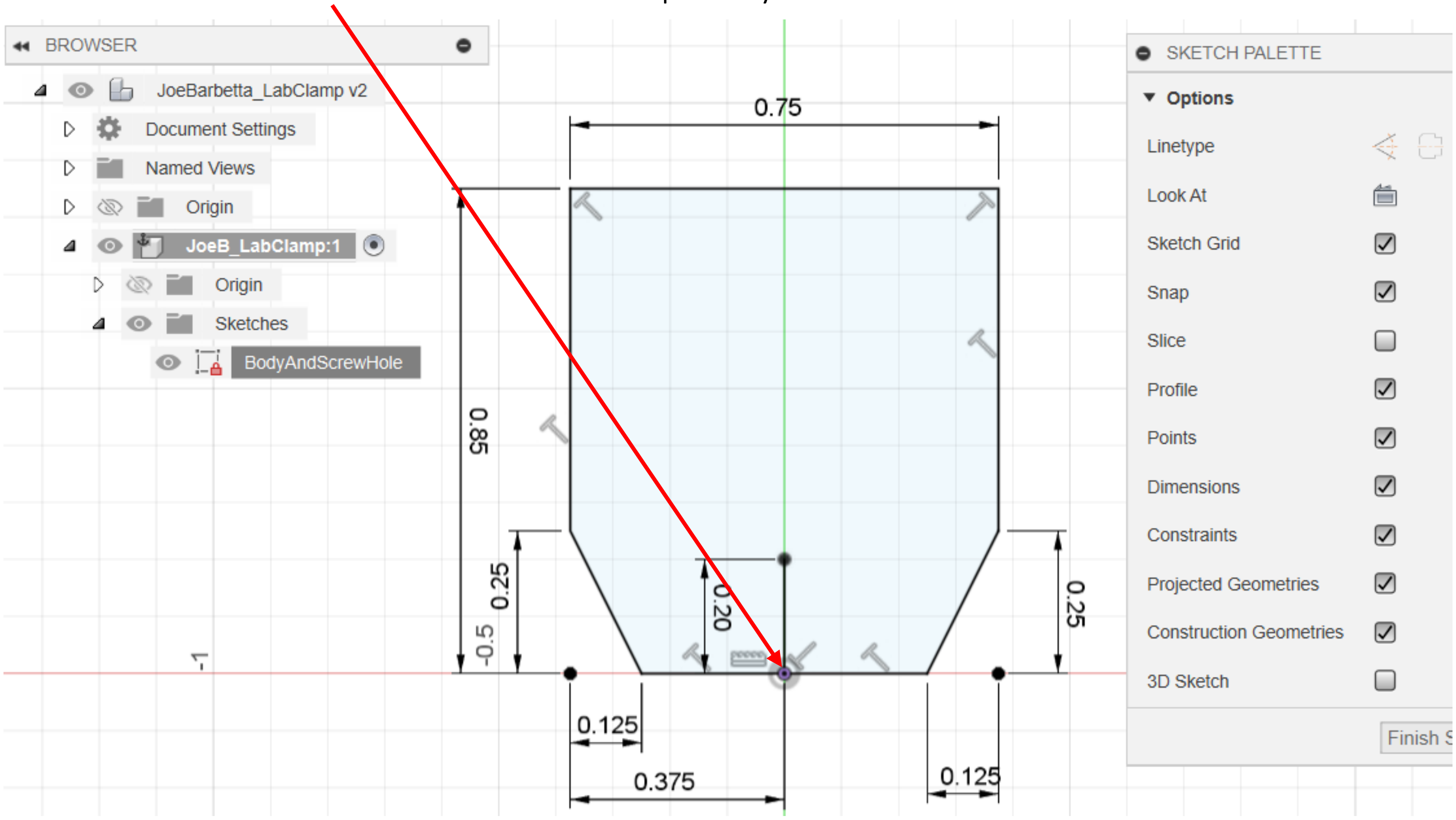
-click the the **horizontal arrow** and enter **0.25**



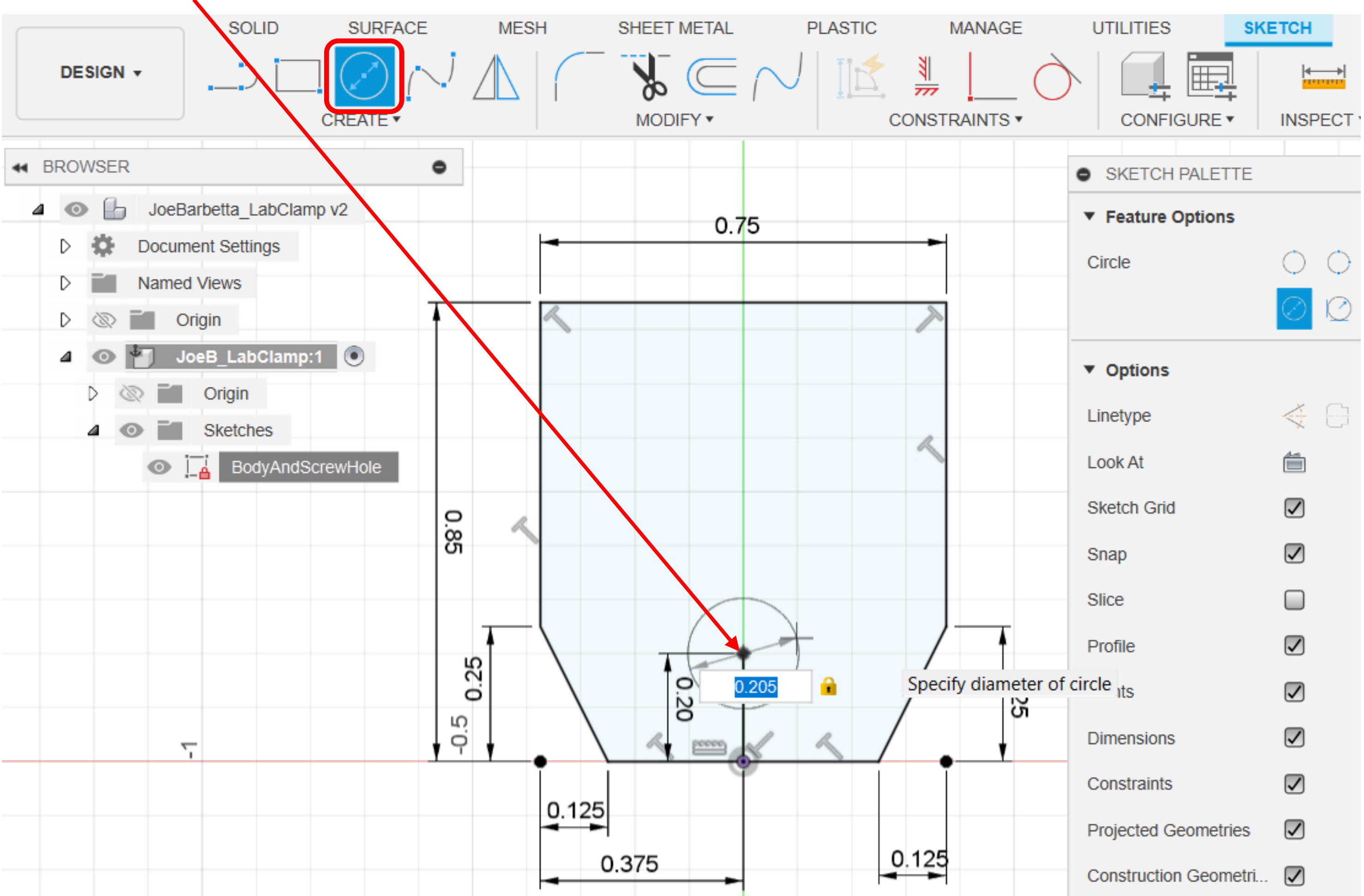
- perform the same **Chamfer** operation on the **bottom right corner**
- drag the different **Dimension values** to organize the dimensions similar to that below



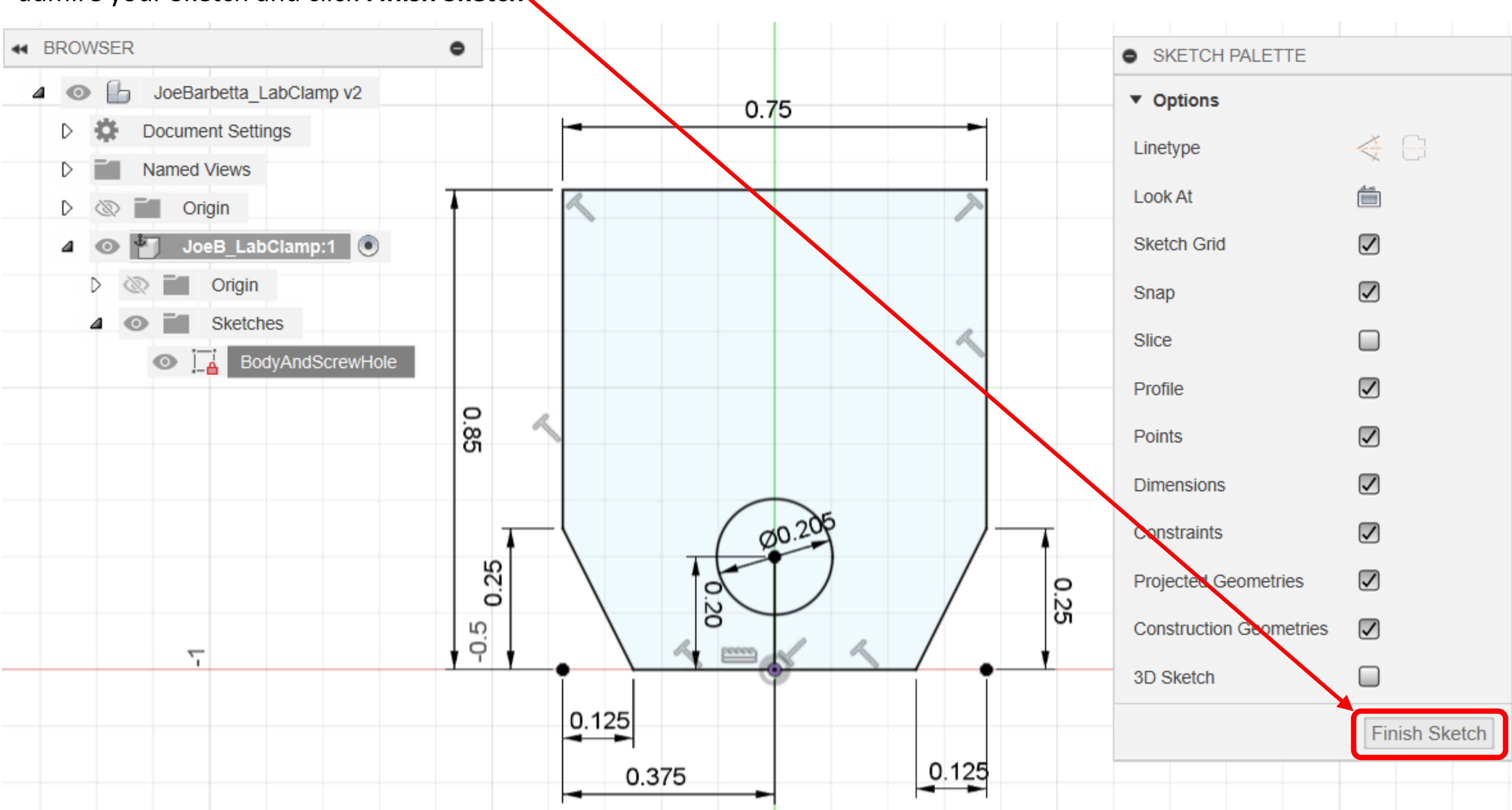
- create a line from the **center of the bottom line** and upward by **0.200**



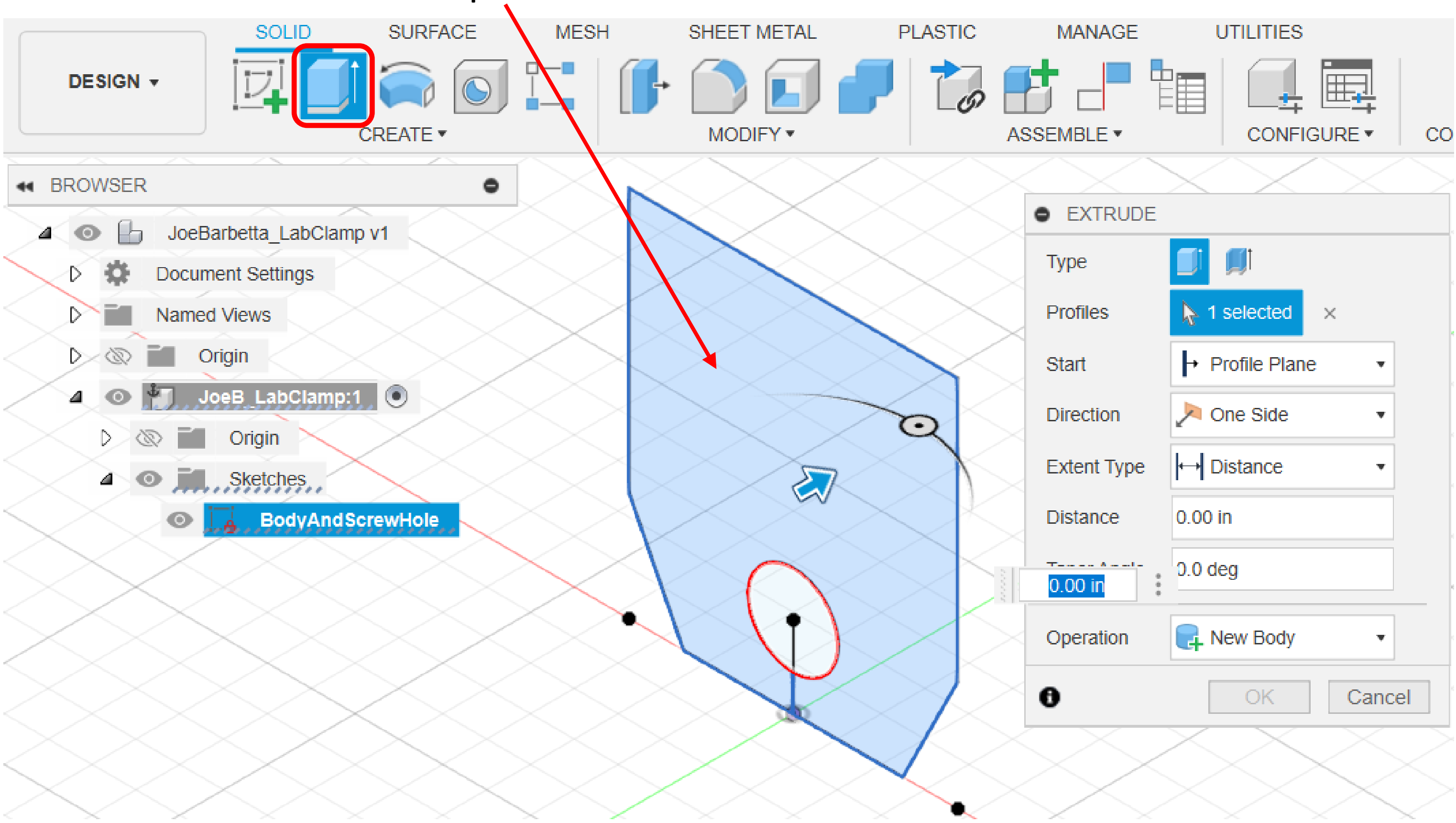
- select the **Center Diameter Circle** tool. If it is not visible, find it in the CREATE menu.
- click on the **top of the last line created**, extend the circle outward, type **0.205**, and press the **Enter key**



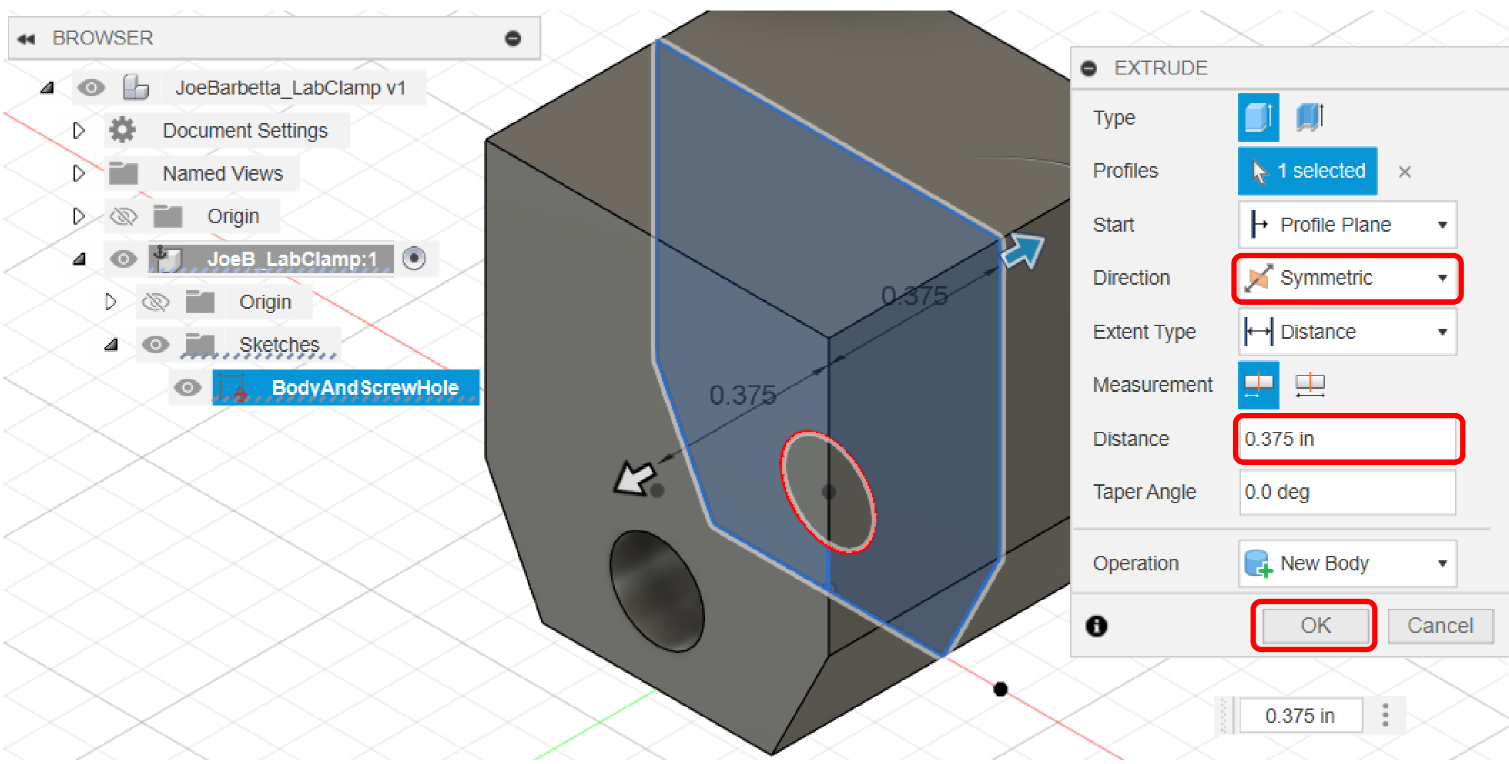
- admire your Sketch and click **Finish Sketch**



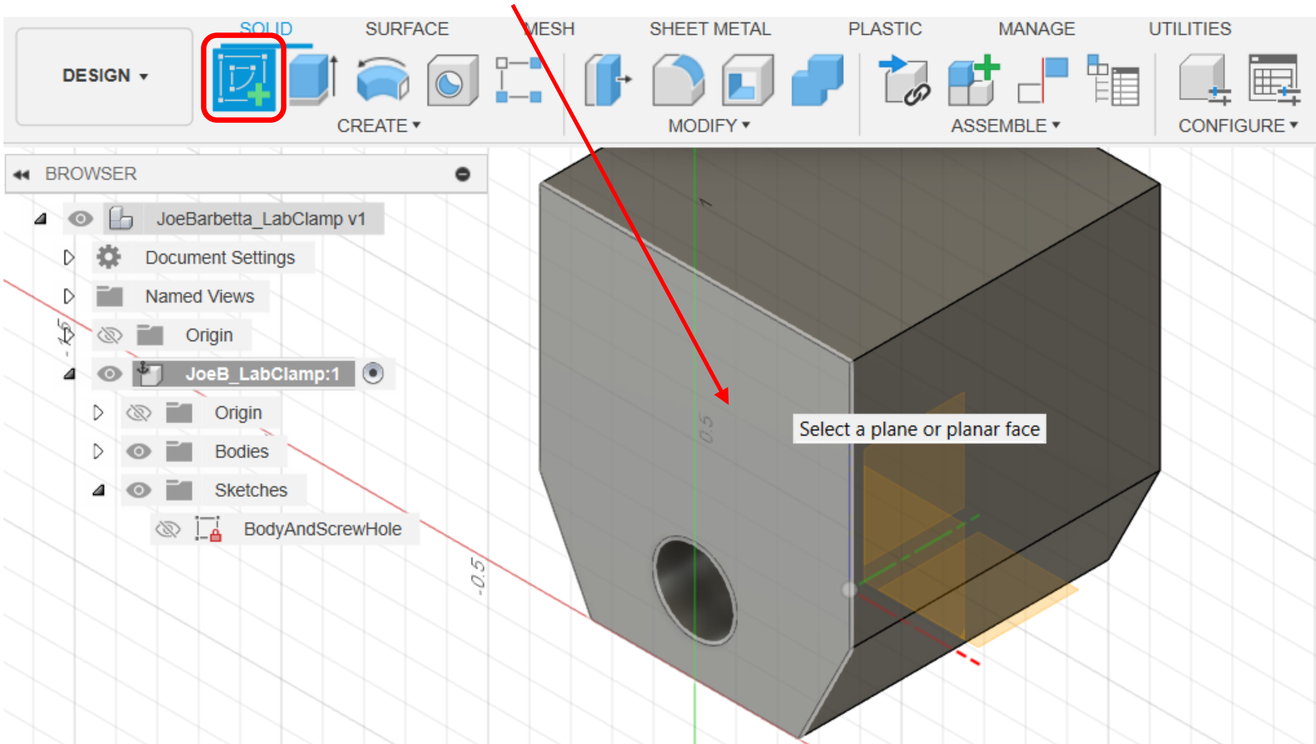
- click on the **Home** icon at the **View Cube** and adjust the view similar to that below
- select the **Extrude** tool and click on the **profile**



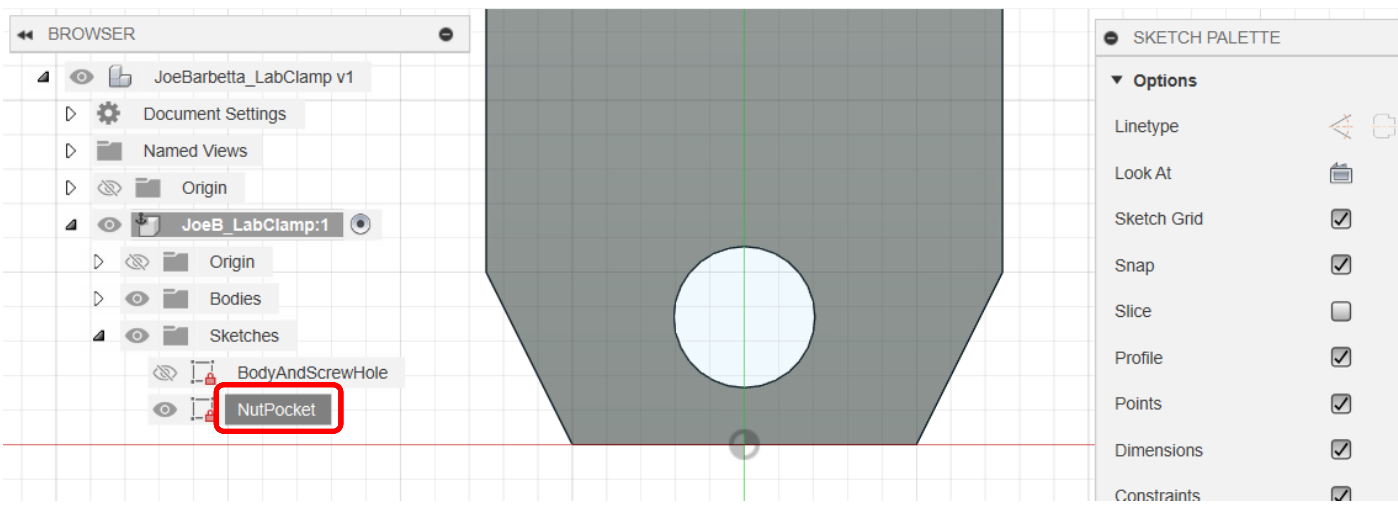
- change the **Direction** to **Symmetric**
- for **Distance** enter **0.375** and click **OK**



