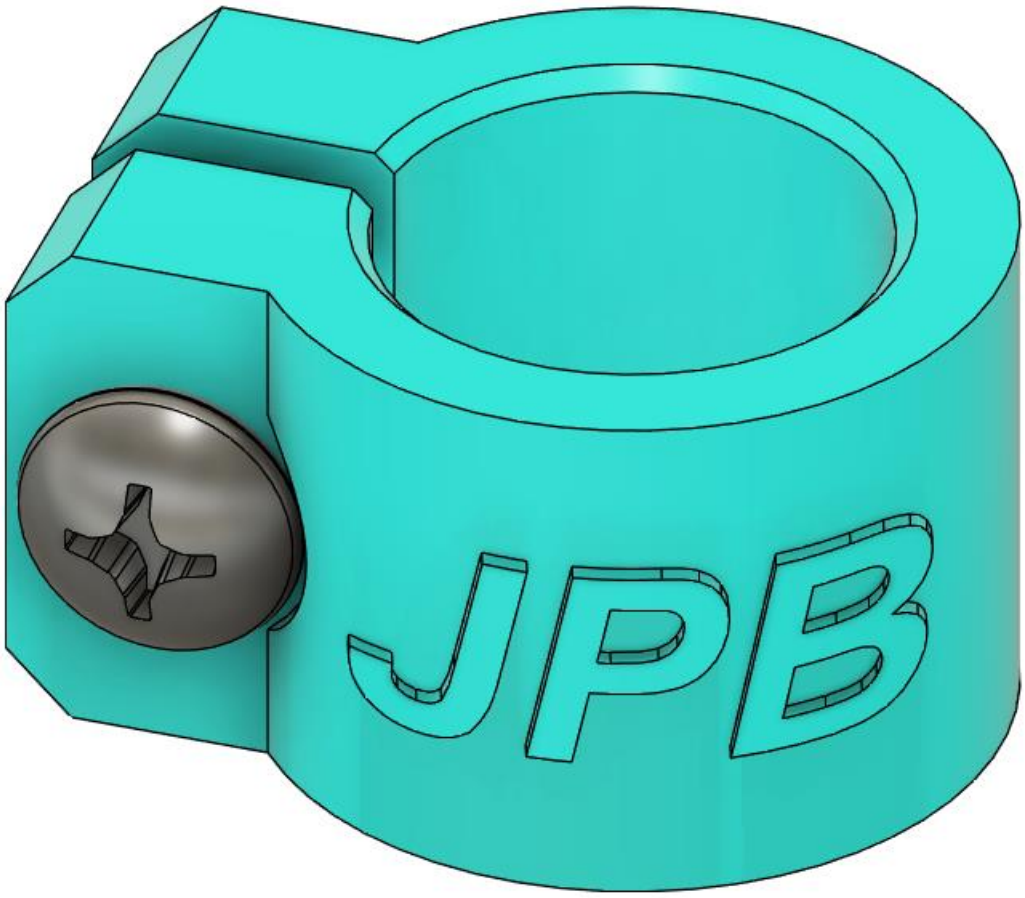
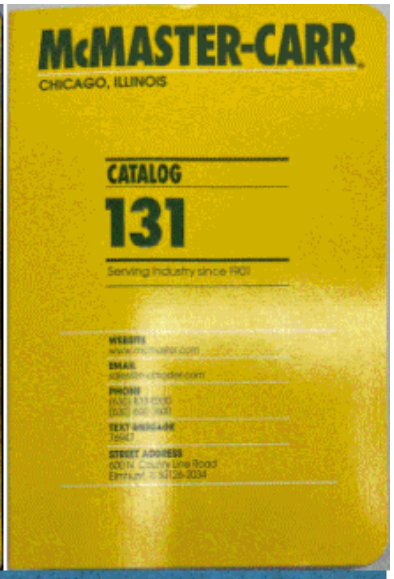
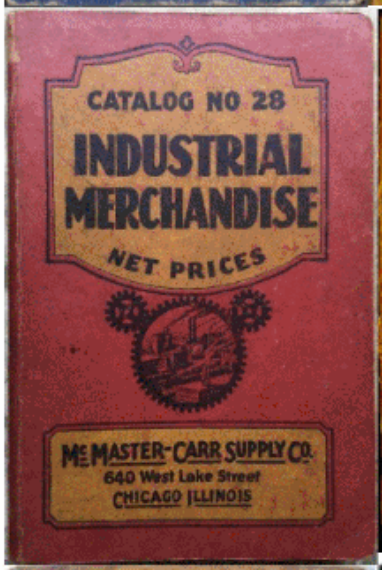
Mr. Barbetta's *"I don't care about 3D Printing. I just want the art credit."* class

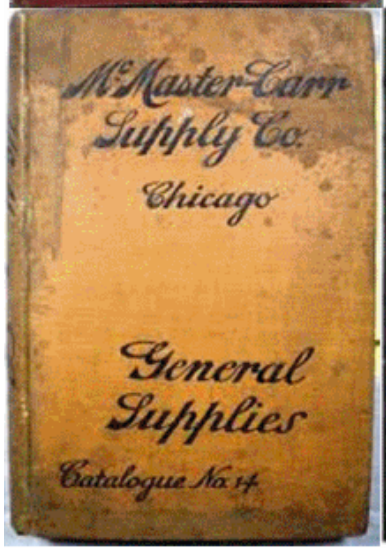
Make a hose clamp.



Today's Lesson is Sponsored by McMaster-Carr





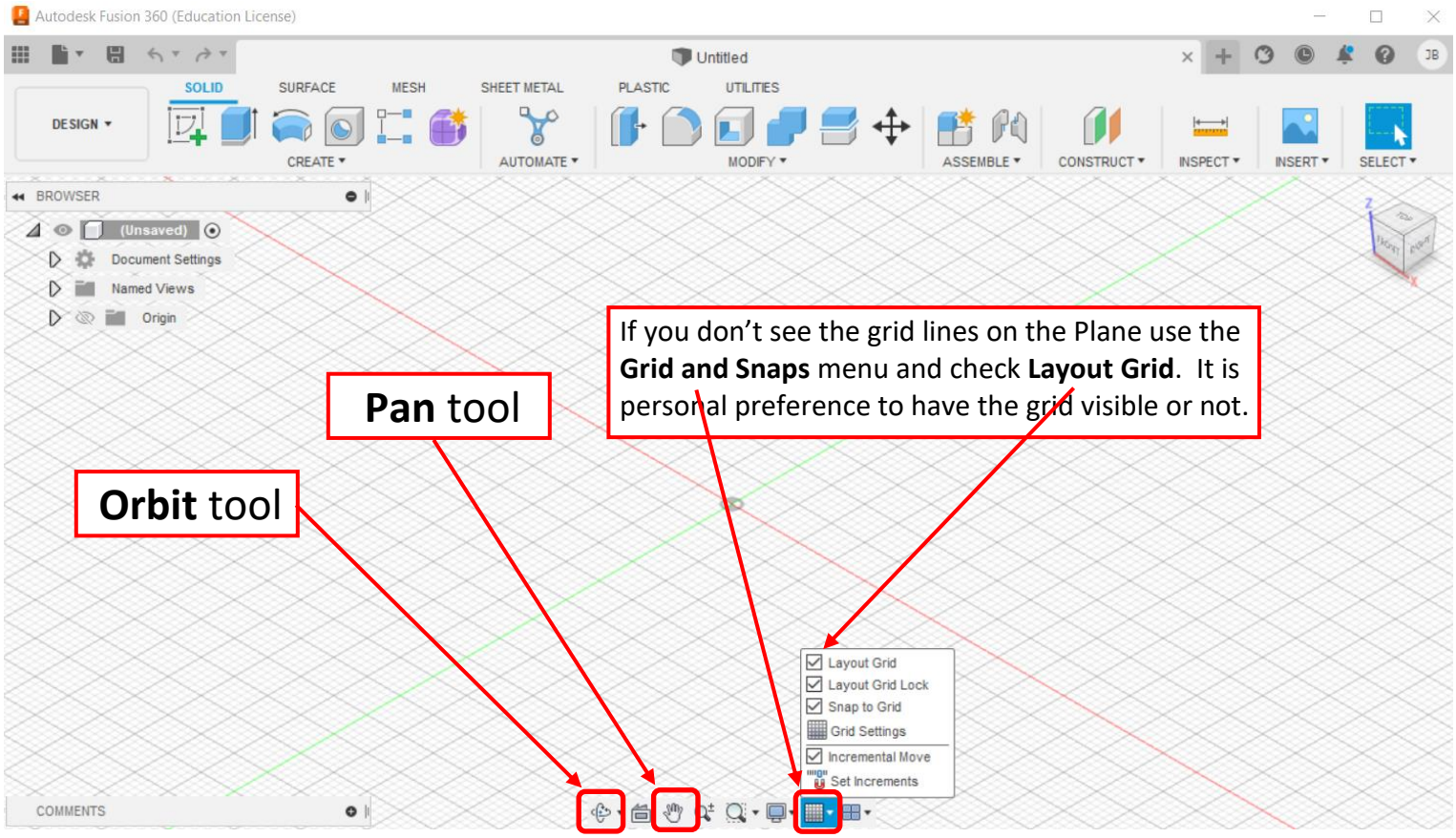


Contents

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