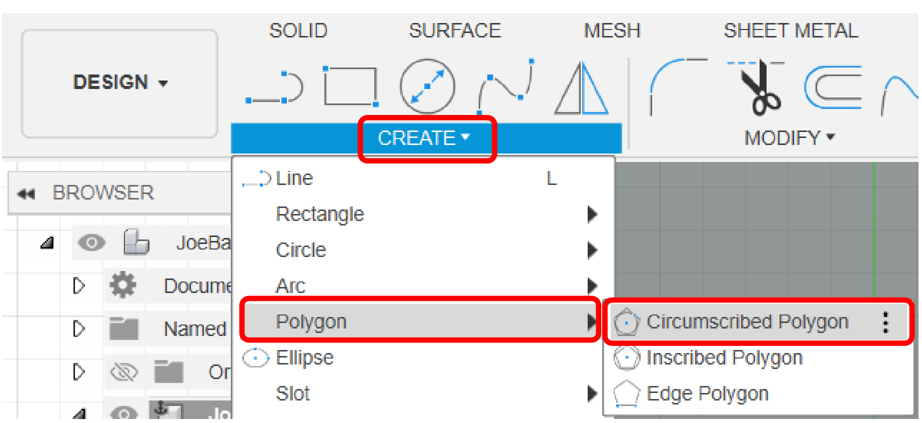
- select **Create Sketch** and click on the **surface** indicated



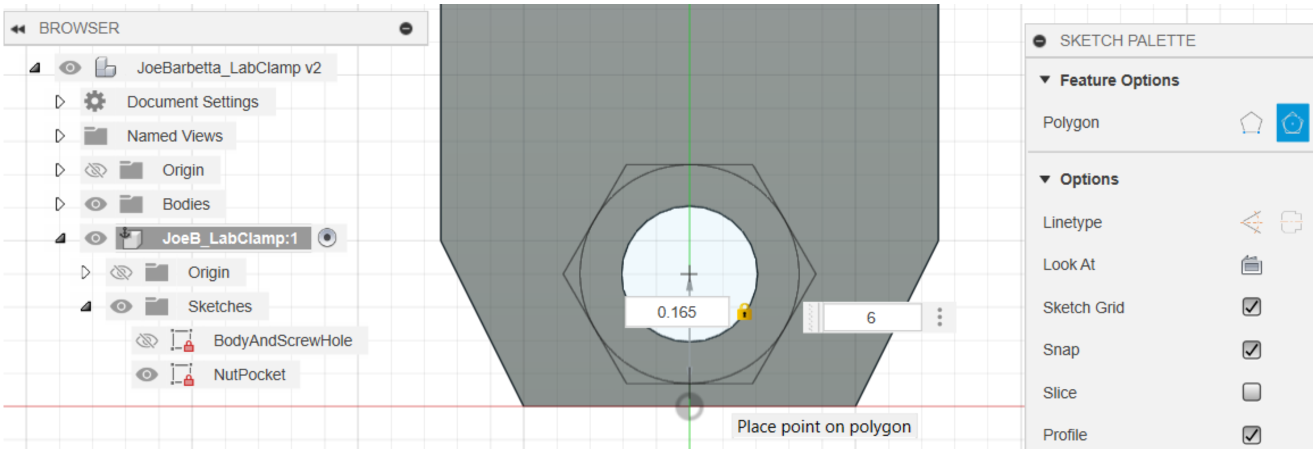
- rename the **Sketch** to **NutPocket** and zoom into the bottom as shown



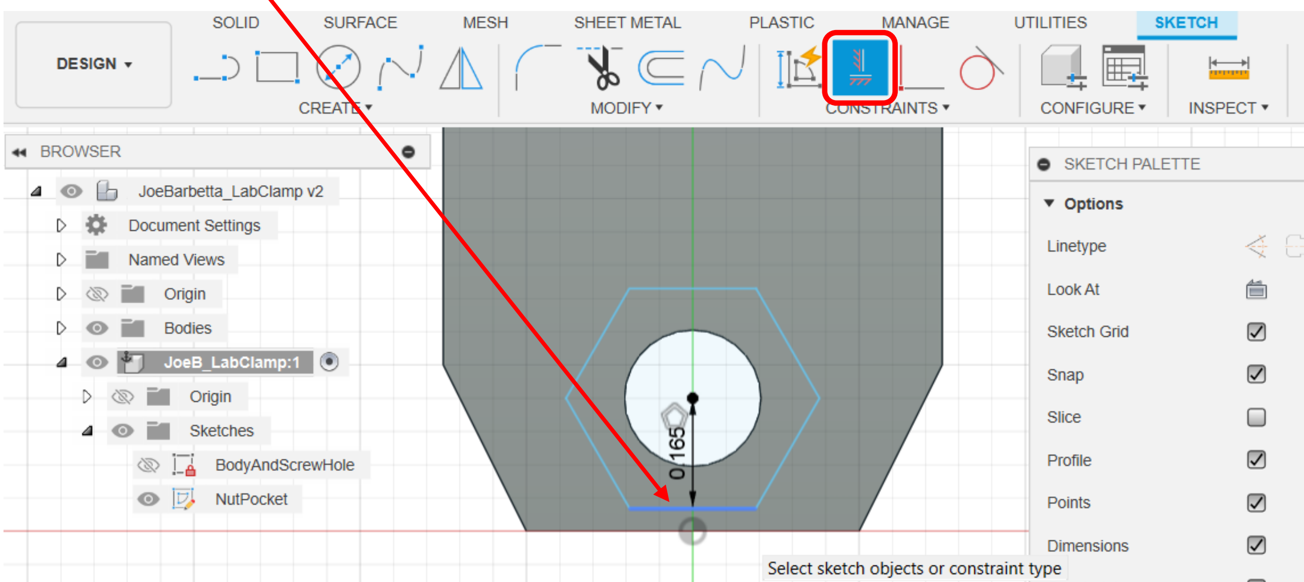
- from the **CREATE** menu, select **Polygon** and **Circumscribed Polygon**



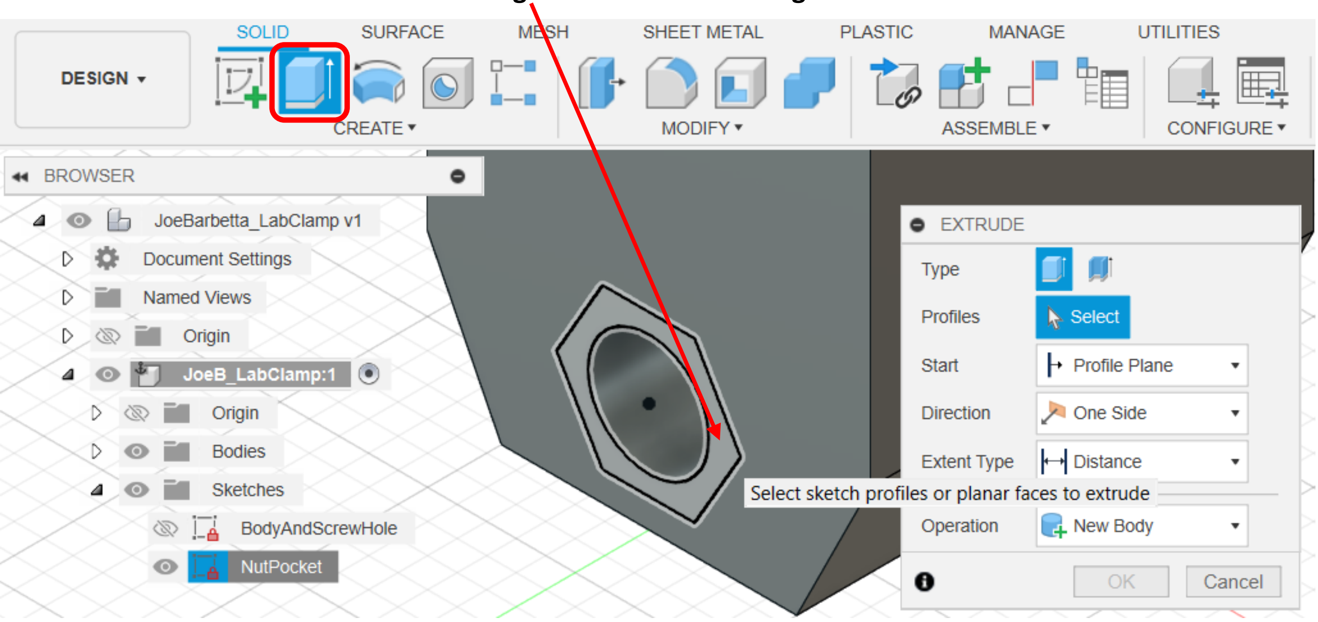
- click on the **center of the hole**, extend the hexagon outward, and enter **0.165**



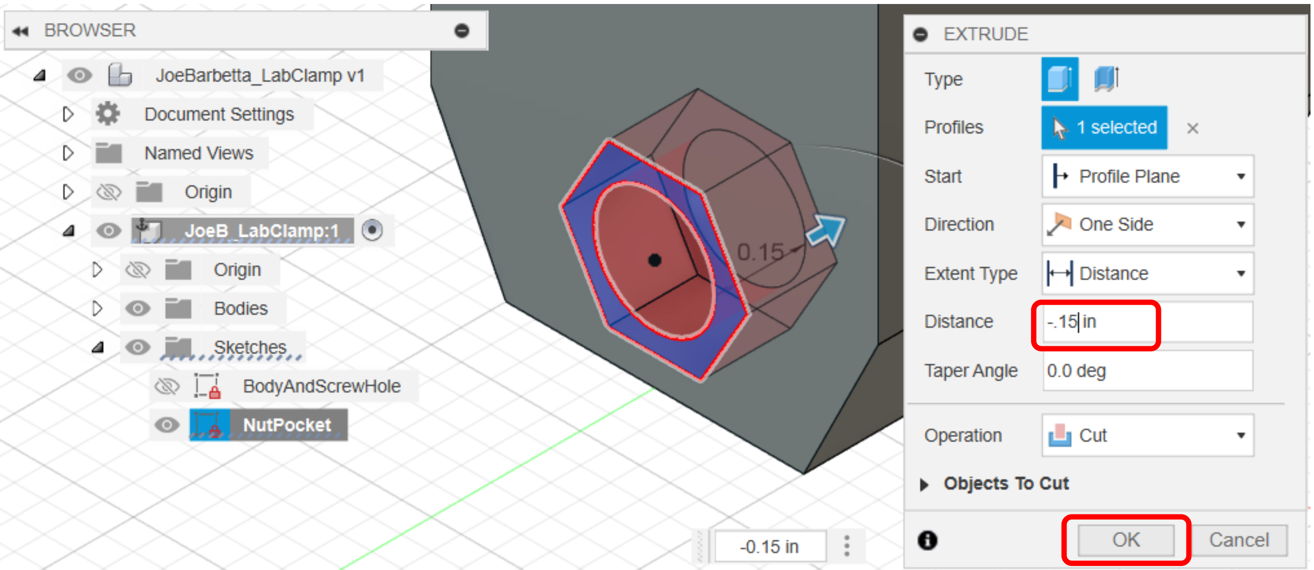
- select the **Horizontal / Vertical** Constraint. If it is not visible, find it in the **CONSTRAINTS** menu.
- click on the **bottom segment** of the hexagon and click **Finish Sketch**



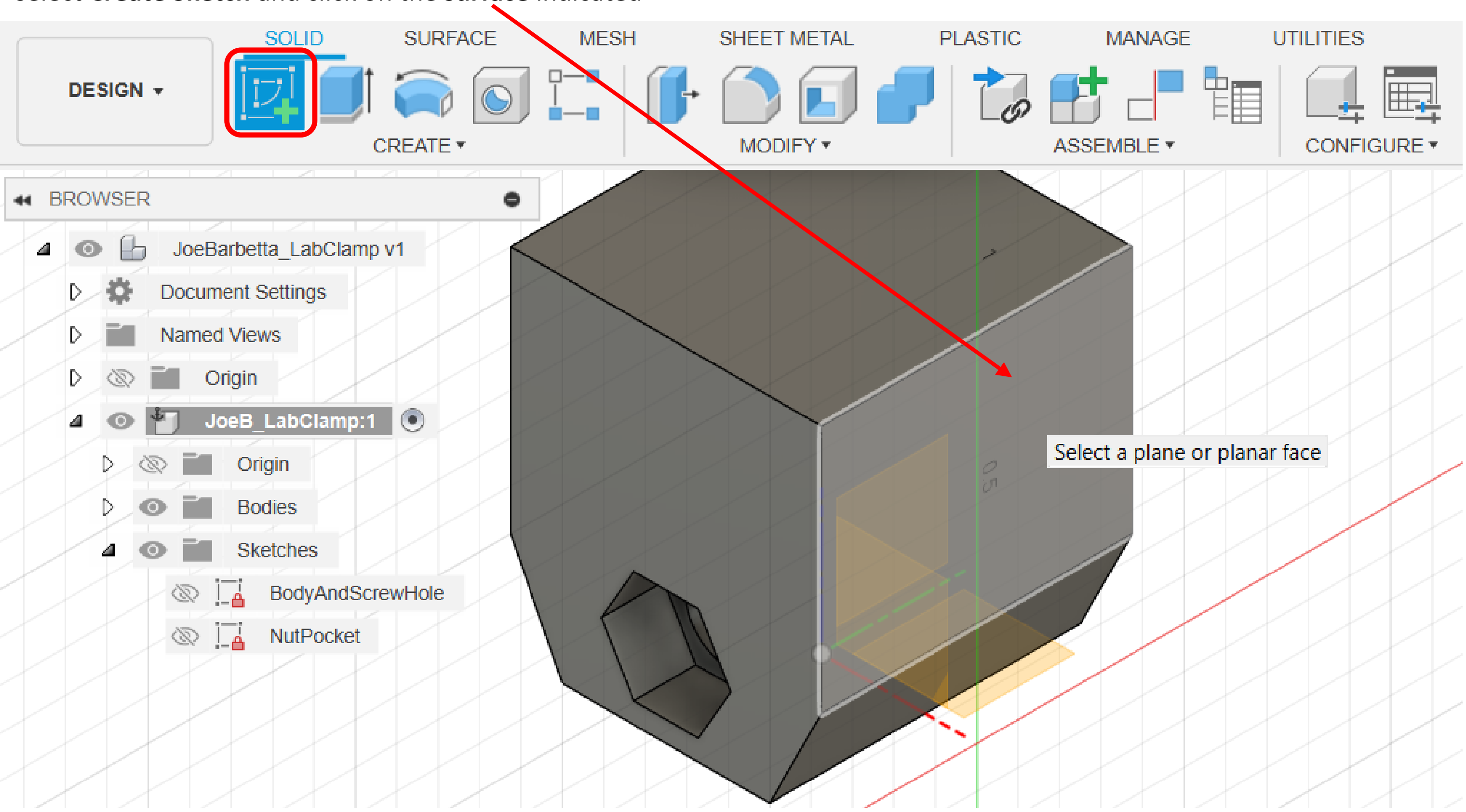
- select the **Extrude** tool and click on the **region between the hexagon and the hole**



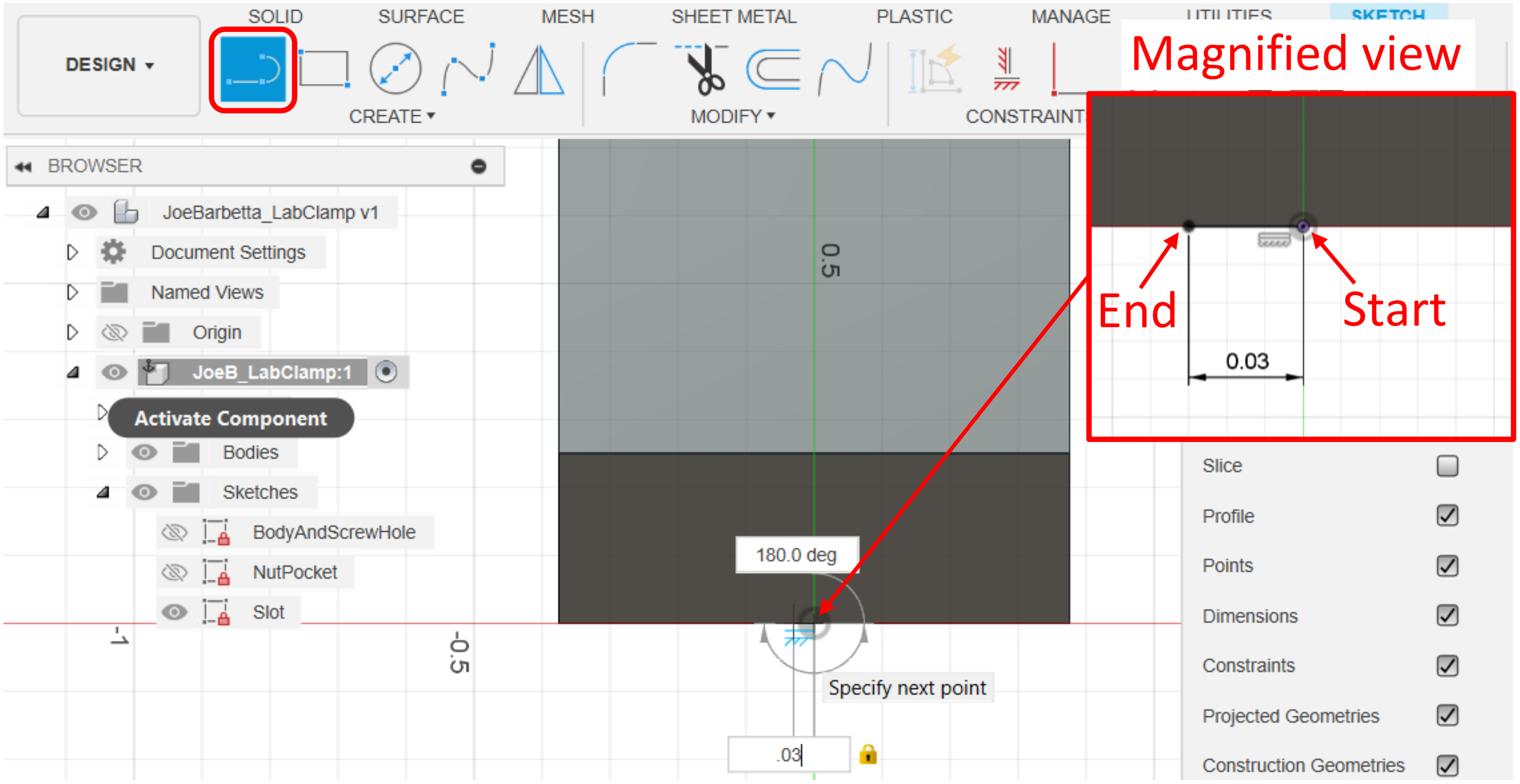
- for **Distance** enter **-0.15** (note the minus sign) and click **OK** and then **Finish Sketch**



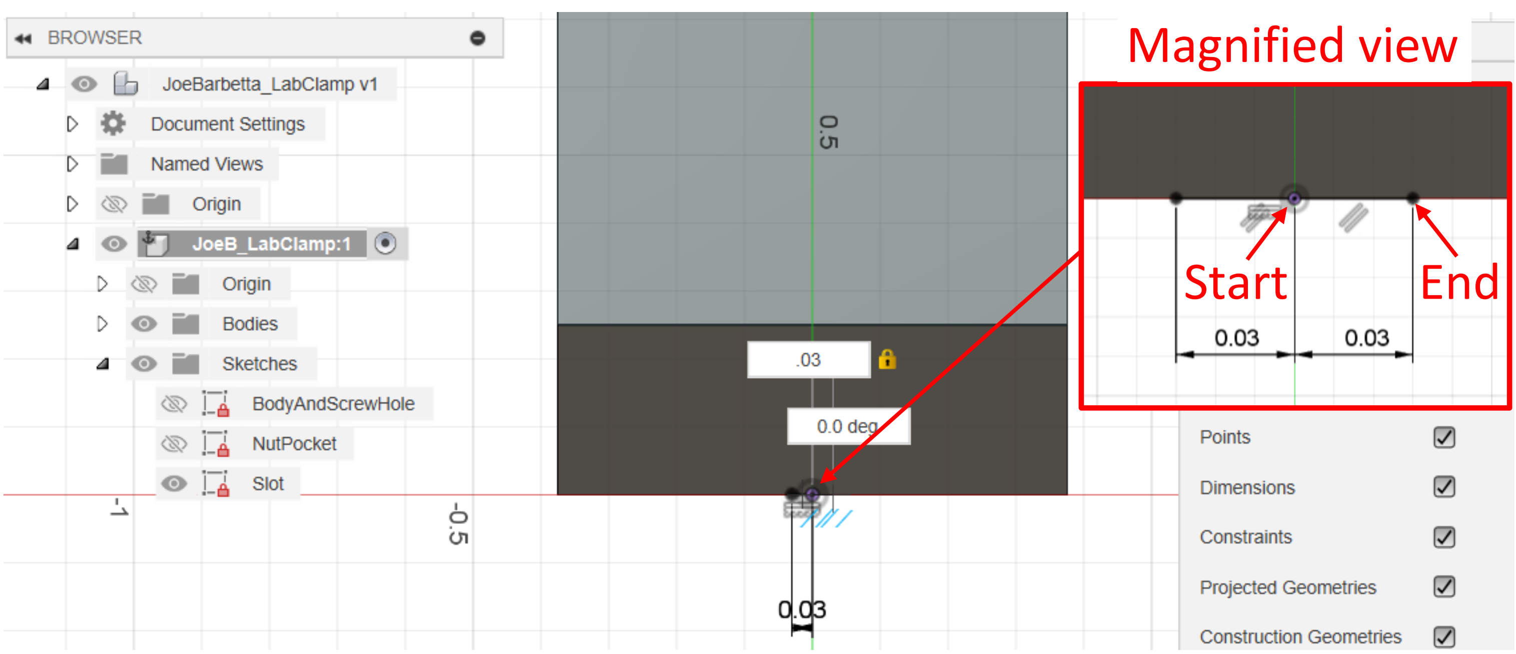
- select **Create Sketch** and click on the **surface** indicated



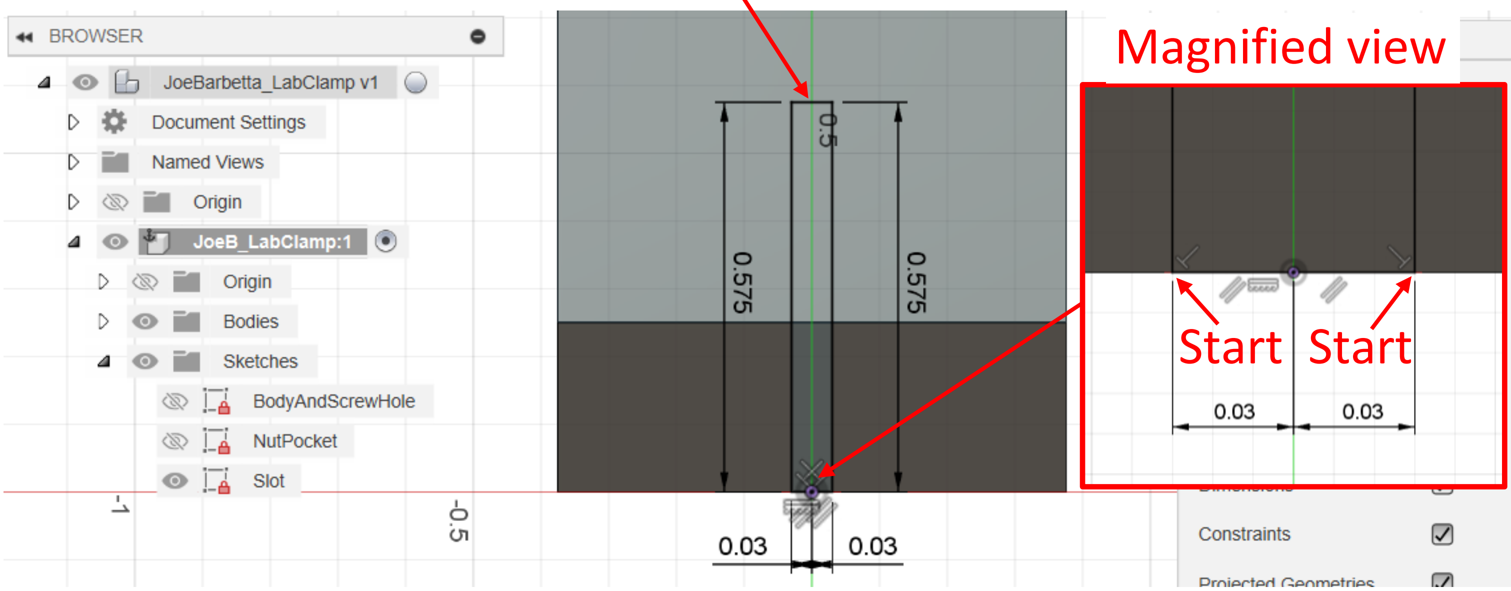
- select the **Line** tool and create starting at the **center of the bottom edge** and extended to the **left** by **0.03**