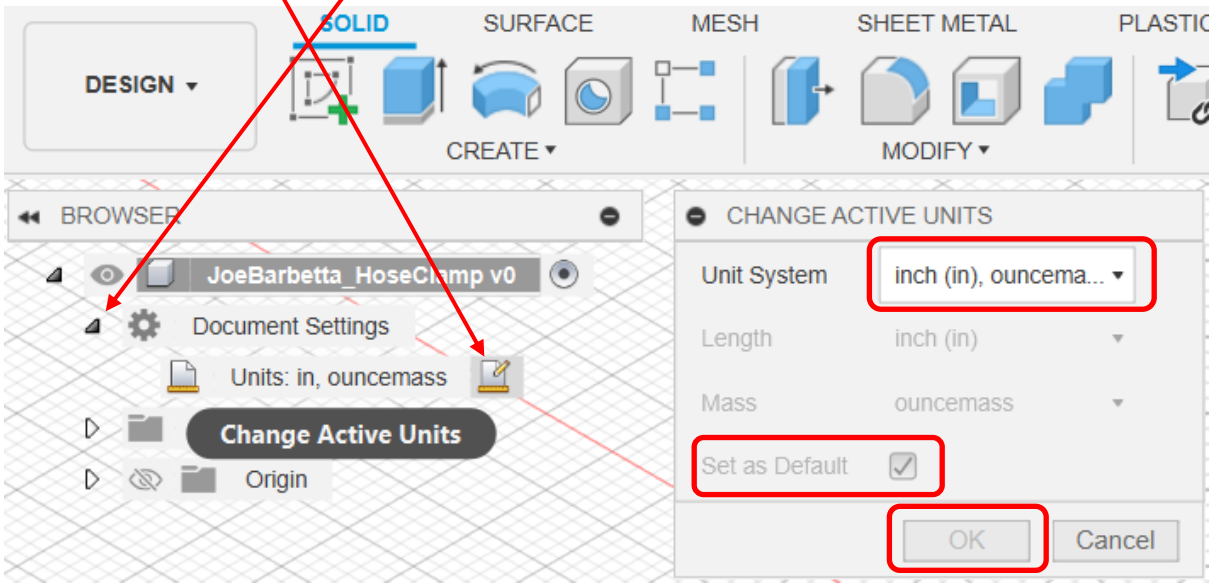
Starting a Design in Fusion (START HERE)

- 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 **_HoseClamp** e.g. **JoeBarbetta_HoseClamp** (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.