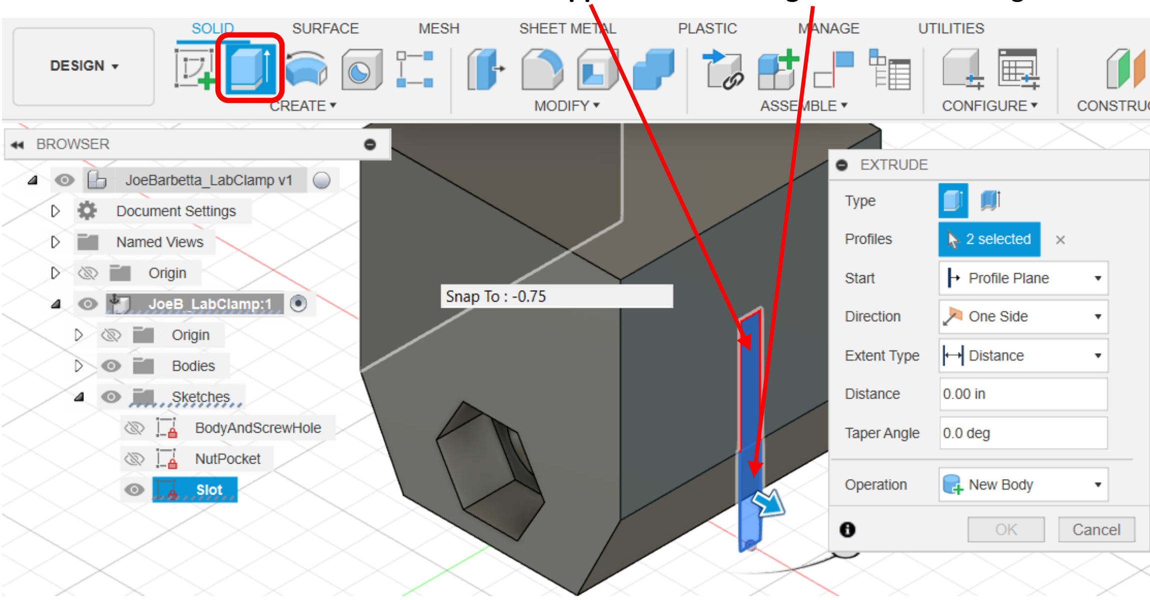
- select the **Line** tool and create starting at the **center of the bottom edge** and extended to the **right** by **0.03**



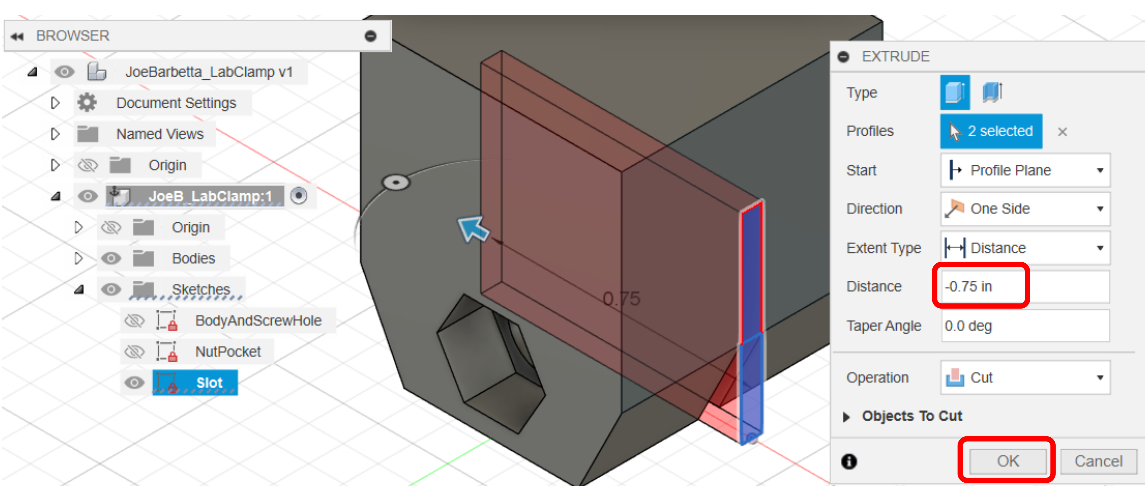
- using the Line tool again create **2 vertical lines** starting at the **endpoints** of the 0.03 lines and upward by **0.575**
- use the **Line** tool one last time to close the top with a **horizontal line**. Noe value needs to be entered.



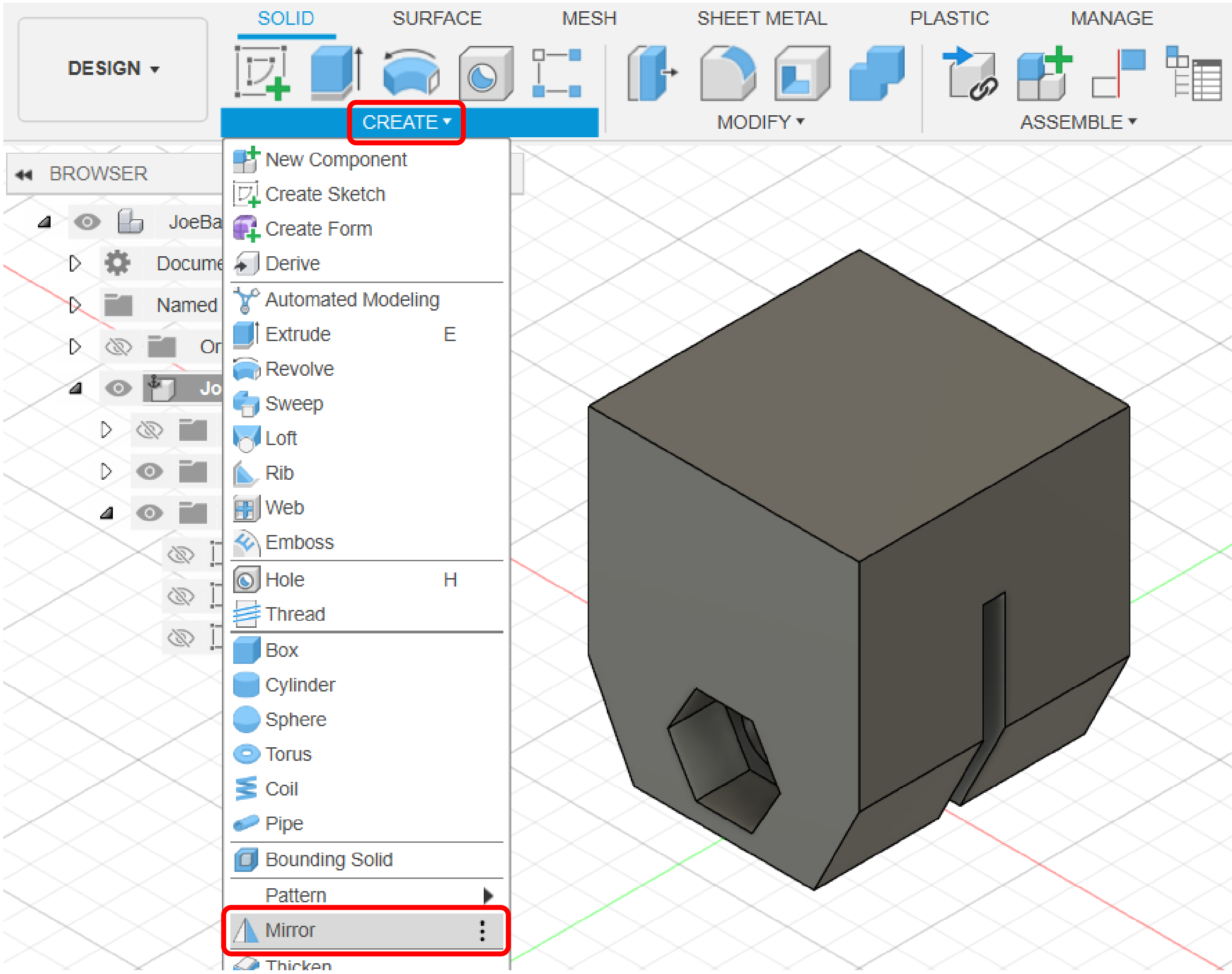
- select the **Extrude** tool and click on both the **upper and lower regions** of the rectangle to turn the entire blue



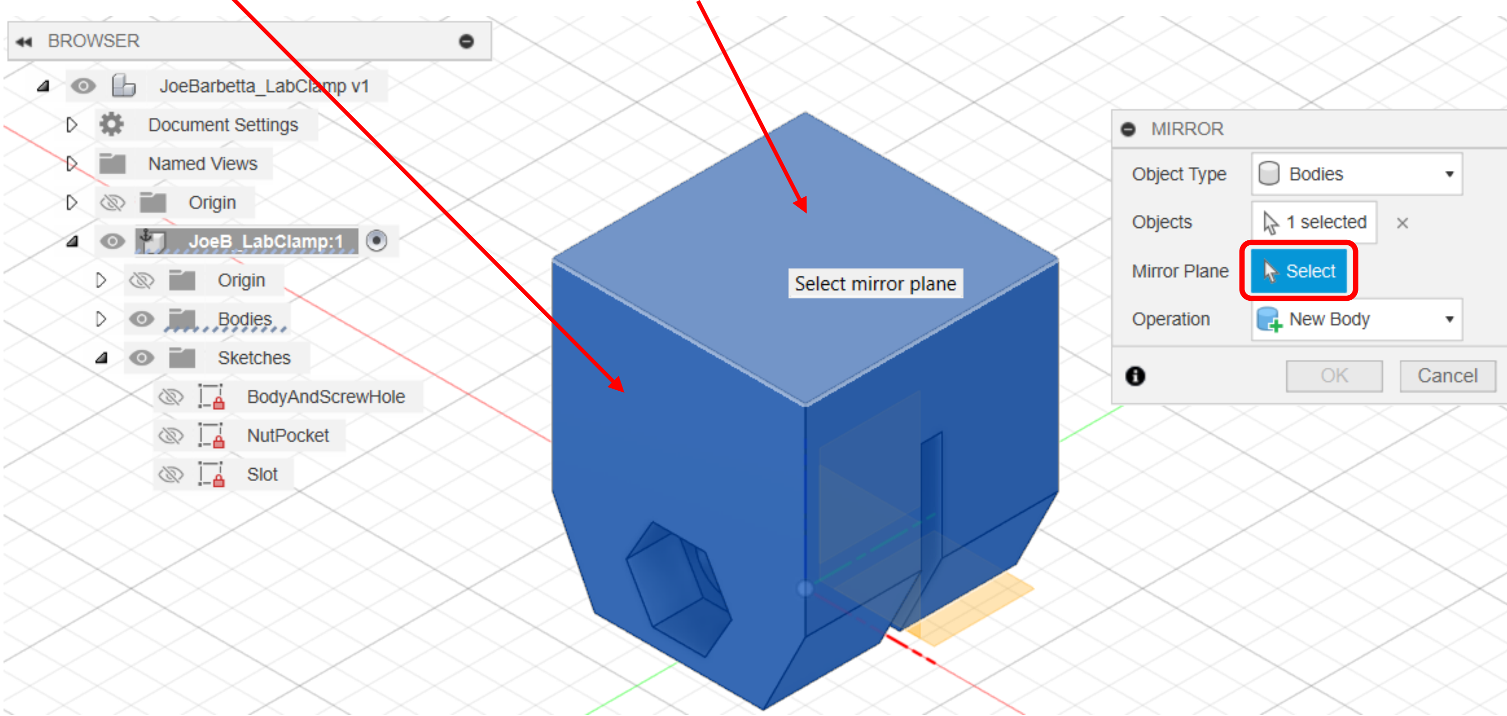
- for **Distance** enter **-0.75** (note the minus sign) and click **OK**



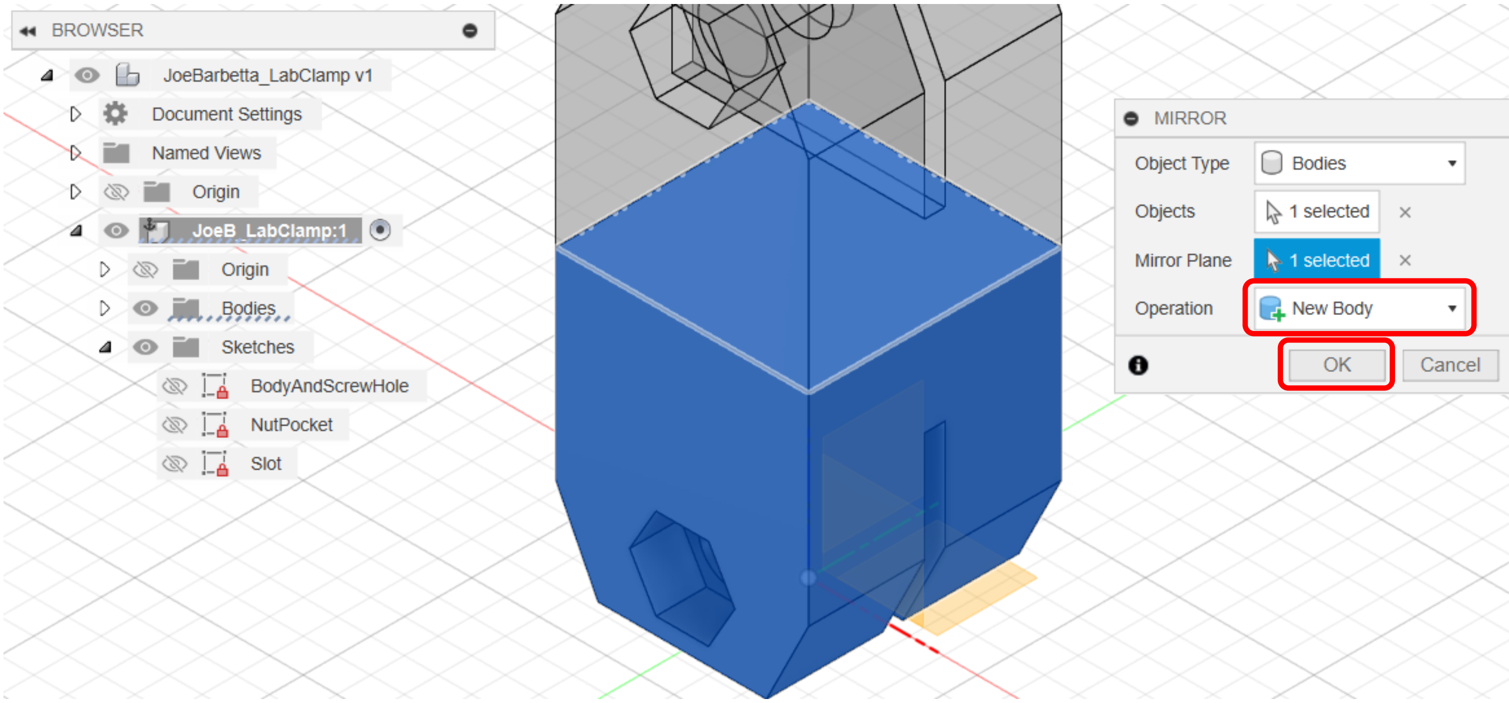
- from thr **CREATE** menu select **Mirror**



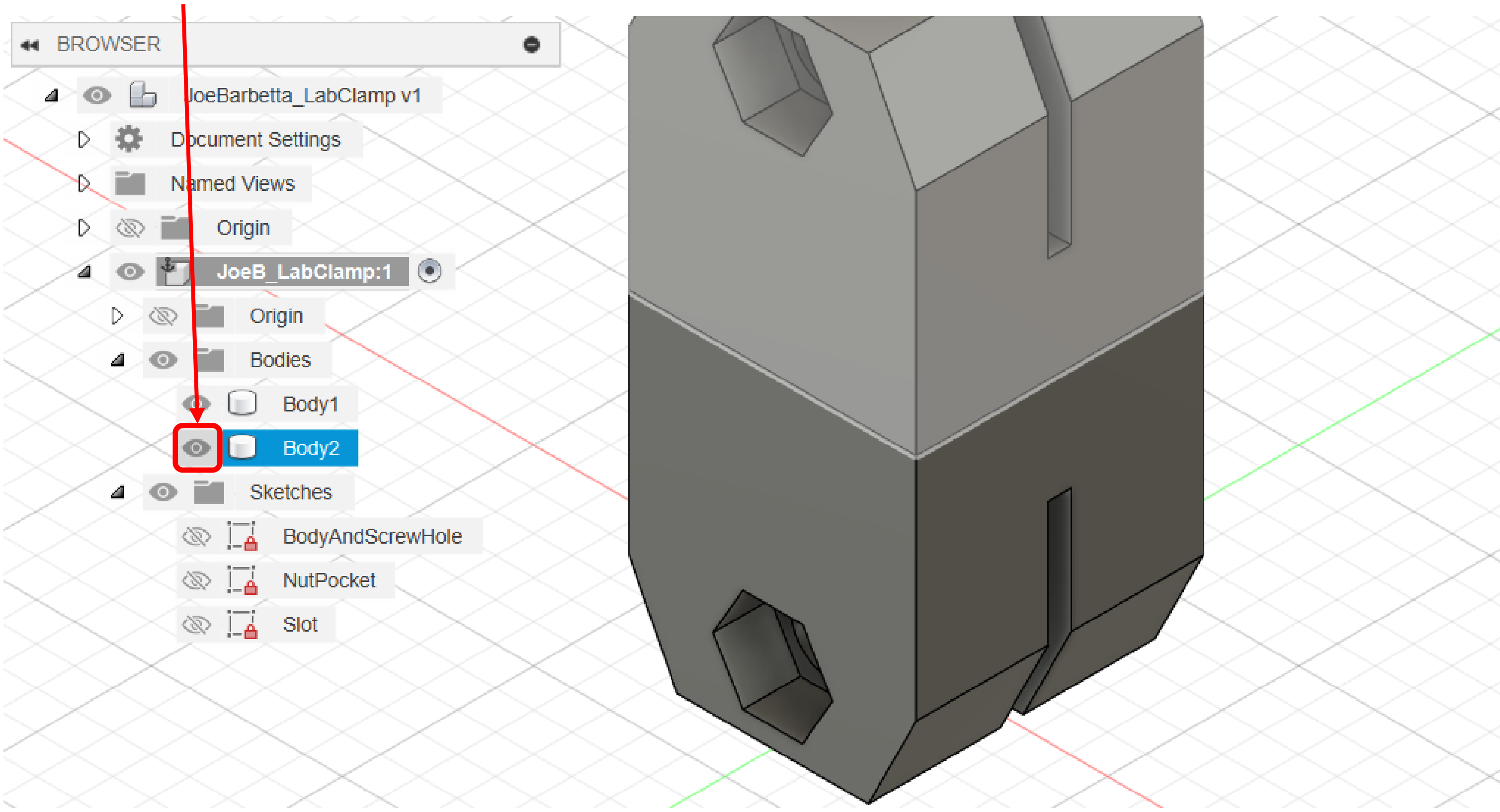
- click on the **Body**
- click on **Select** for **Mirror Plane** and then click on the **top surface**



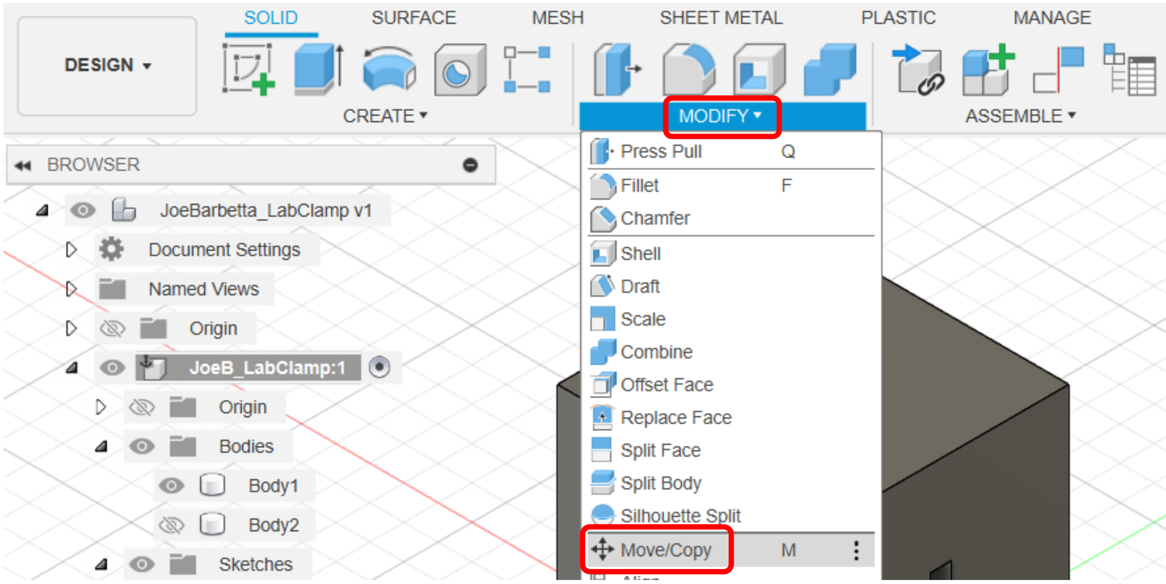
- change **Operation** to **New Body**
- click **OK**



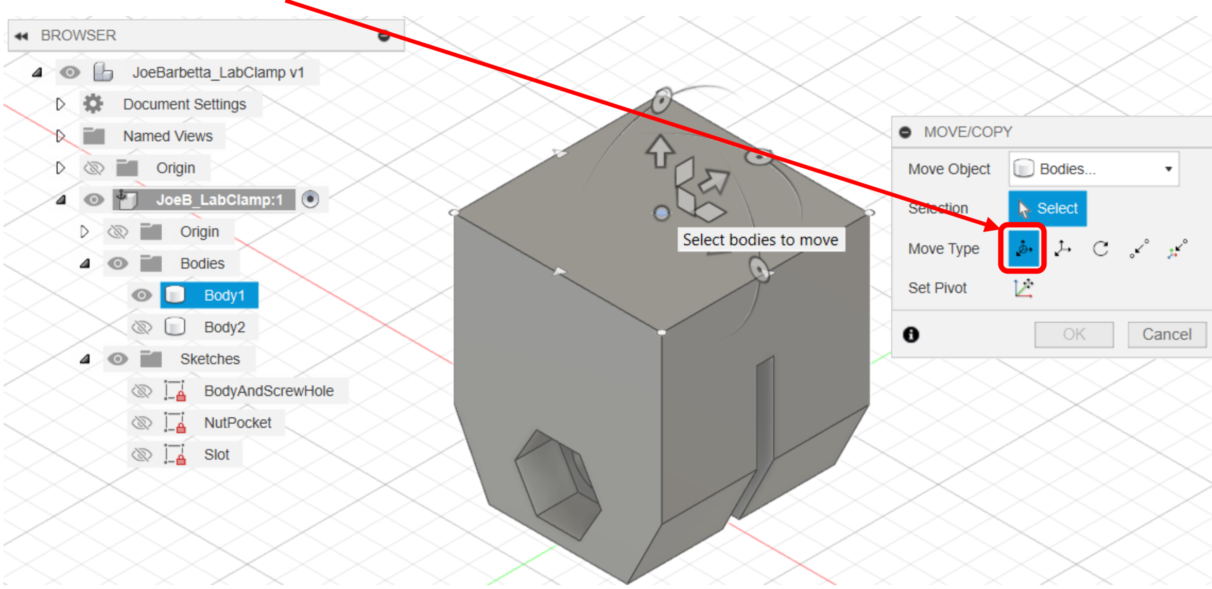
- click on the **eye** icon for **Body2** to hide it



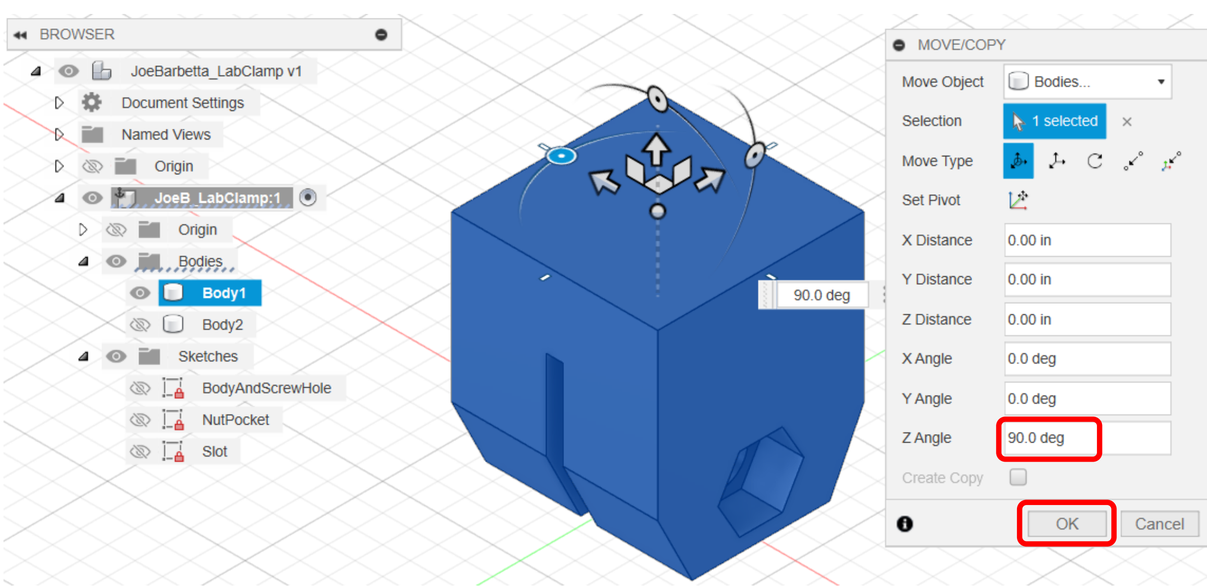
- from the **MODIFY** menu, select **Move/Copy**



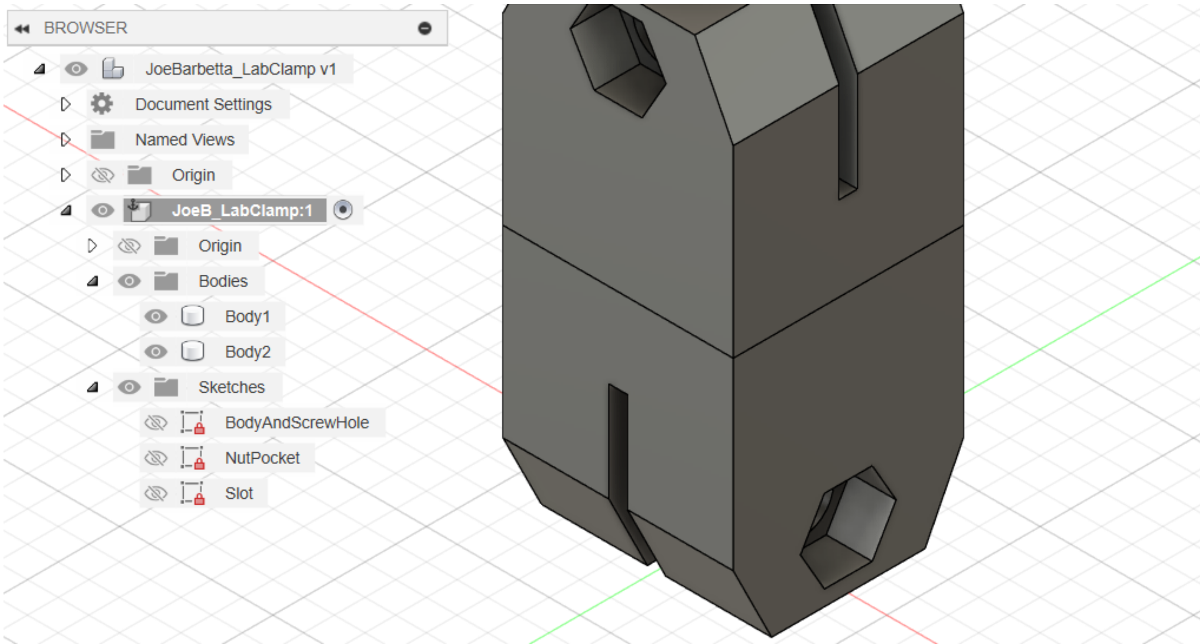
- ensure the **Free Move** icon is selected for **Move Type** is and click on the **center of the top surface**



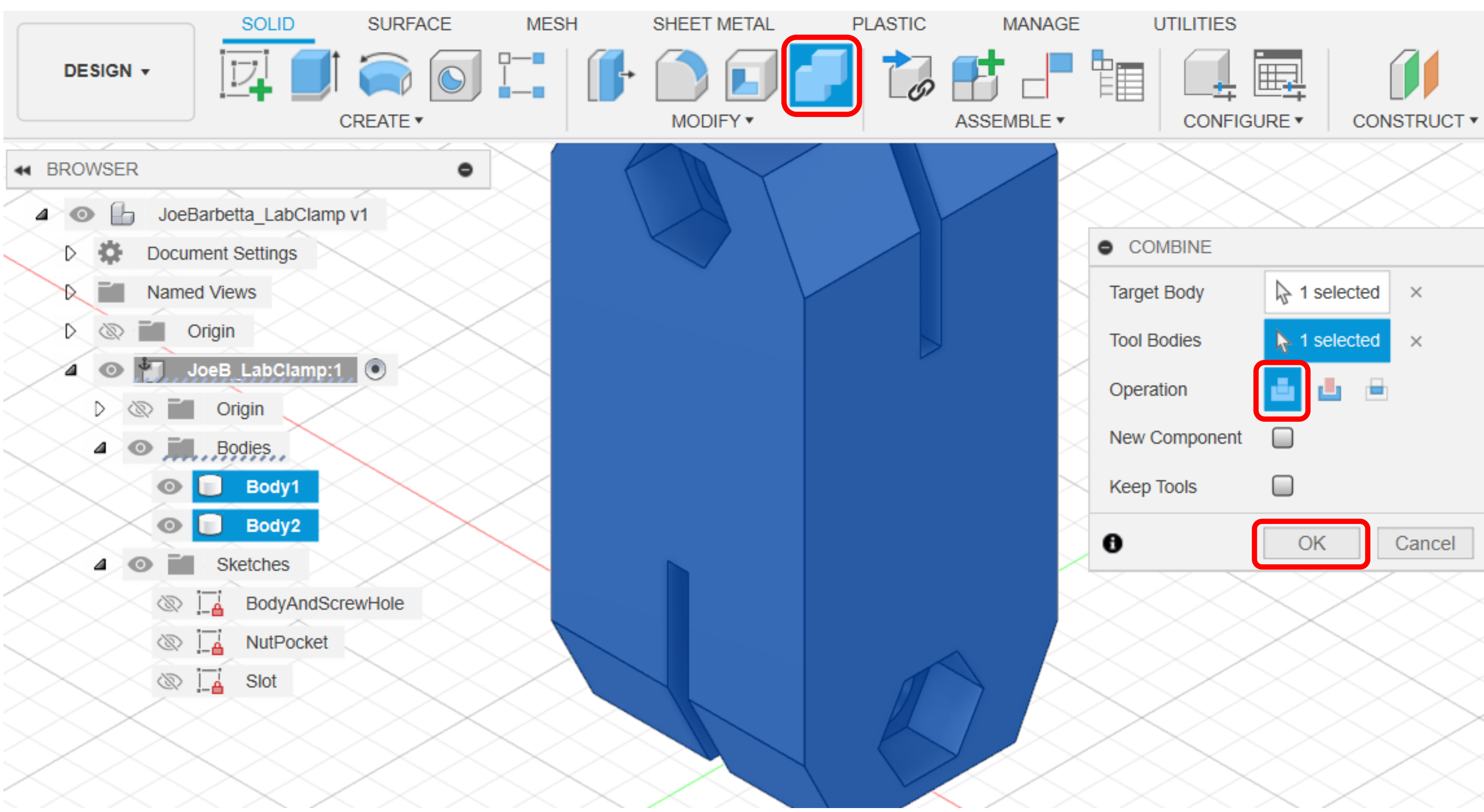
- change **Z Angle** to **90** deg and click **OK**



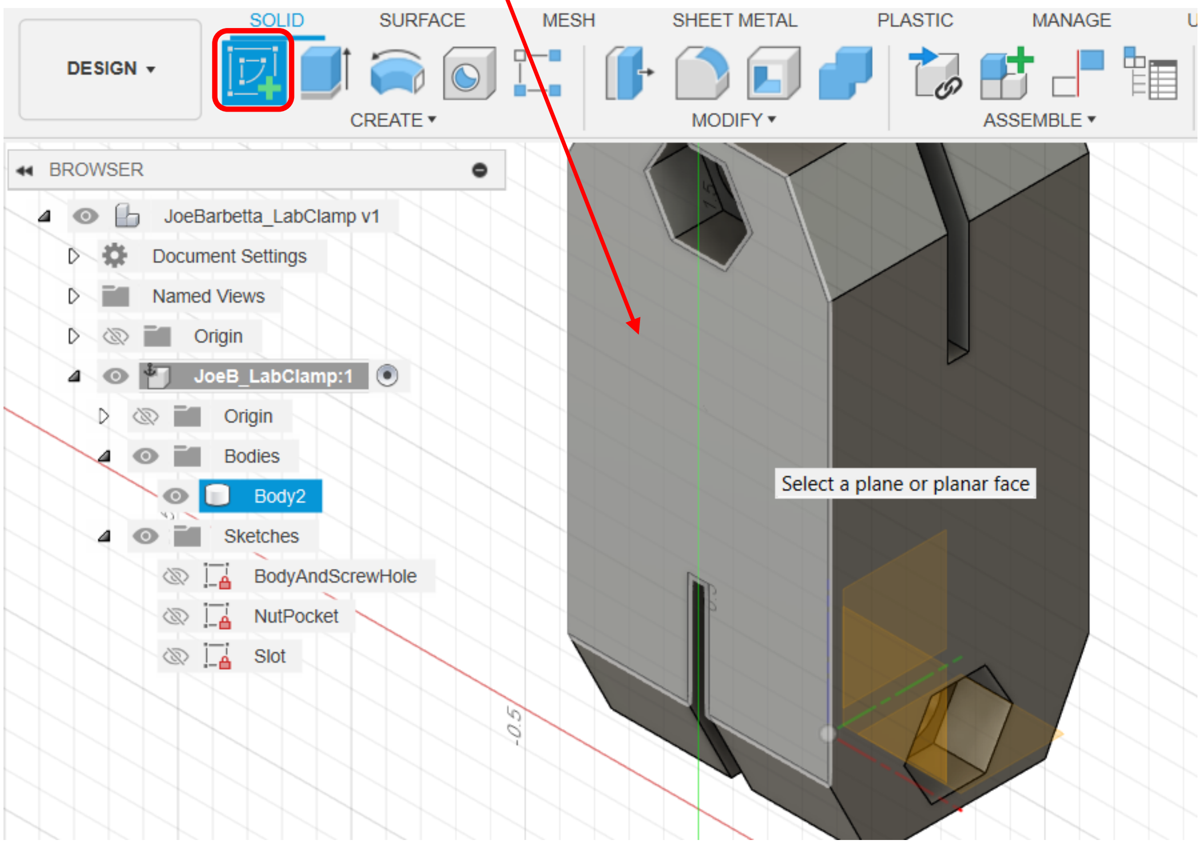
This should be the result of the rotation.



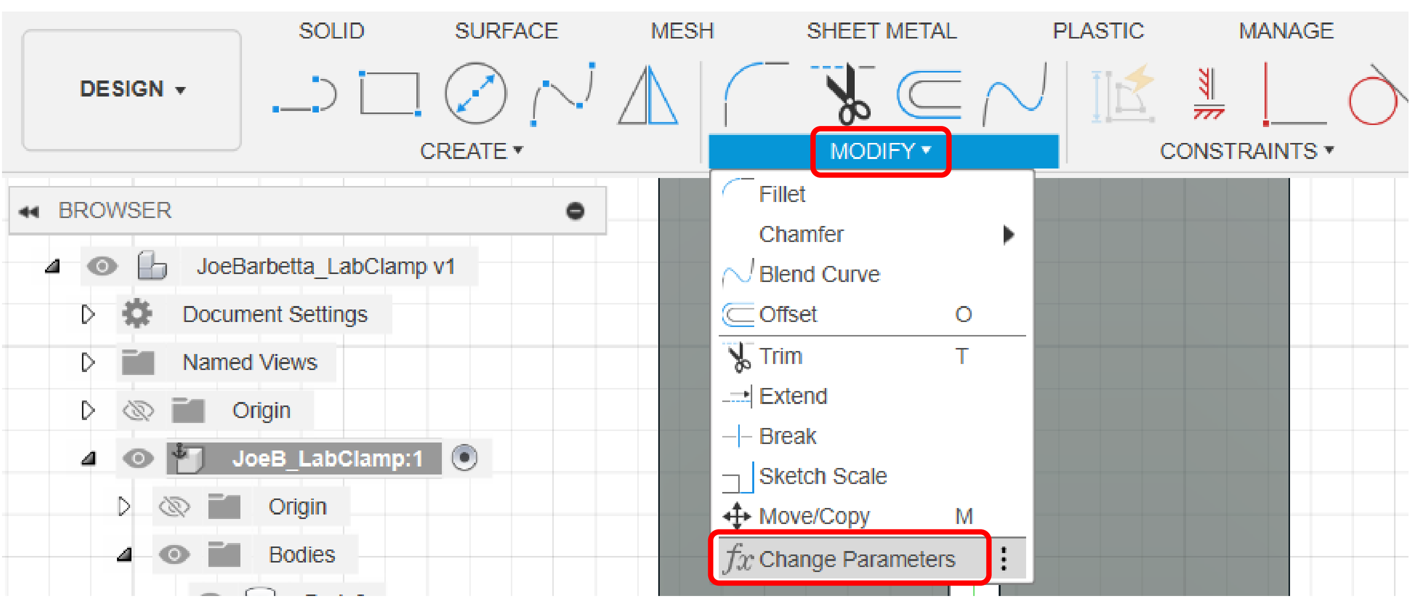
- select the **Combine** tool. If it is not visible, find it in the **MODIFY** menu.
- ensure that the **Join** icon is selected for **Operation** and that **Keep Tools** is Not checked
- click on the **upper** and **lower** bodies and click **OK**



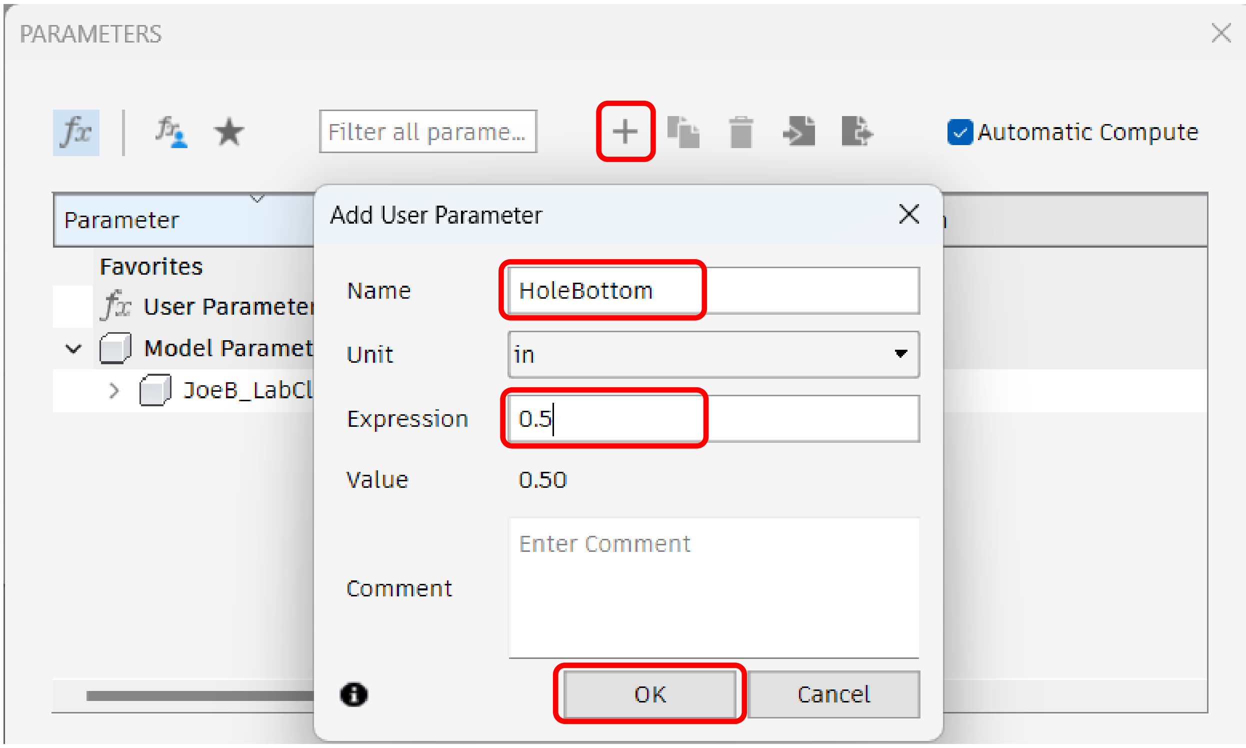
- select **Create Sketch** and click on the **face** as indicated
- **rename the Sketch** to **HoleBottom**



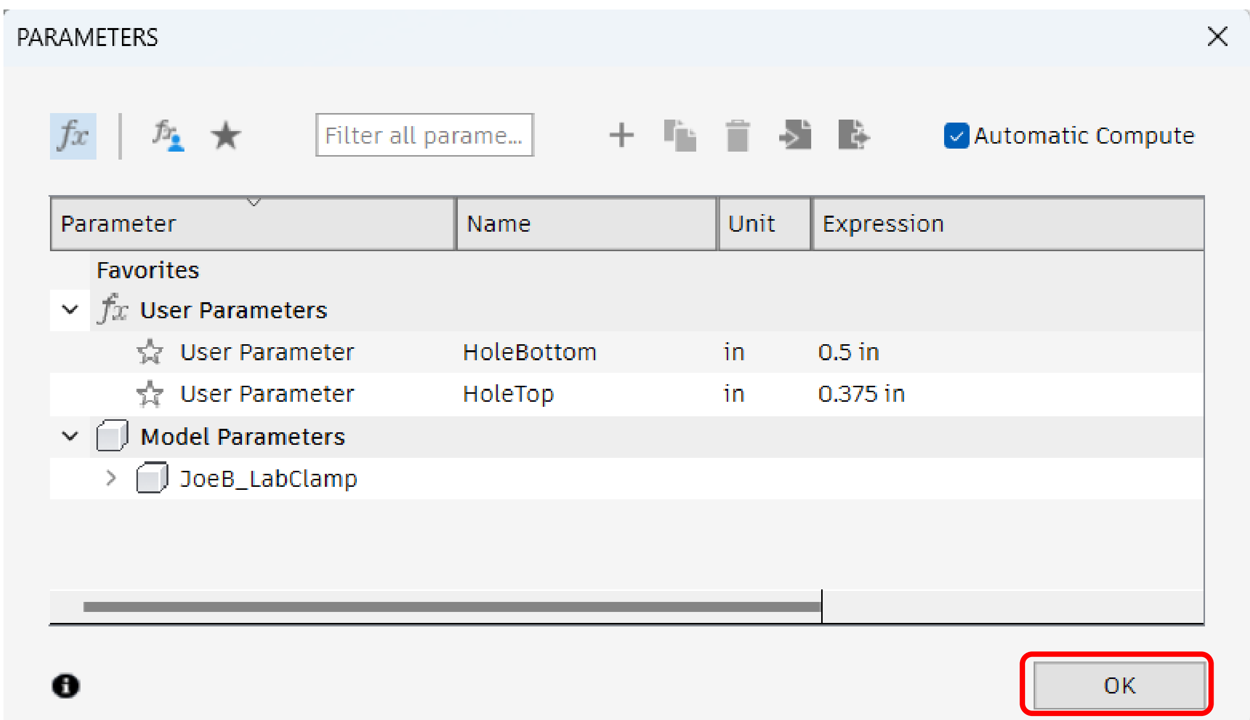
- from the **MODIFY** menu, select **Change Parameters**
- if a **Parametric Text** window appears, click its **OK** button



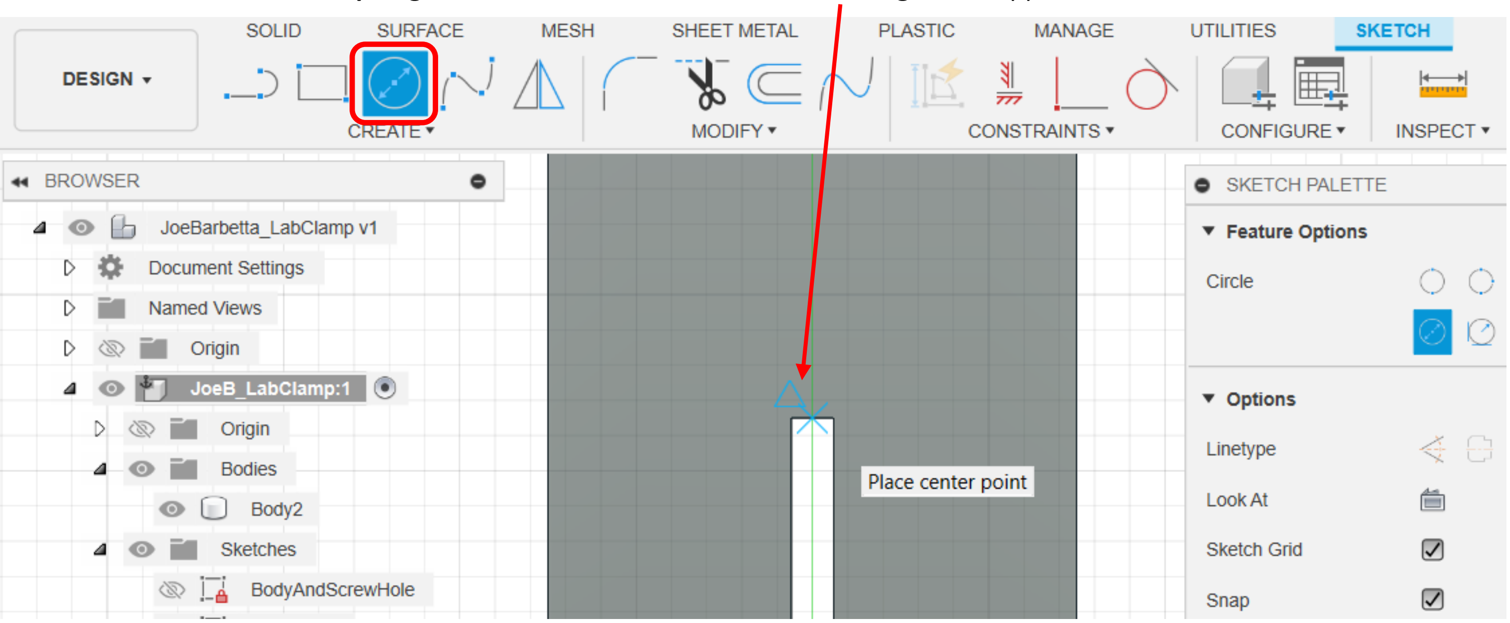
- click on the **+** icon.   If a **Parametric Text** window appears, click its **OK** button.
- for **Name** enter **HoleBottom** and for **Expression** enter **0.5** and click **OK**