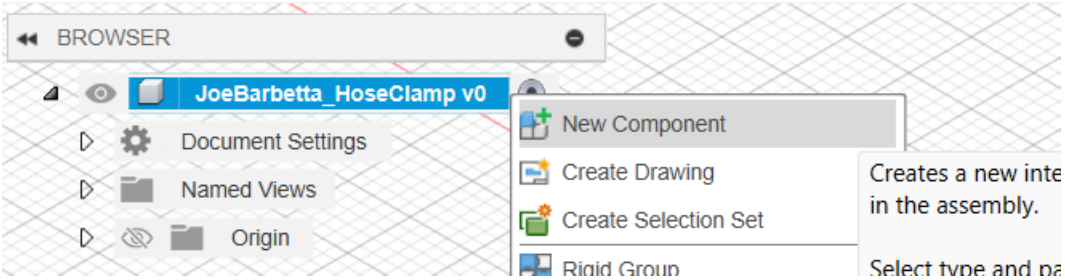


Note that the default units are in mm, which we just changed to inches.

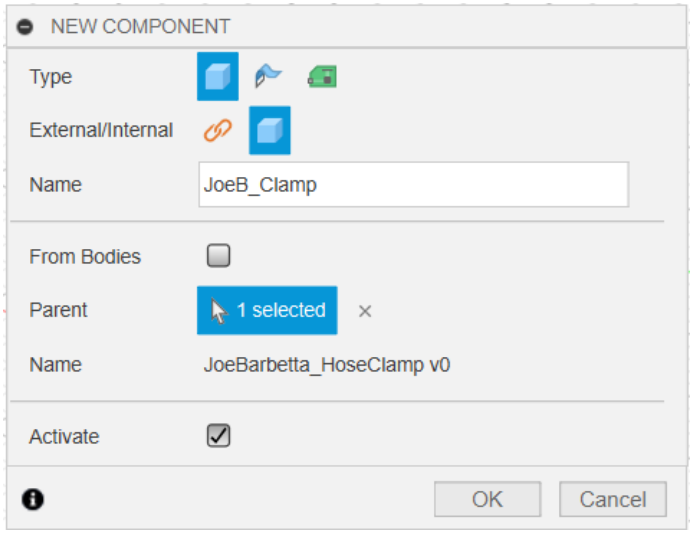
Did you know that the default units have changed over the years? The earliest version used cubits as the default unit.