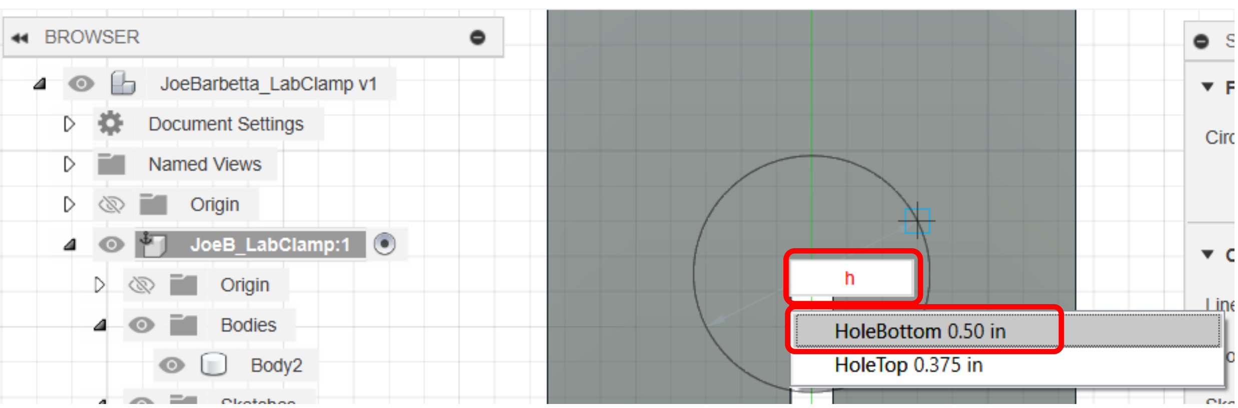
- click on the **+** icon again
- for **Name** enter **HoleTop** and for **Expression** enter **0.375** and click **OK**
- click on **OK** at the bottom right of the PARAMETERS window



- select the **Center Diameter Circle** tool. If it it not visible, find it in the CREATE menu.
- move the mouse over the **top edge of the slot** and click when a **blue triangle** icon appears

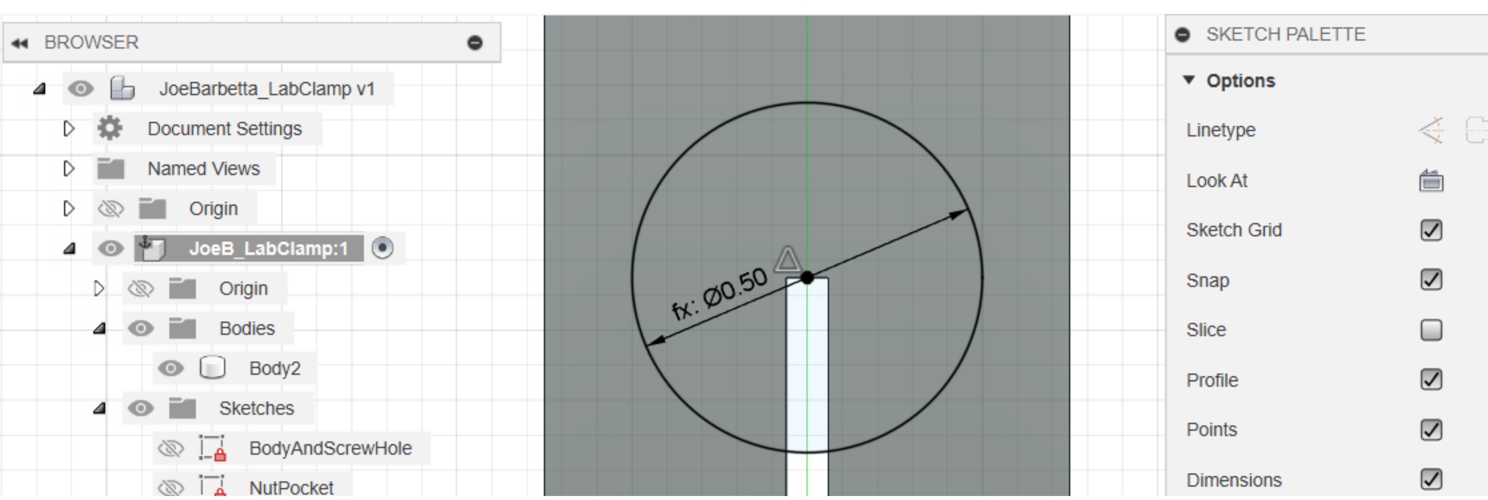


- extend the circle outward, type **h**, and press the **Enter key** to select **HoleBottom**

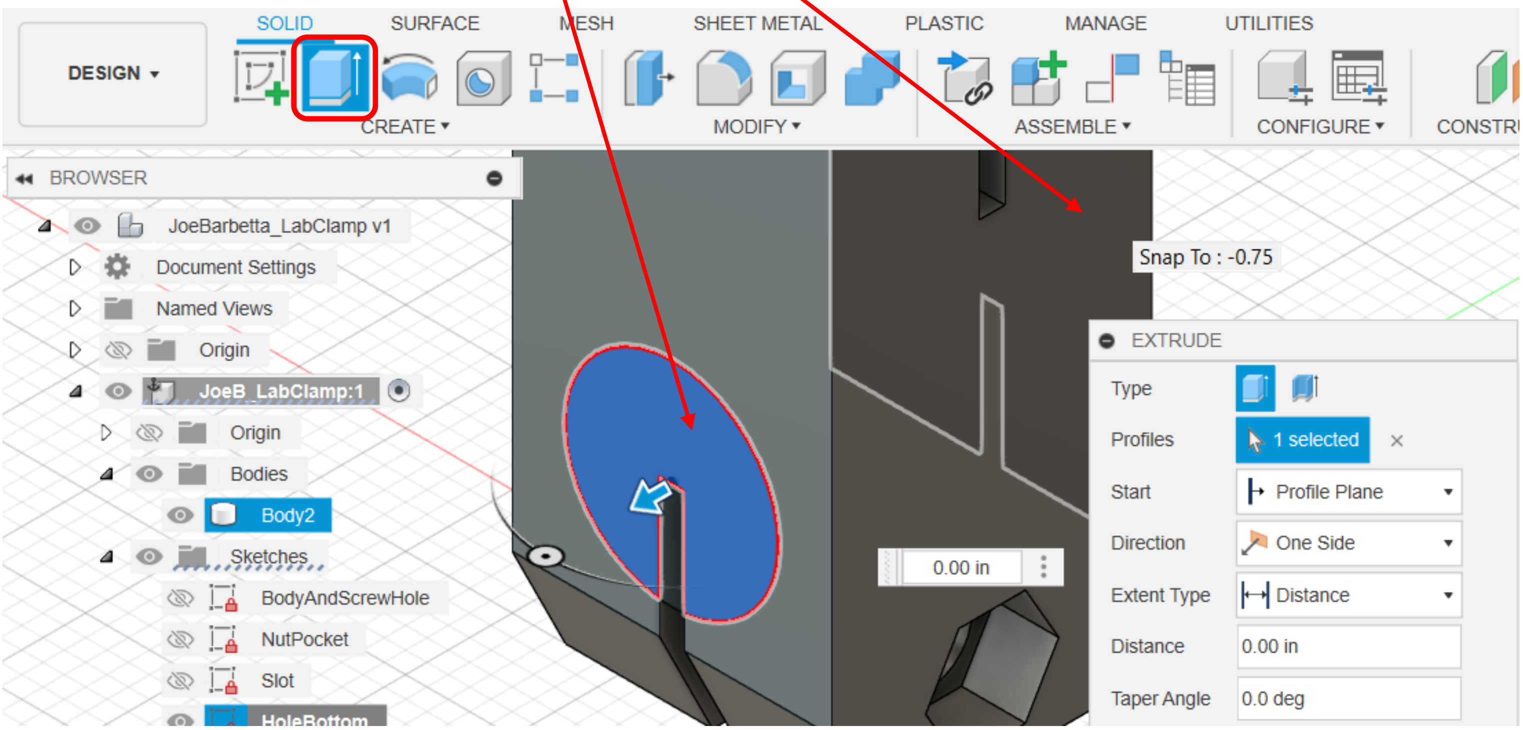


The circle should look like that below. It is ok if the Dimension line looks different.

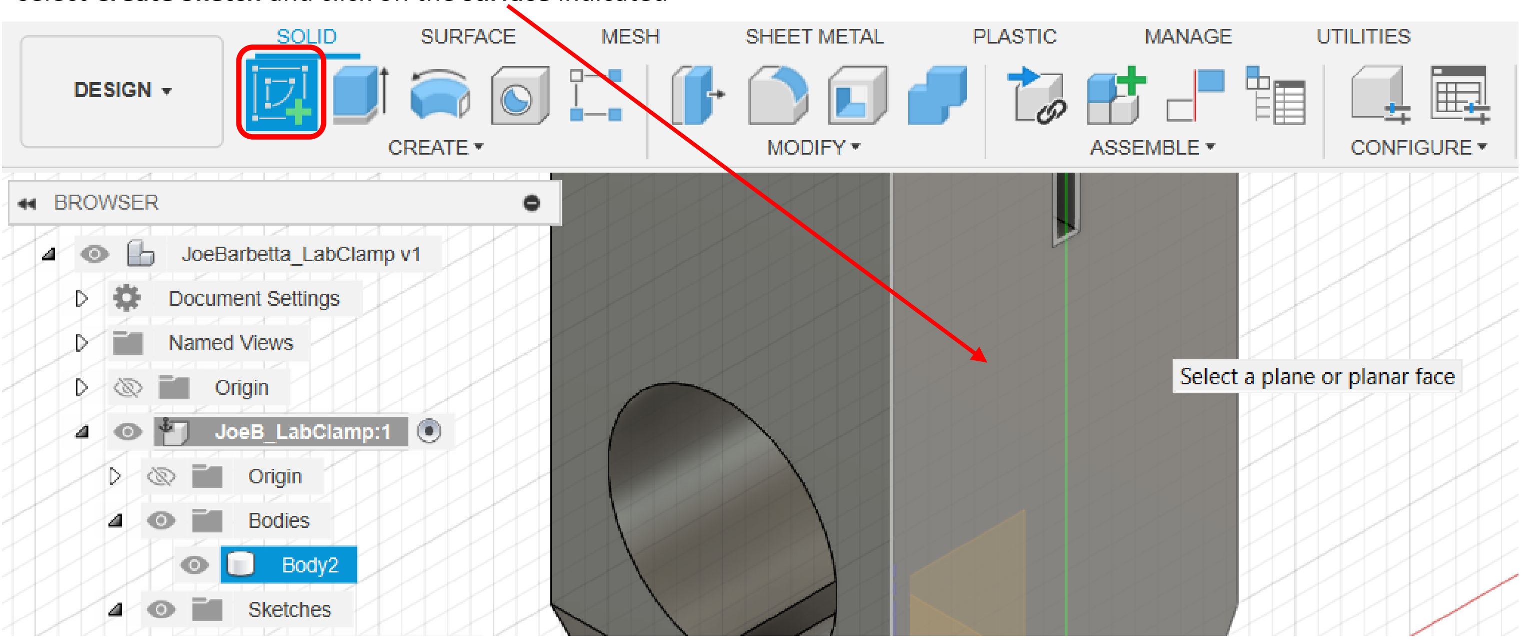
- click **Finish Sketch**



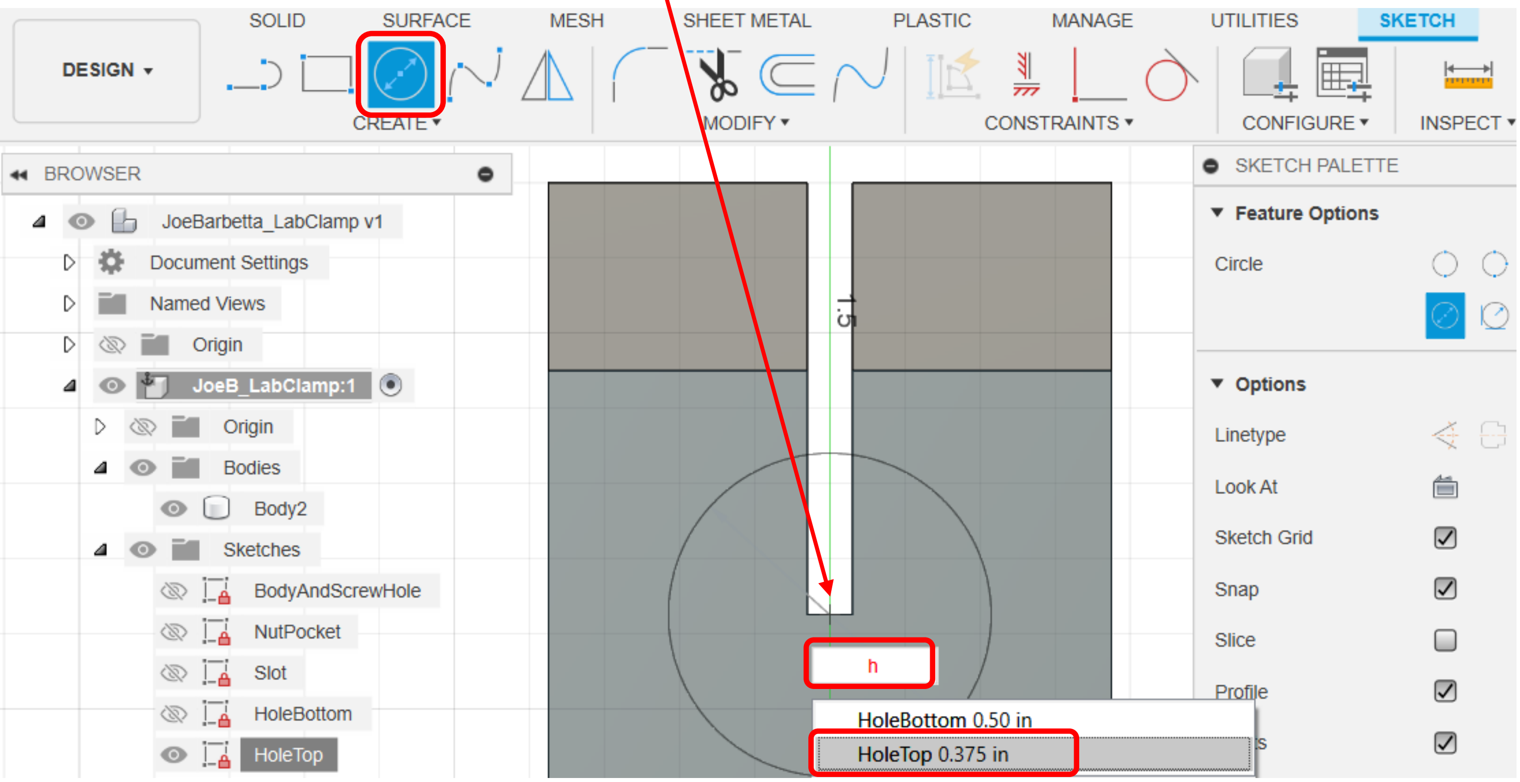
- zoom into the side where the circle is
- select the **Extrude** tool and click on the **interior** of the circle to highlight it blue
- move the mouse over the area shown to select the **opposite face** of the body (Snap to: -0.75 should show) and click
- click **OK**



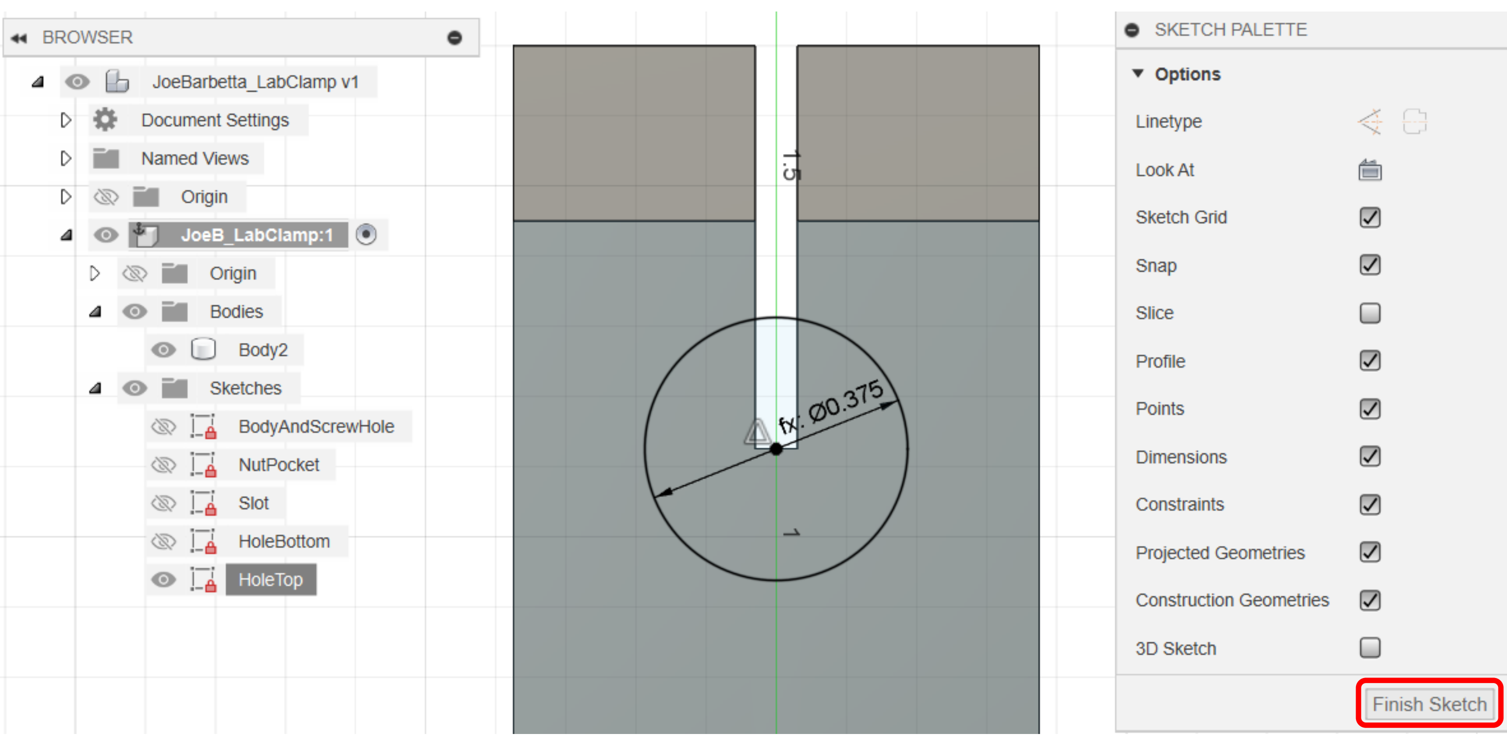
- select **Create Sketch** and click on the **surface** indicated



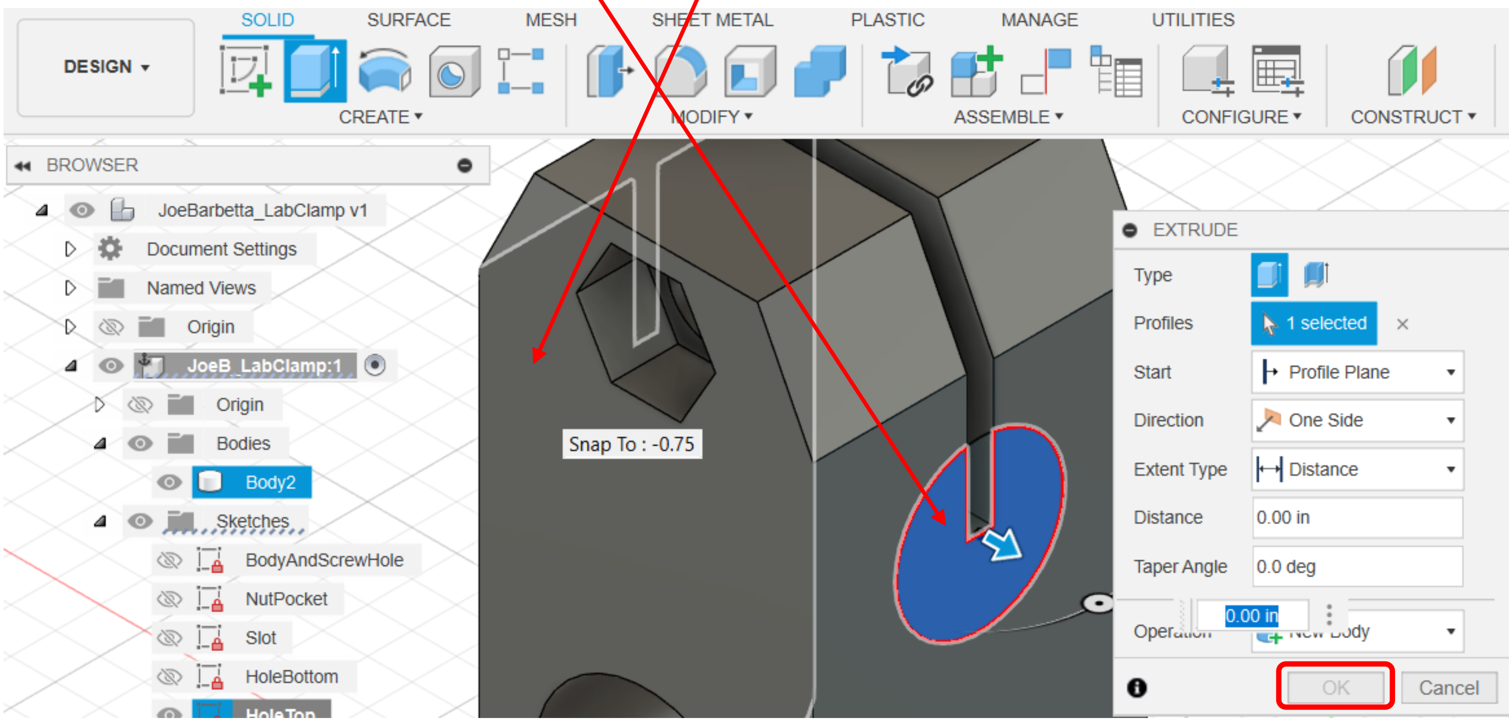
- zoom into the top of the body as shown
- **rename the Sketch** to **HoleTop**
- select the **Center Diameter Circle** and click on the **center** of the slot edge
- extend the circle outward, type **h**, use the **Down Arrow key** to select **HoleTop**, and press the **Enter key**