Creating a New Component

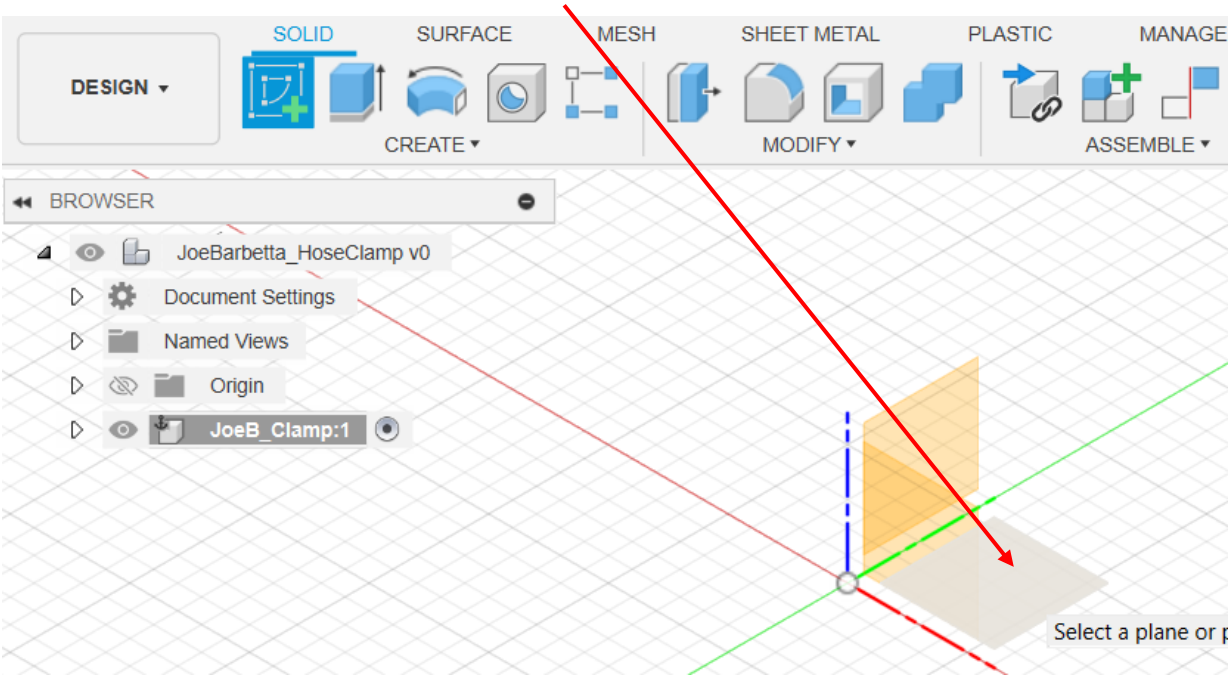
- right-click on the Project Name and select New Component



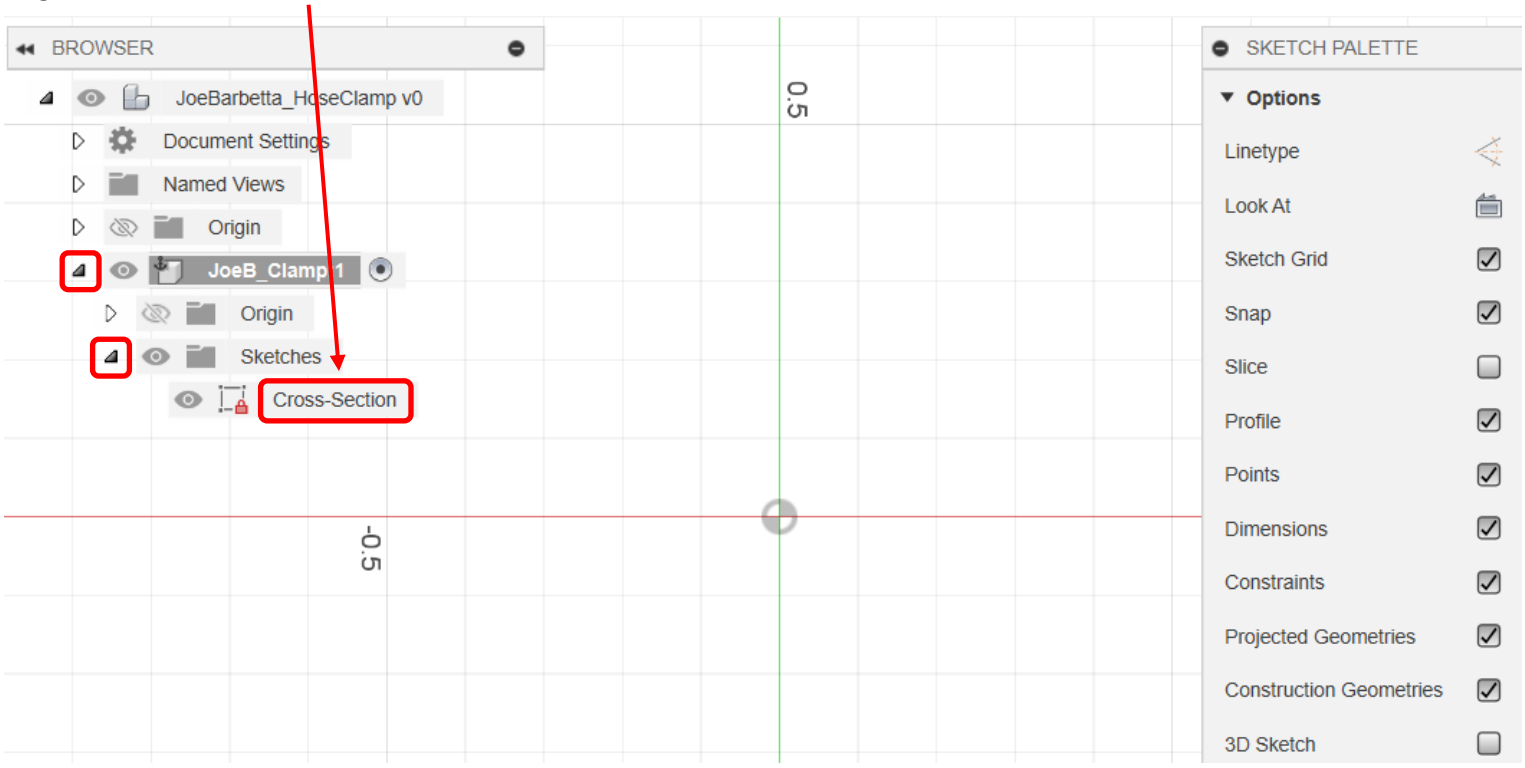
- in the **Name** box enter your **first name followed by your last name initial and _Clamp** (note the underscore), **e.g. JoeB_Clamp**
- click OK