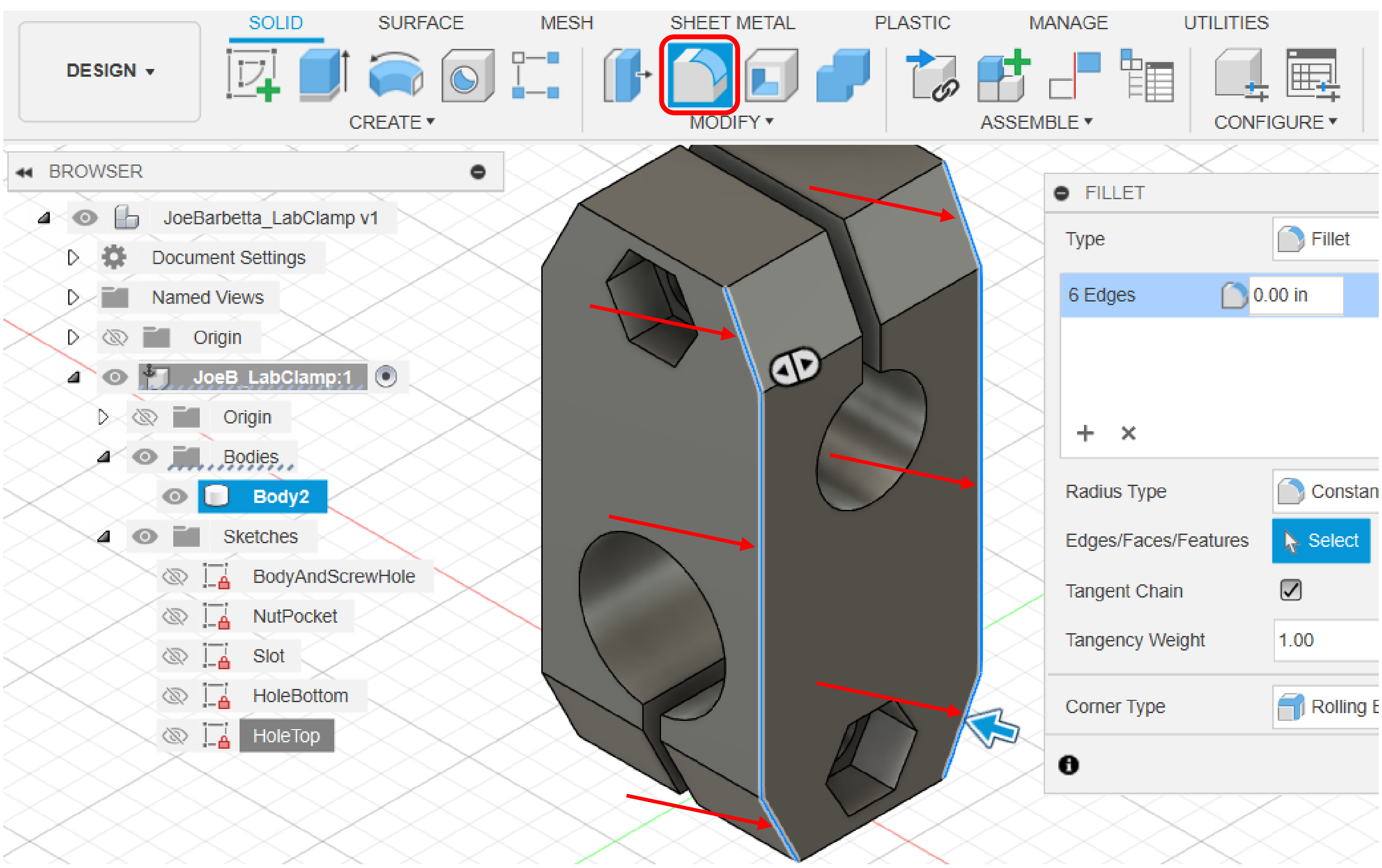
- click **Finish Sketch**



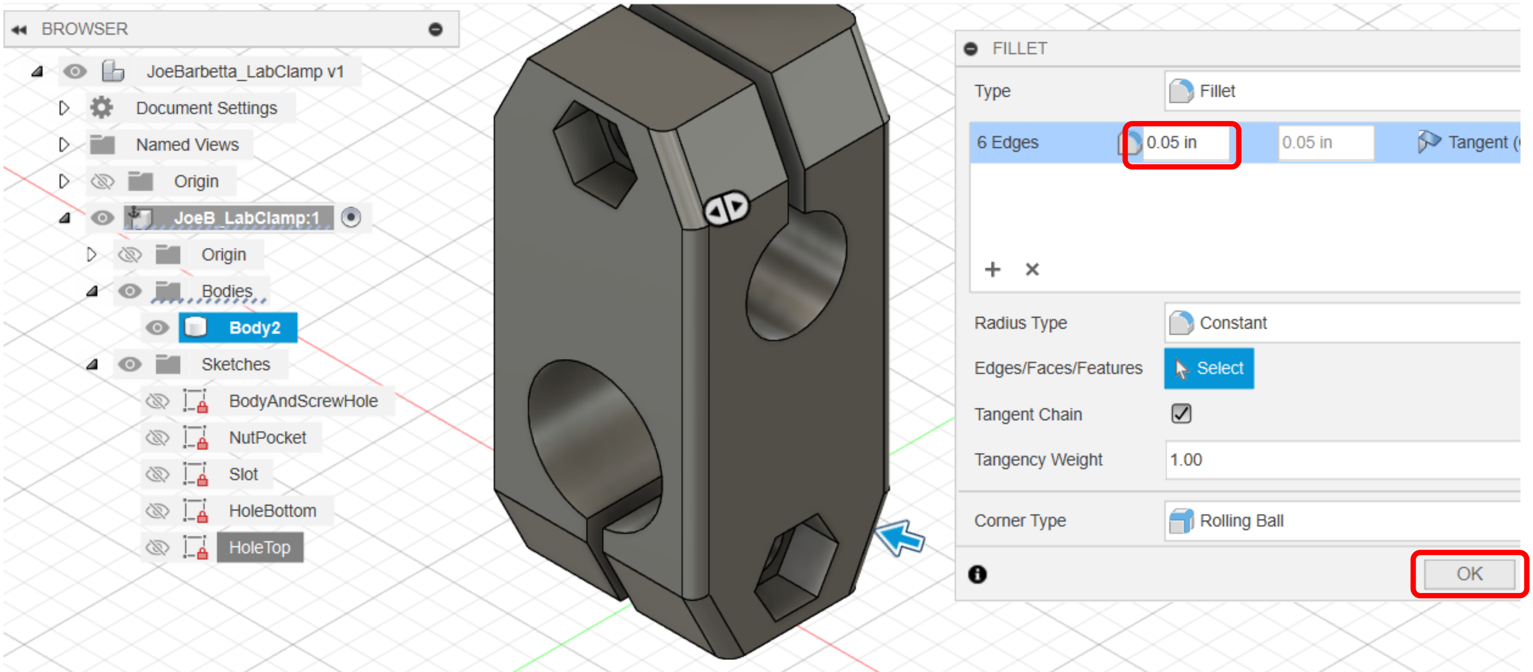
- zoom into the body as shown
- select the **Extrude** tool and click on the **interior of the circle** to highlight it blue
- move the mouse over the area shown to select the **opposite face** of the body (Snap to: -0.75 should show) and click
- click **OK**



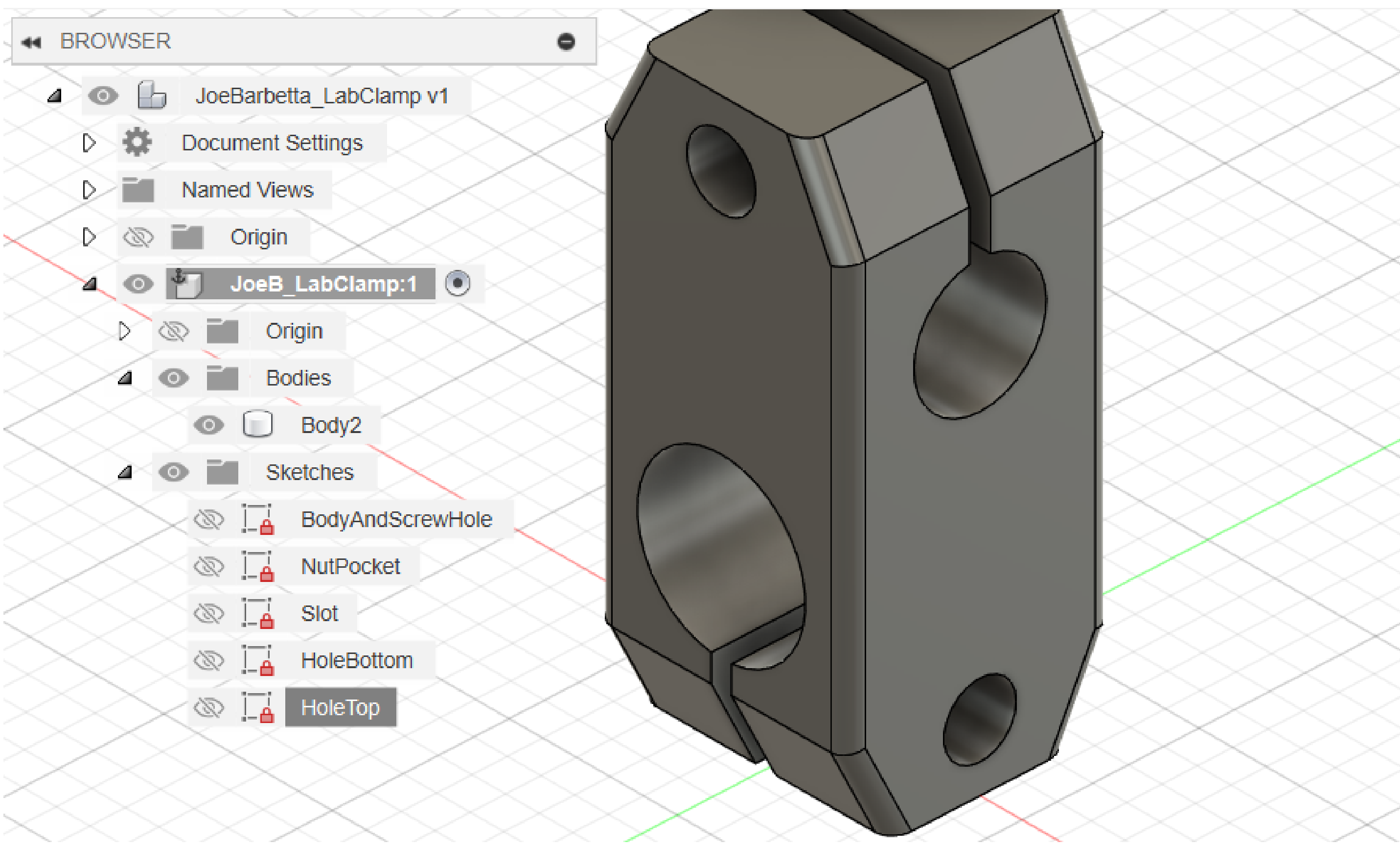
- select the **Fillet** tool. If it is not visible, find it in the MODIFY menu.
- click on the **6 edges** indicated by the arrows to turn them blue



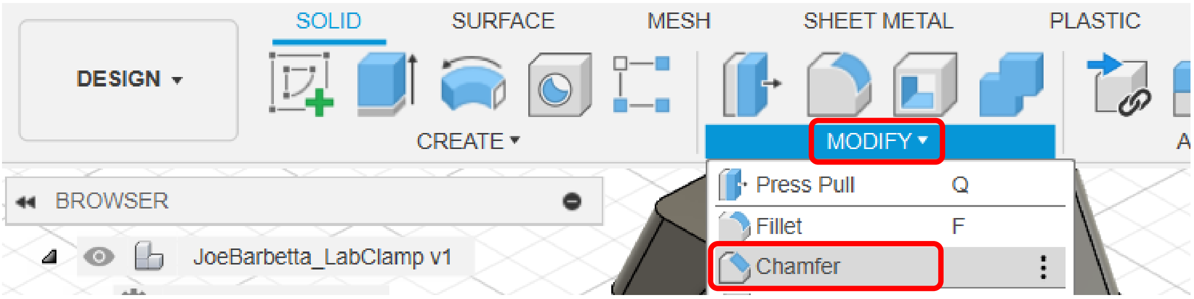
- enter **0.05** and click **OK**



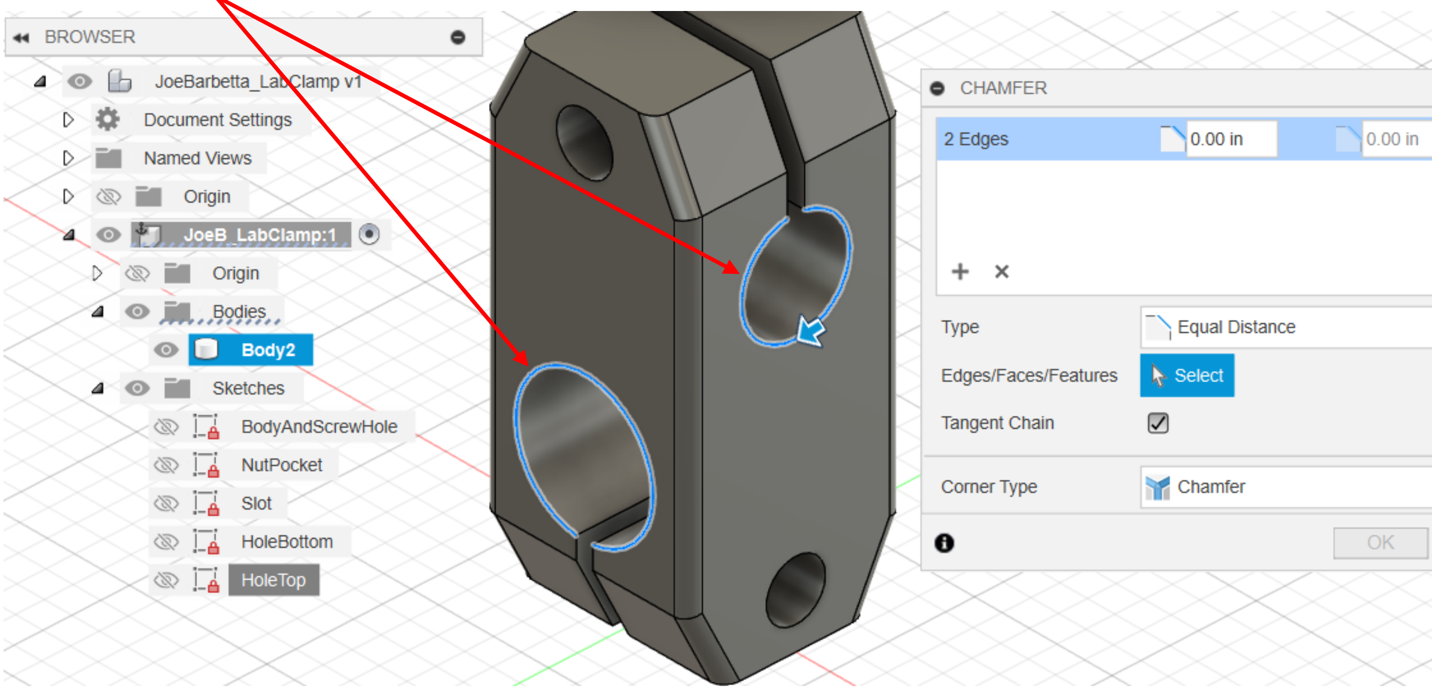
- use the **View Cube** to rotate the view and add a **Fillet** to the other **6 edges**. The result shows below.



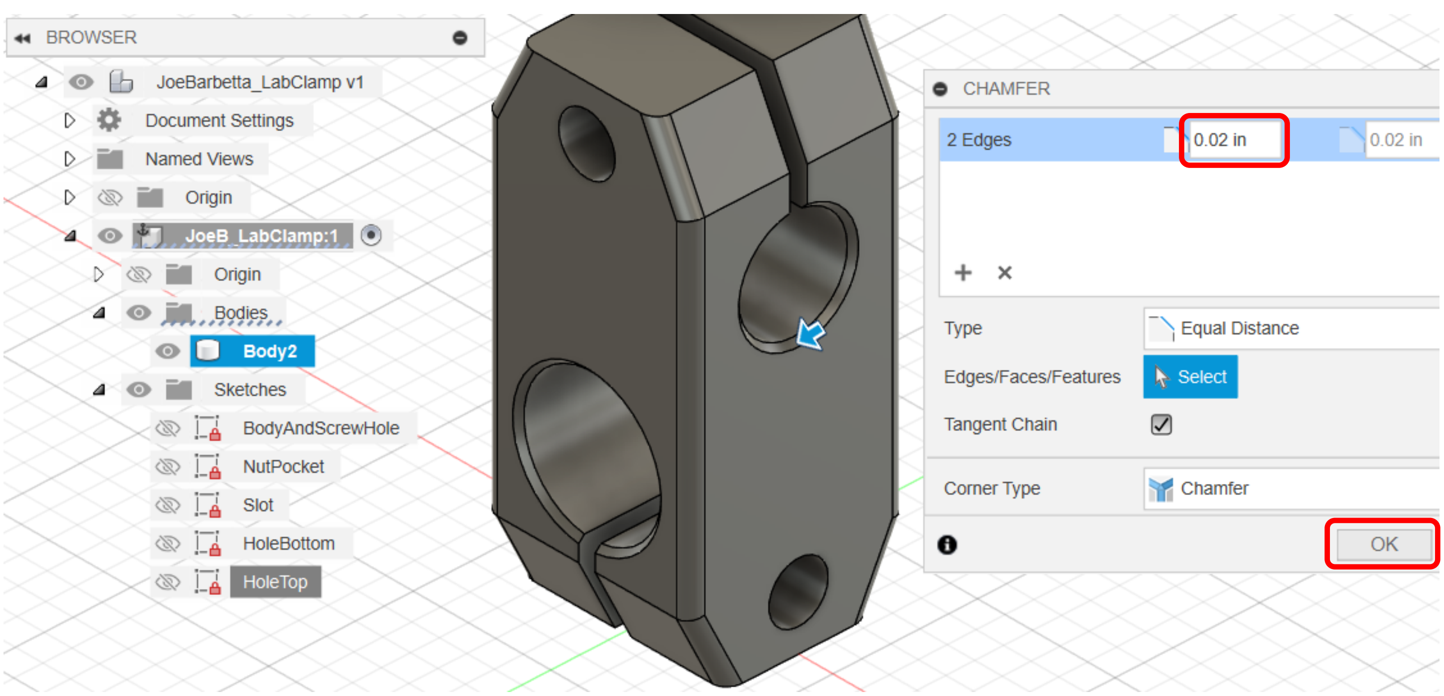
- from the **MODIFY** menu, select **Chamfer**



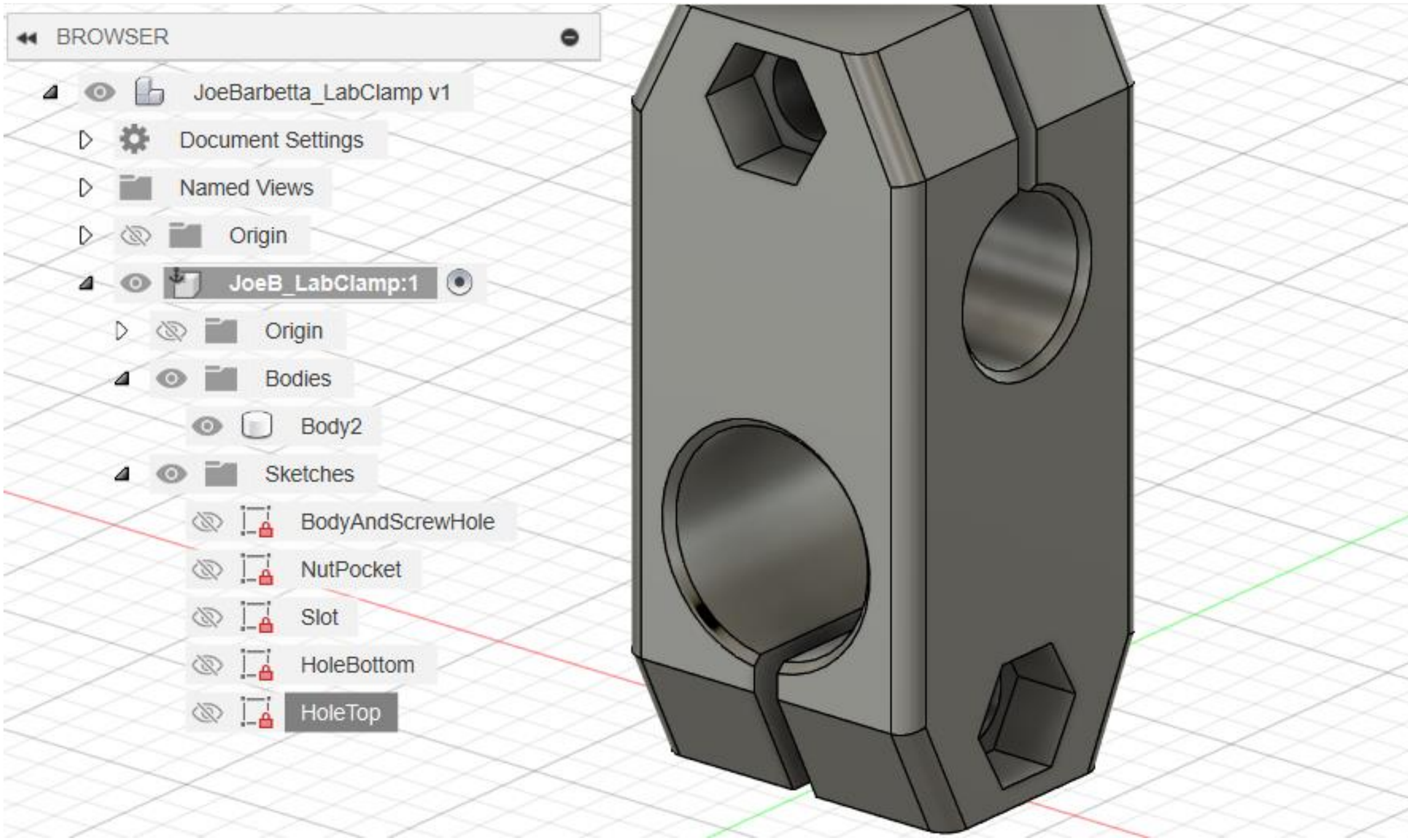
- click on the **edges of the large holes** as shown to highlight them blue



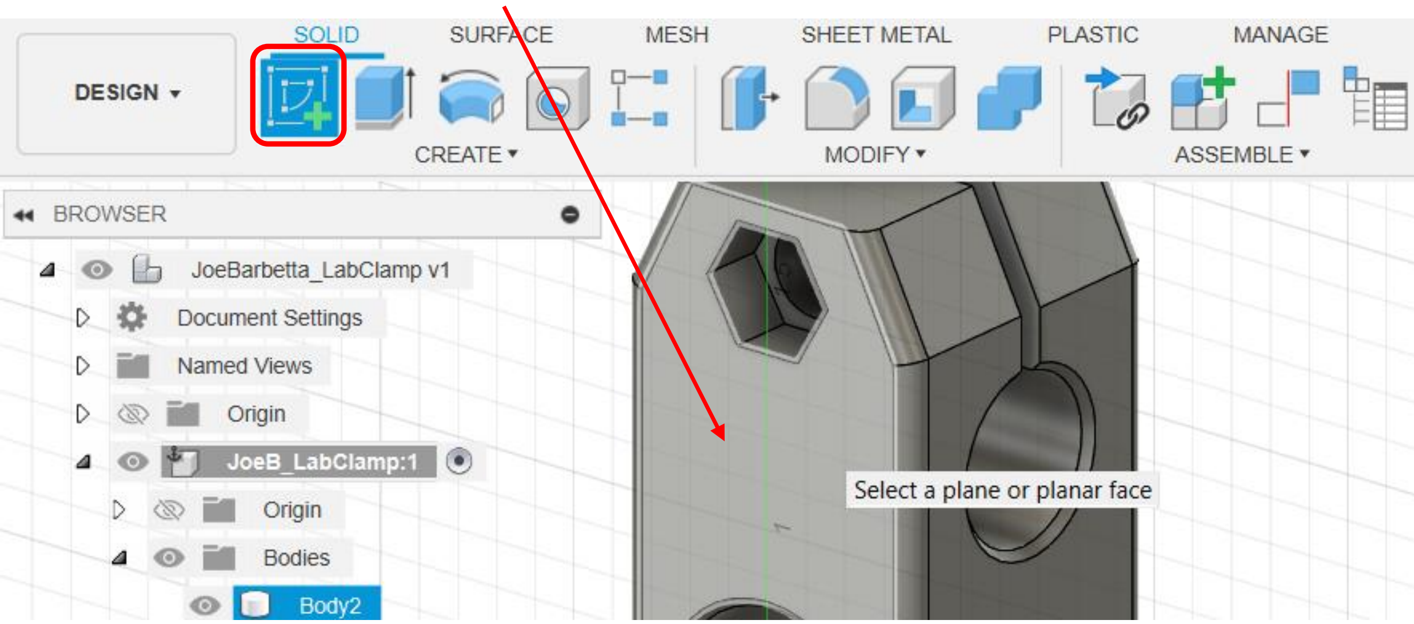
- enter a value of **0.02** and click **OK**



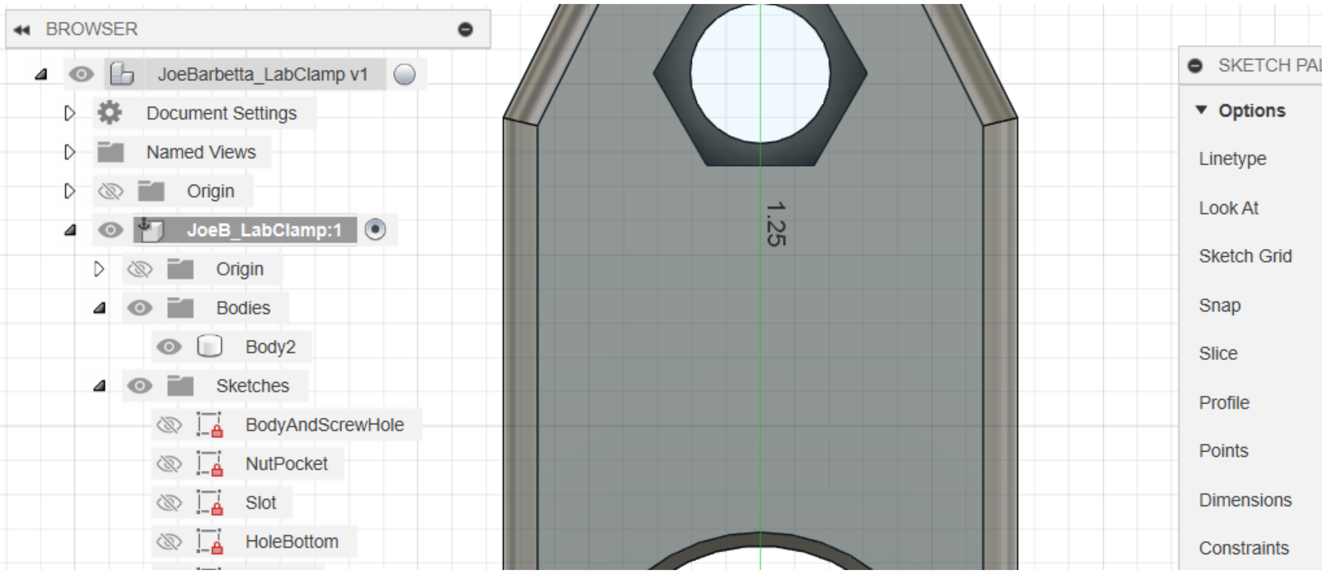
- use the **View Cube** to access the opposite edges of the holes and add **0.02 Chamfers**



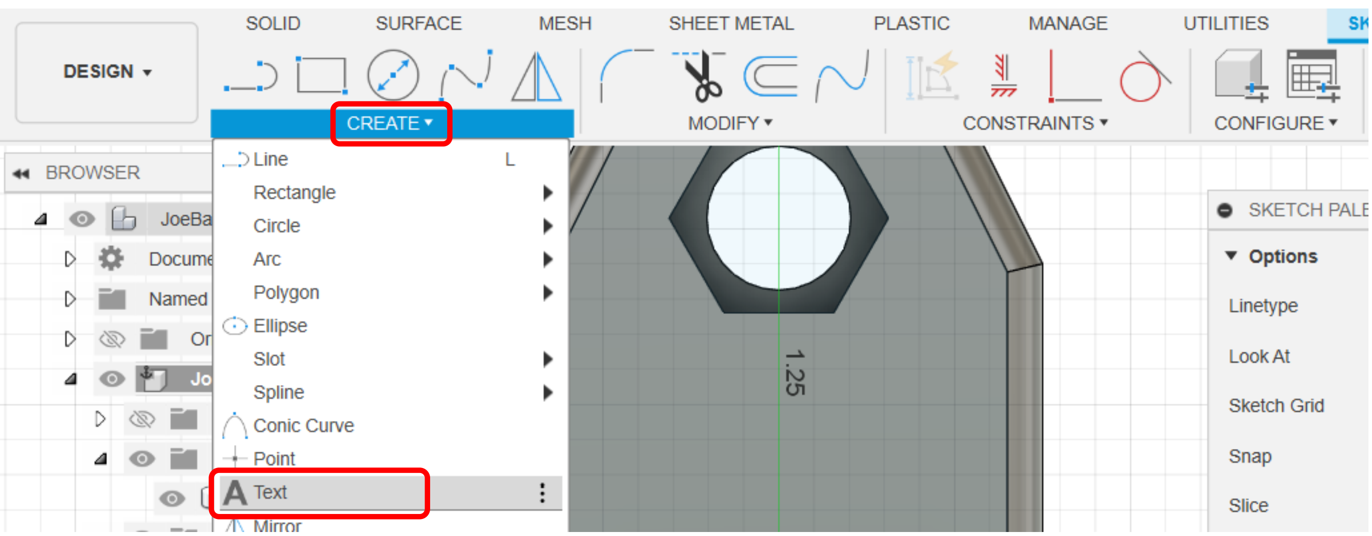
- select **Create Sketch** and click on the surface indicated



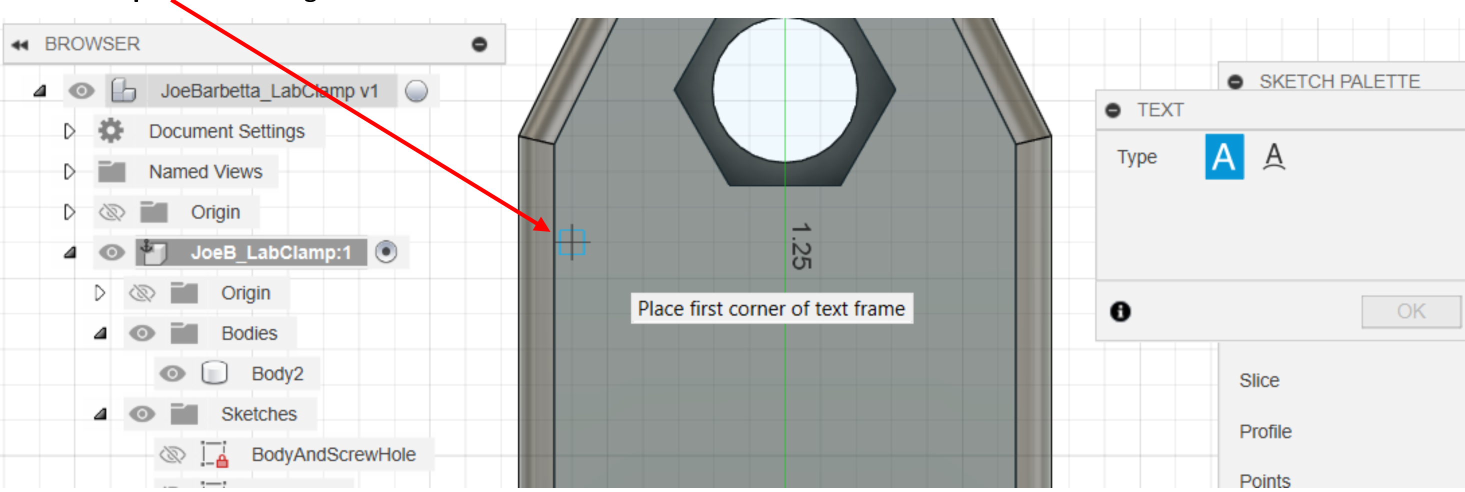
- zoom in to achieve a view similar to that below
- **rename the Sketch** to **Text**



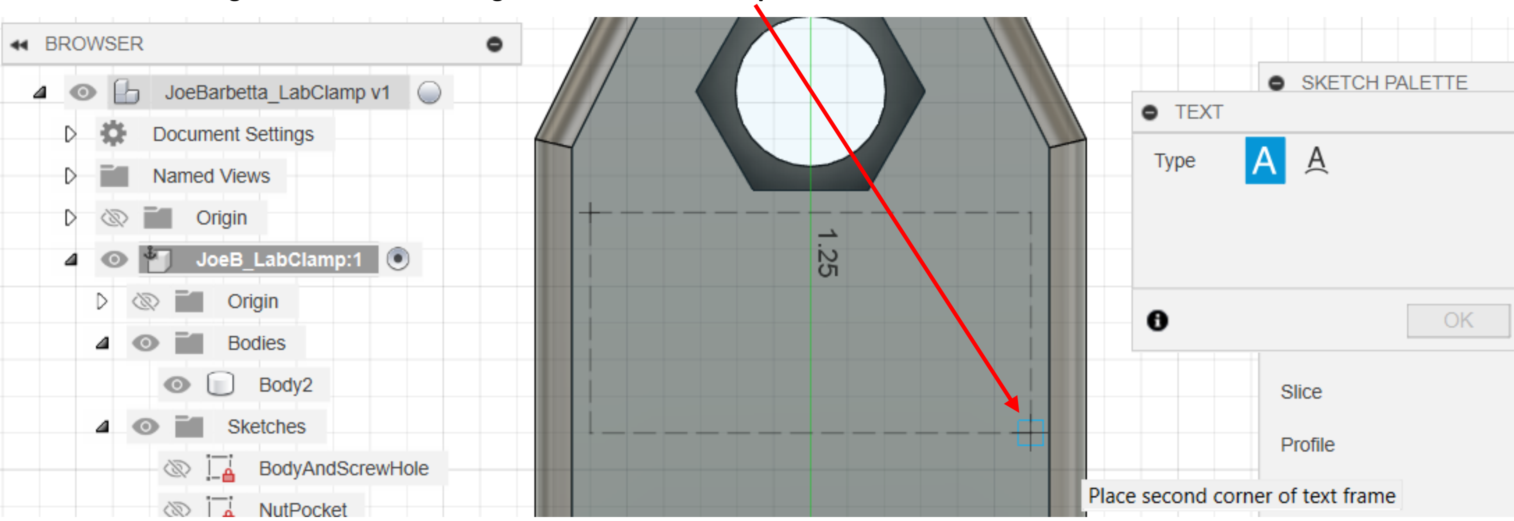
- from the **CREATE** menu, select **Text**
- if a **Parametric Text** window appears, click it **OK** button



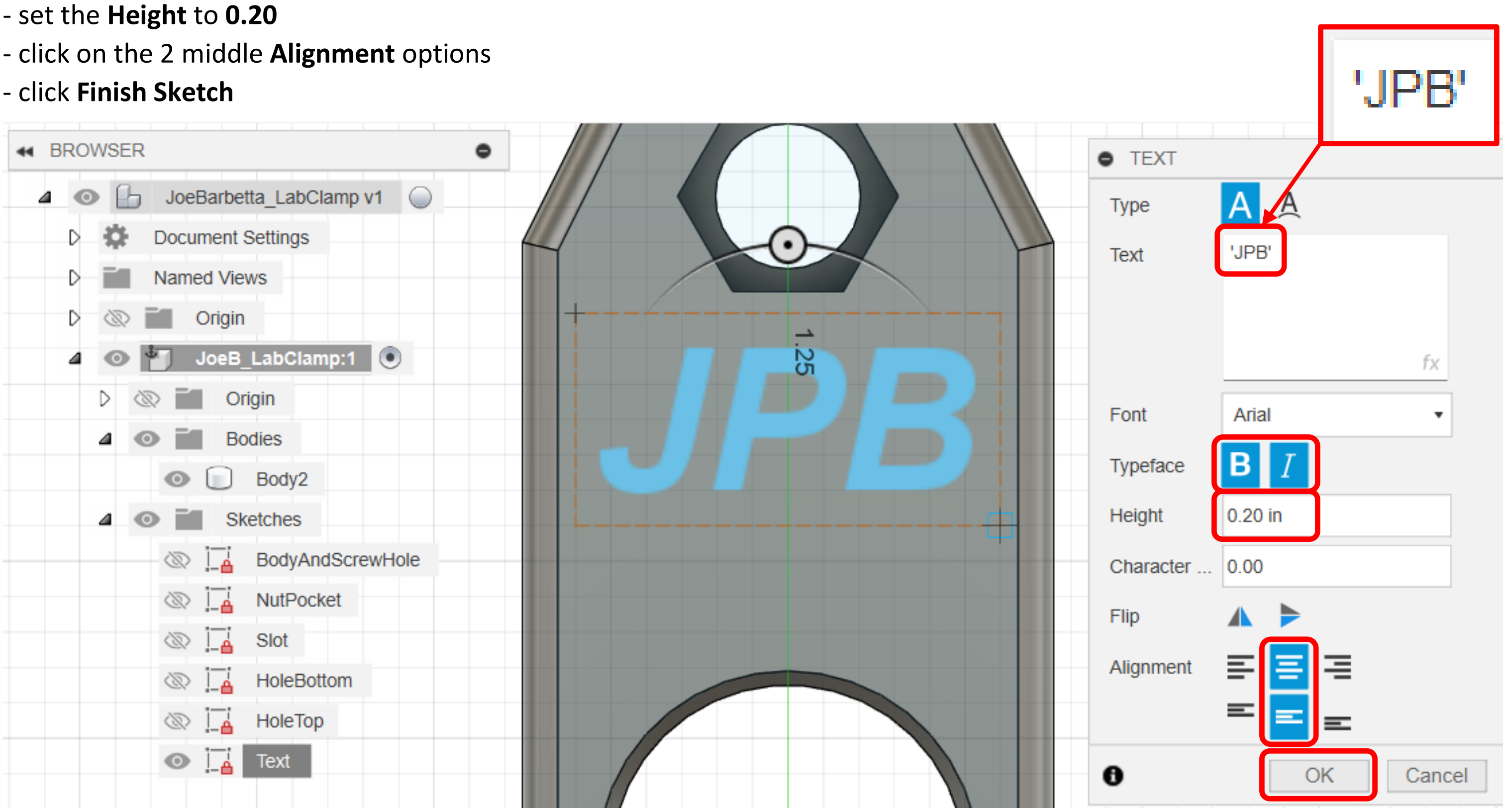
- click on a **point** in the region as shown. The exact location is not critical.



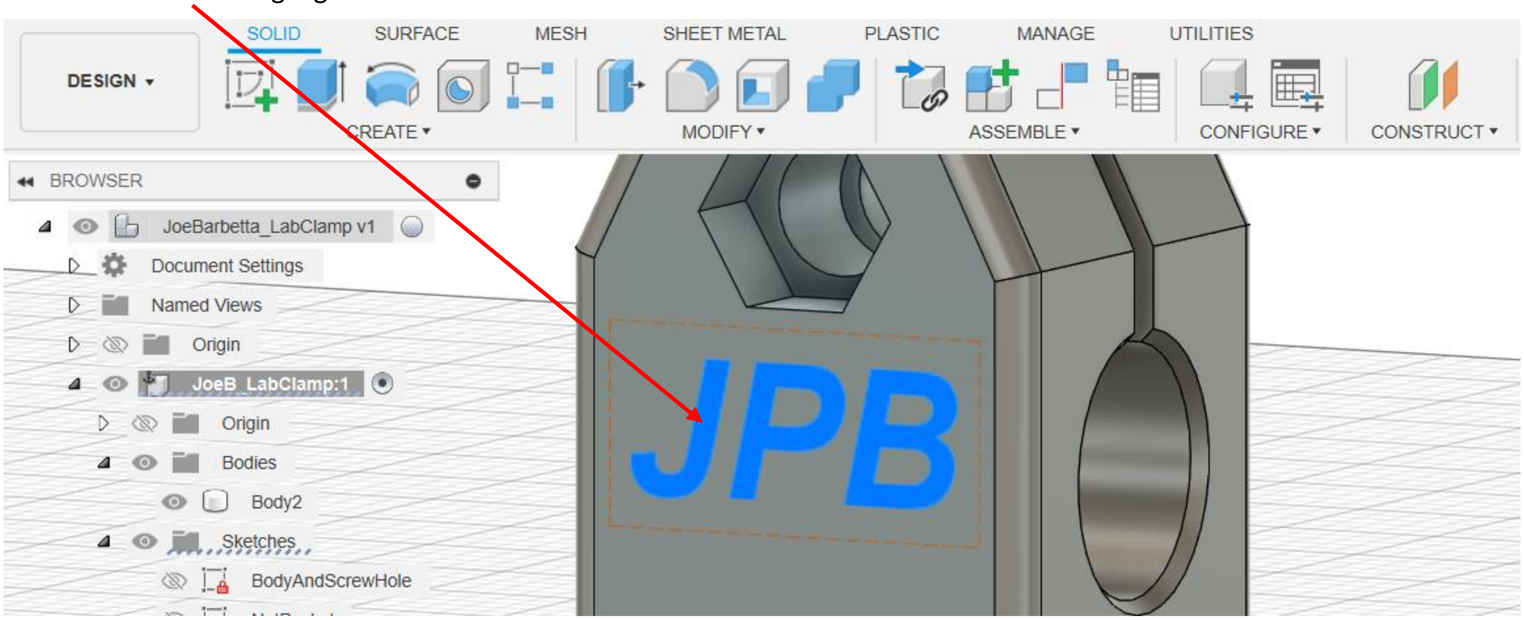
- extend the rectangle down and to the right and click on a **2nd point** as shown. The exact location is not critical.



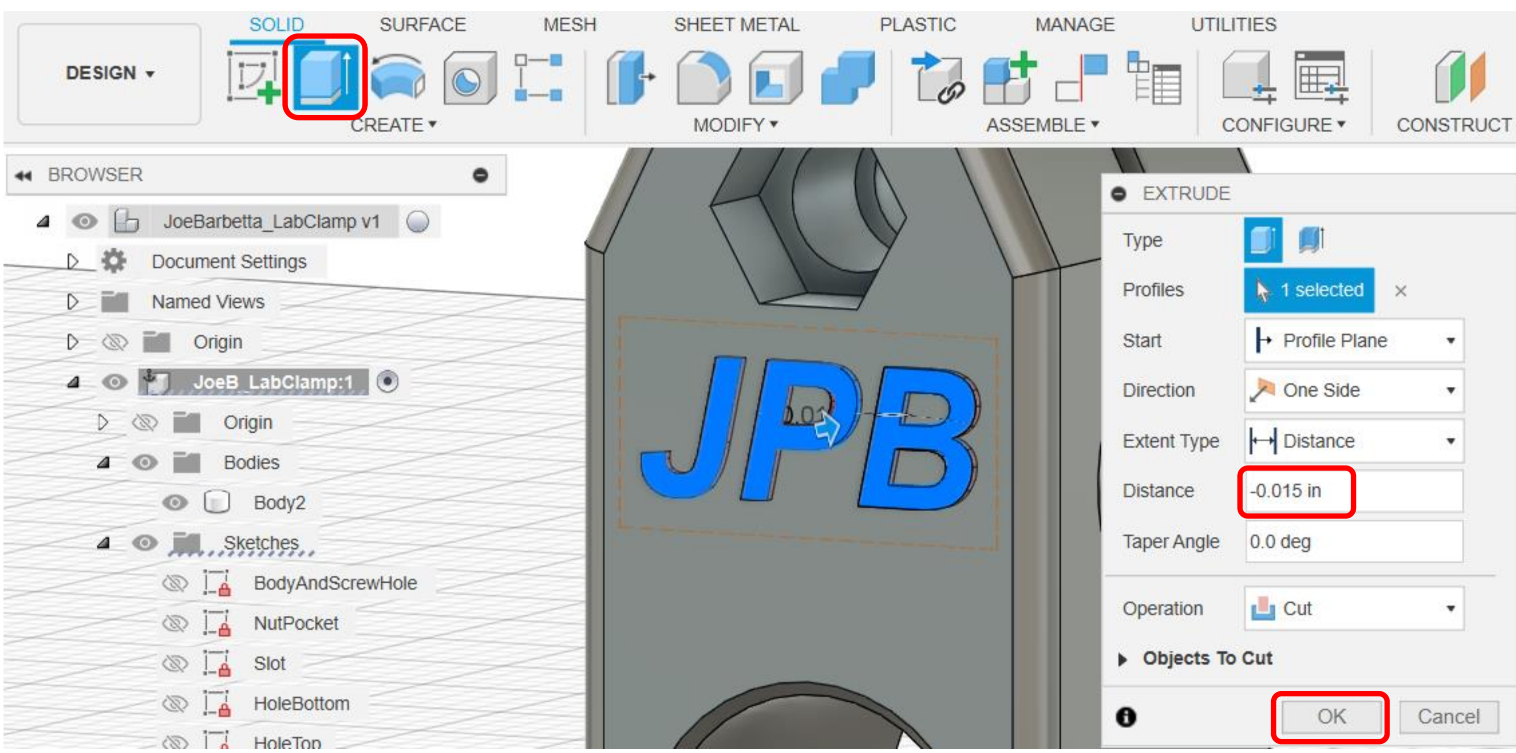
- in the **Text** box, enter your **3 initials preceeded by and followed by a single quote**
- click on the **Bold** and **Italic** icons to highlight them blue
- set the **Height** to **0.20**
- click on the 2 middle **Alignment** options
- click **Finish Sketch**



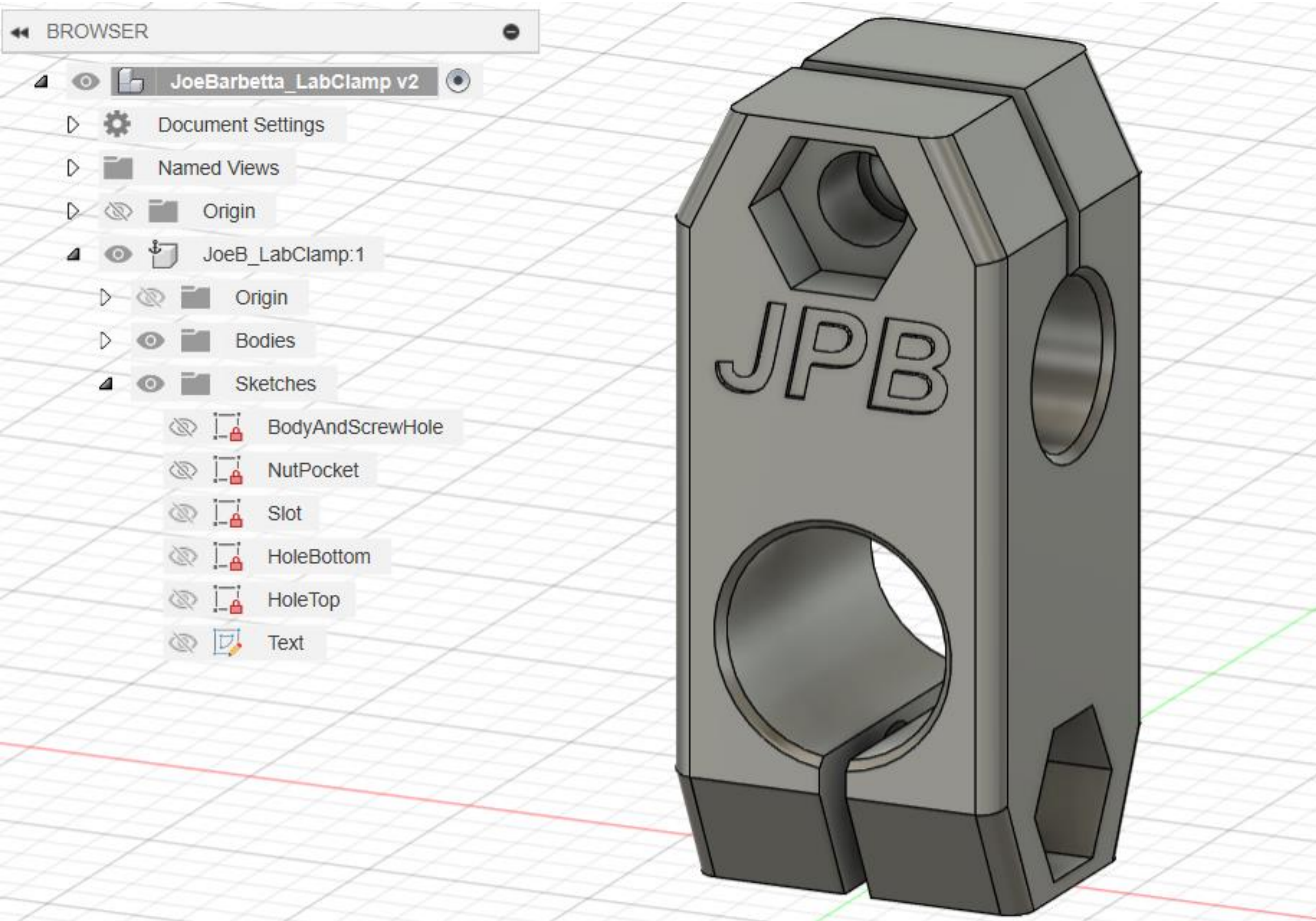
- click on the **text** to highlight it blue



- select the **Extrude** tool
- set **Distance** to **-0.015** (note the minus sign) and click **OK**

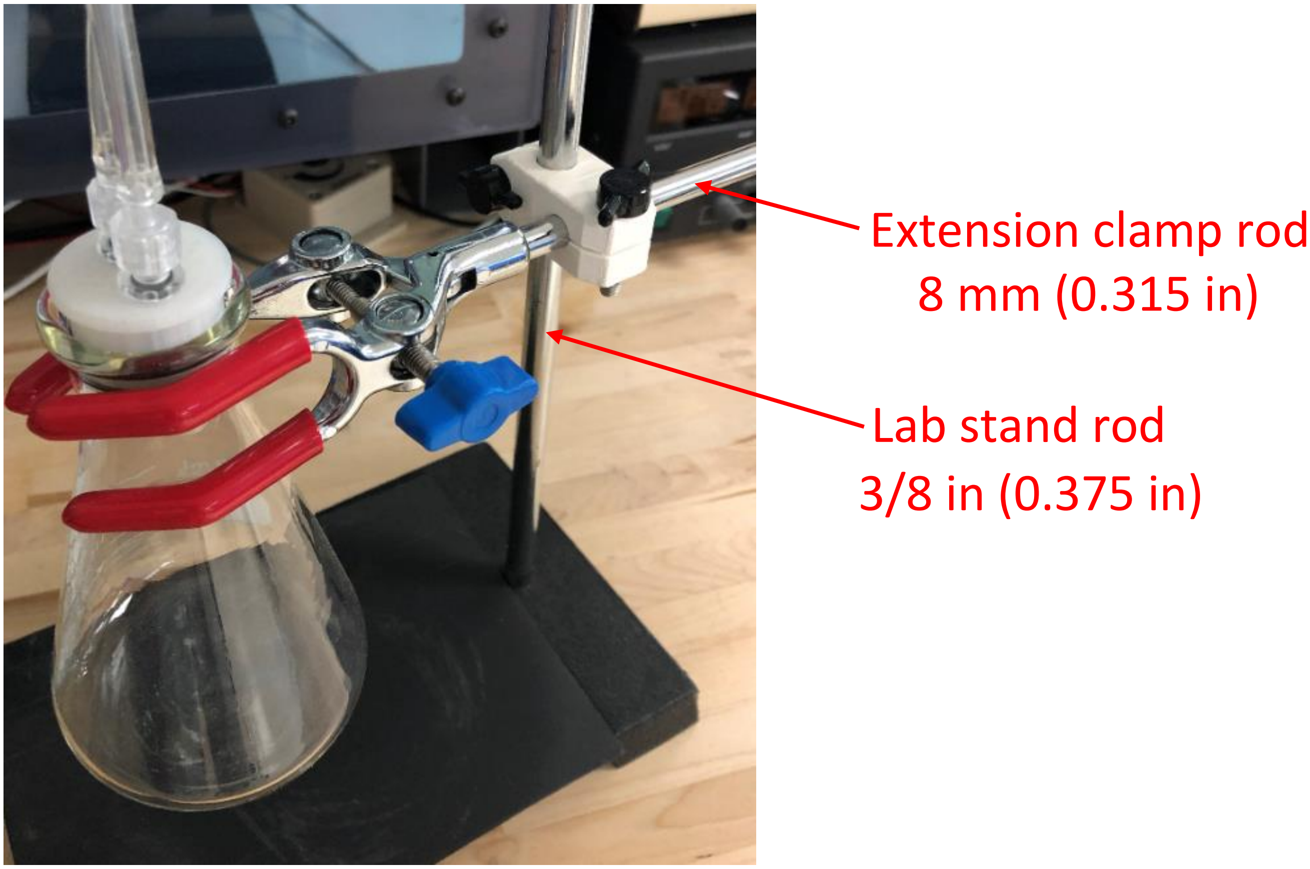


- yell **“That’s a beautiful lab clamp!”**
- ensure the screenshot for submission has the Sketches open and shown on the left
- if you are making a clamp for a mad scientist continue to the next page

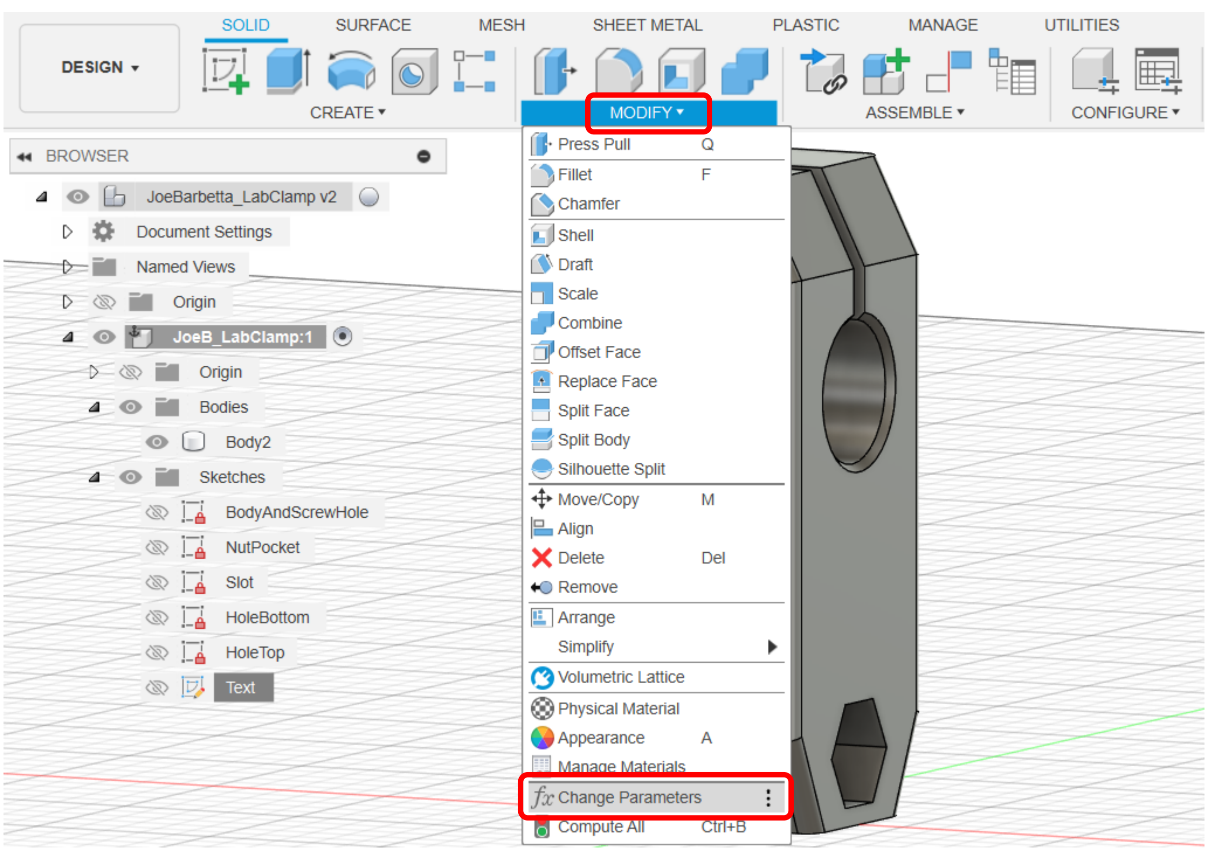


Modifying User Parameters

Determine the hole diameters needed. For this example, the **lab stand** rod has a diameter of **3/8 in** and the **extension clamp** rod has a diameter of **8 mm**.



- from the **MODIFY** menu select **Change Parameters** near the bottom of the menu list

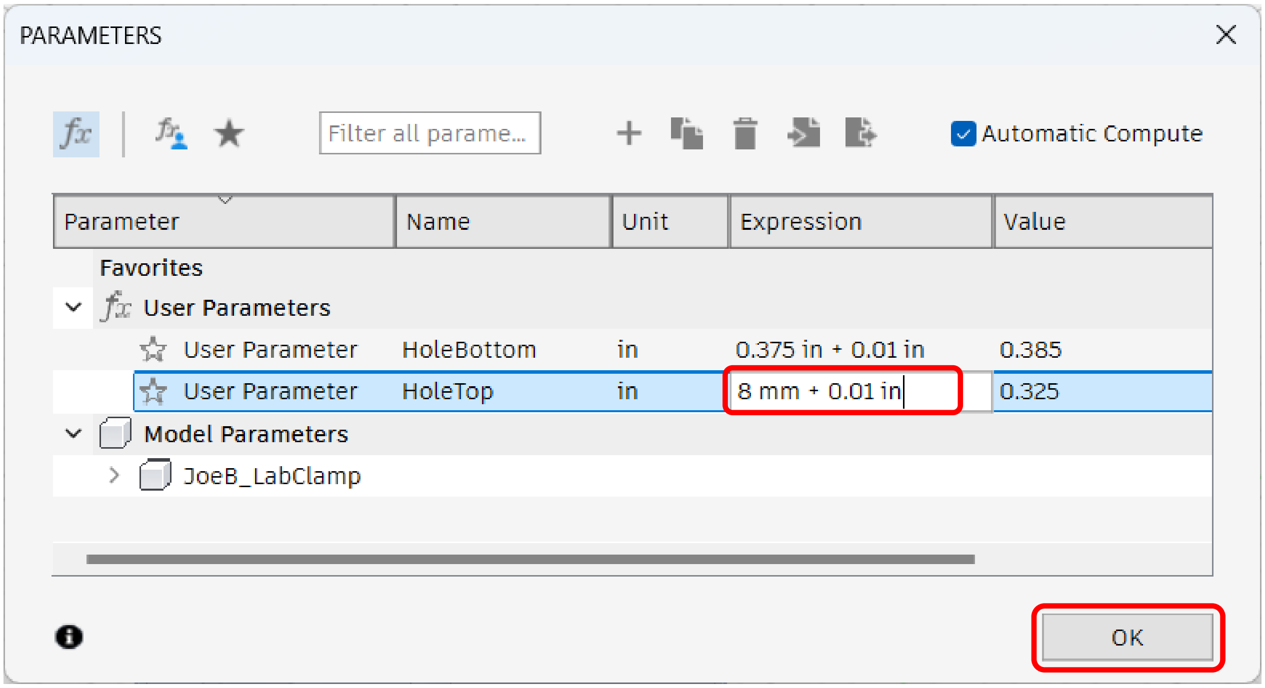


- click on an Expression fields and change the values

The field is called Expression because formulas can be entered.

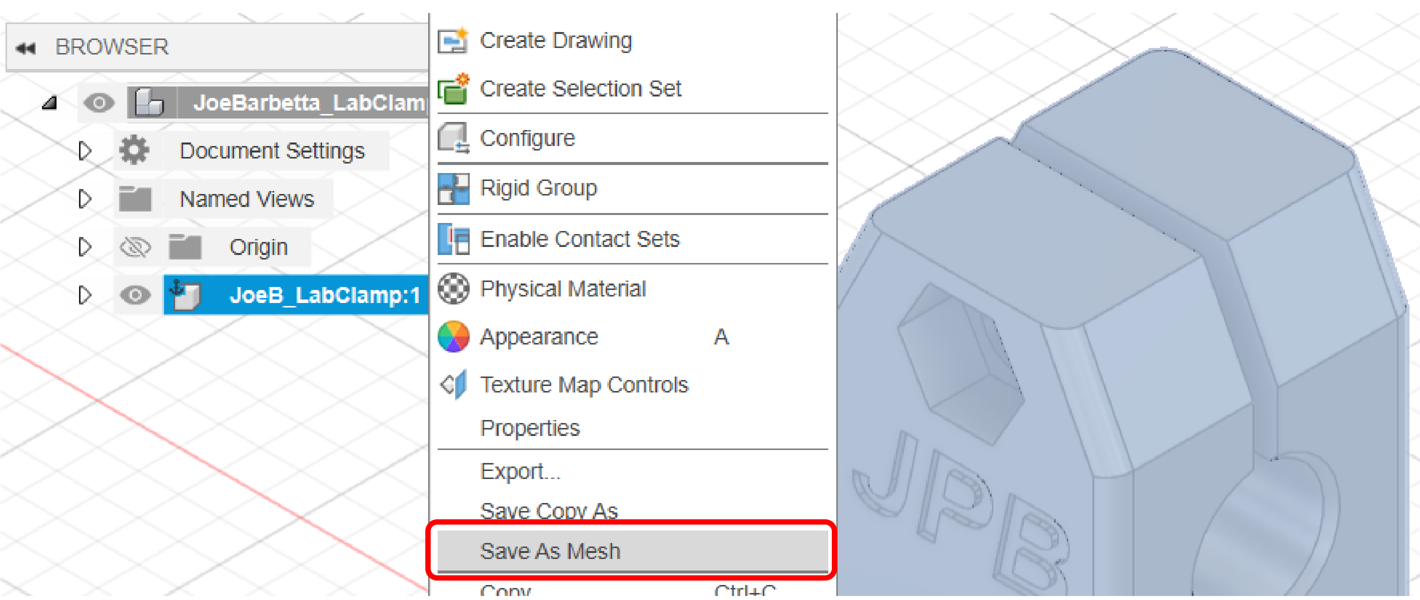
For example the **HoleBottom** was set to **0.375 in + 0.01 in**. The 0.01 will enlarge the hole slightly. A value of 0.385 in could have been entered, but splitting it up communicates the desired value and an enlargement value. Because the top of the hole may cave in slightly at the top the enlargement value can account for this. This value can be based on your experience with your printer. Note that this design is meant to be printed without supports, which can be difficult to remove from holes.

HoleTop is shown set to **8 mm + 0.01 in**. Fusion will perform the conversion to inches and, as with the previous parameter, it communicates the desired hole and the enlargement value.

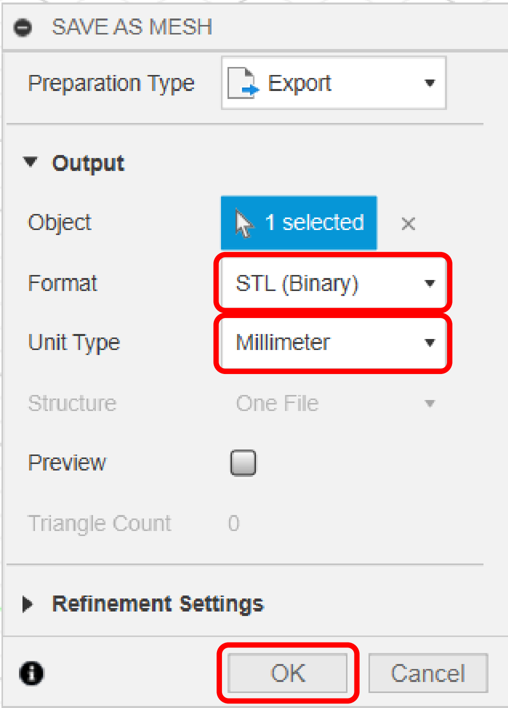


Creating the STL file

- right-click on the **Component Name** and select **Save as Mesh**

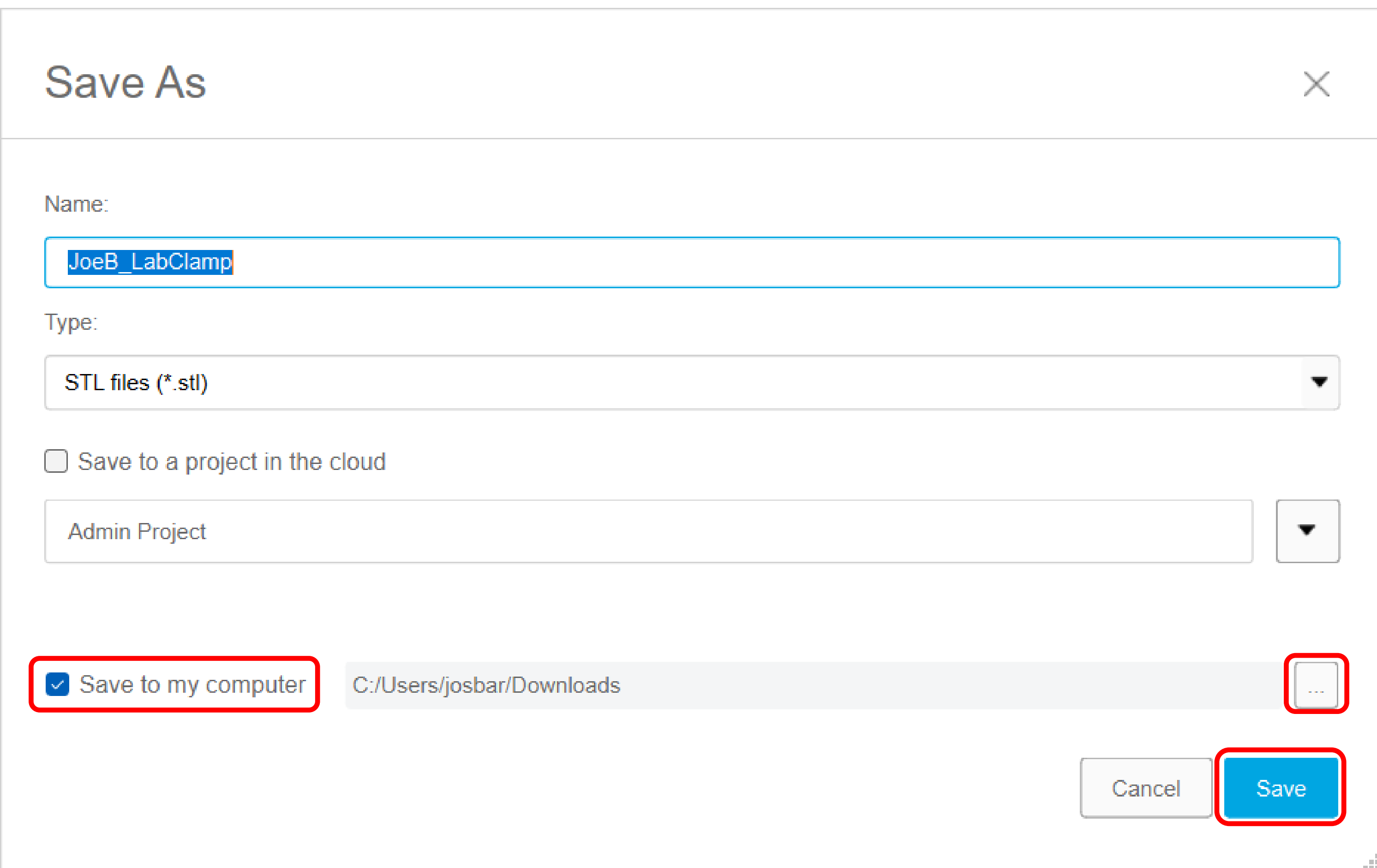


- set **Format** to **STL (Binary)** and **Unit Type** to **Millimeter** and click **OK**

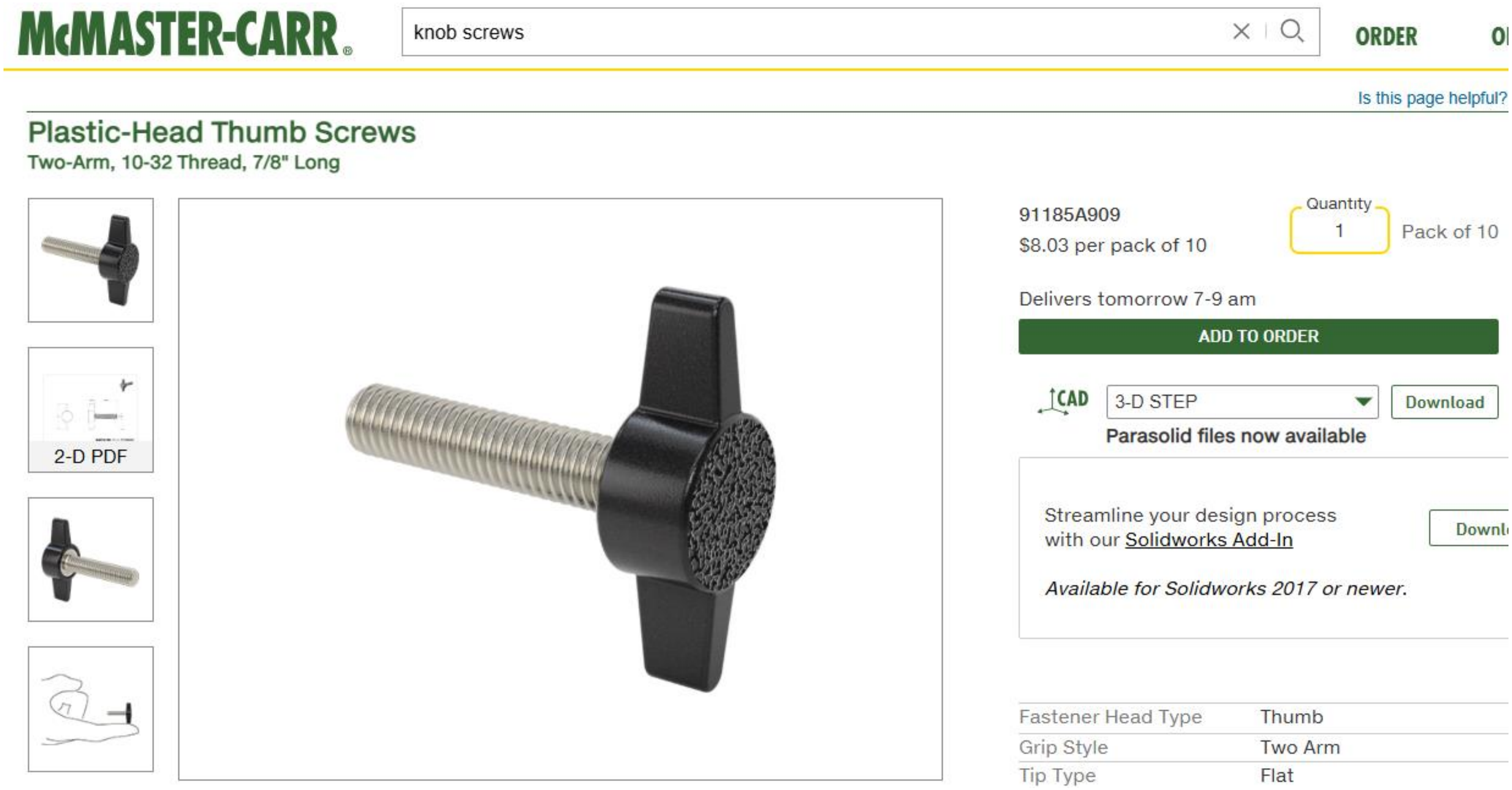


- ensure that **Save to my computer** is checked and note the save location and click **Save**

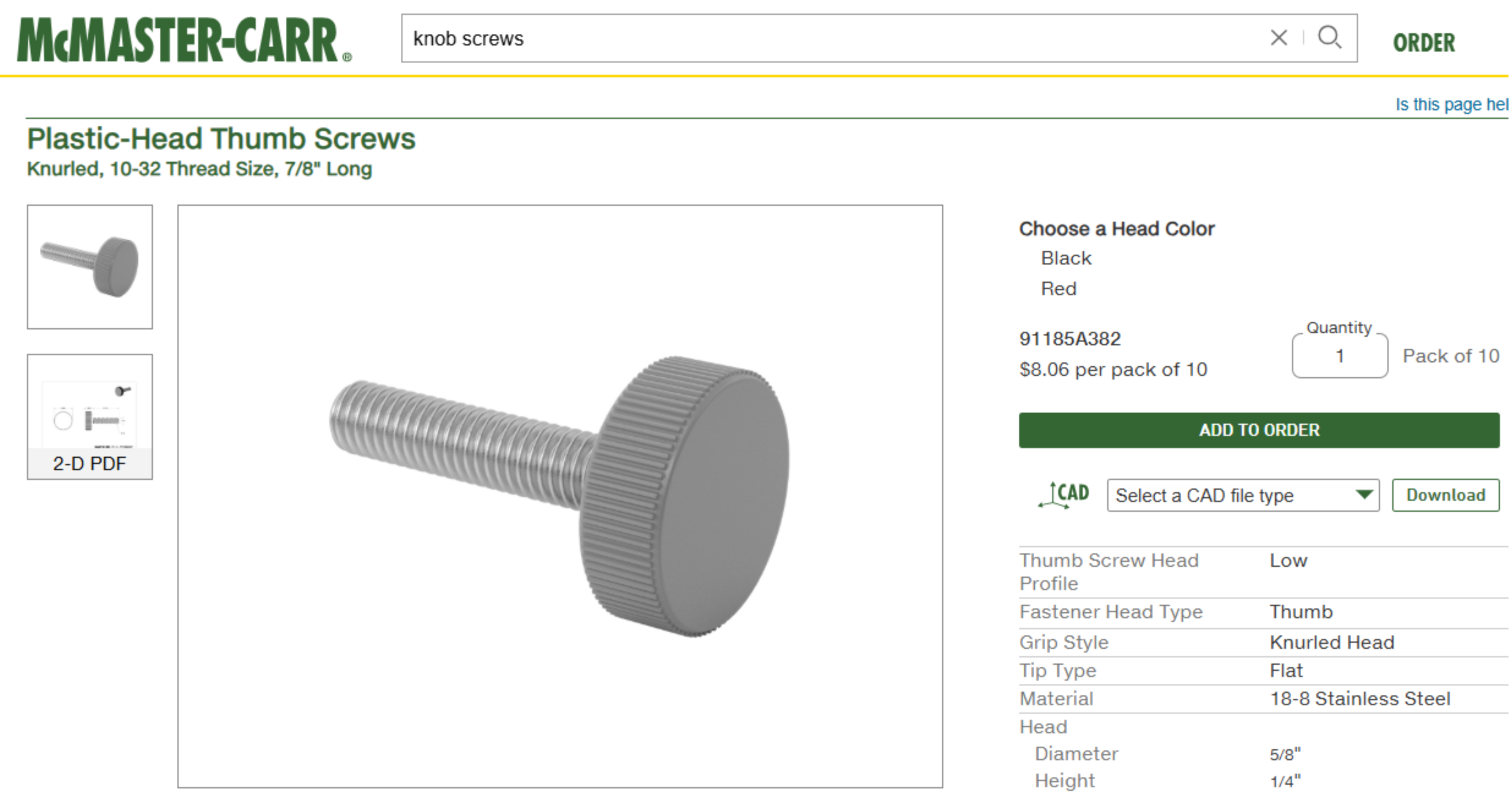
Note that the button at the end of the save location can be used to specify a different folder to save the .stl file to.



This is the Thumb Screw used for the clamp. McMaster-Carr part number: **91185A909**



This is an alternative Thumb Screw. McMaster-Carr part number: **91185A382**



This is the nut used. McMaster-Carr part number: **90730A411**

Note that this is a Narrow-Profile nut and the the typical 10-32 size nut.