- select **Create Sketch** and click on the **bottom rhombus**

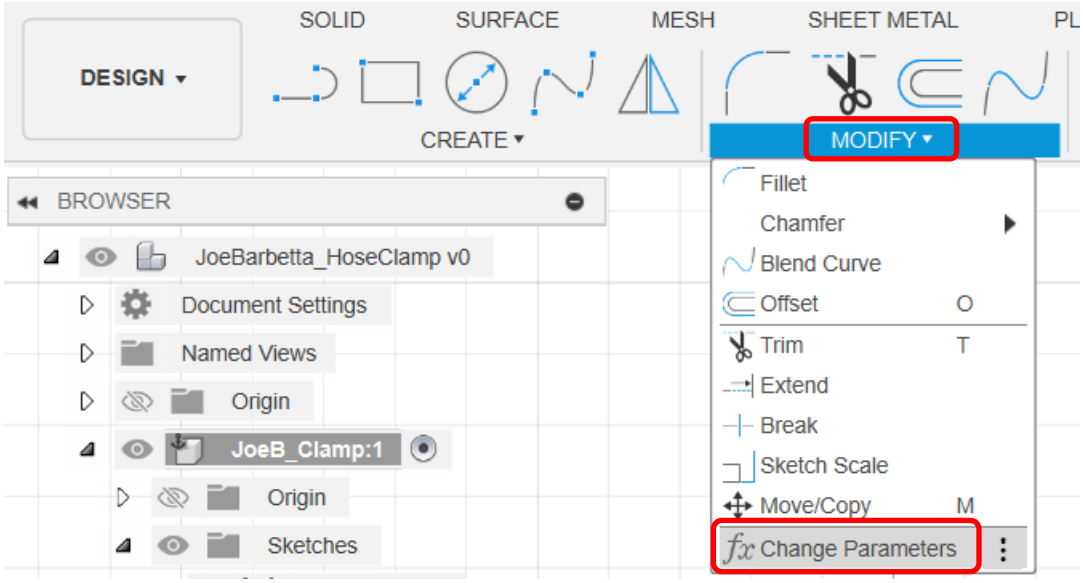


- zoom in to achieve a view similar to that below
- click on the **arrows** next to the **Component Name** and the **Sketches folder**
- right-click on the **Sketch name**, and select **Rename**, and rename it to **Cross-Section**



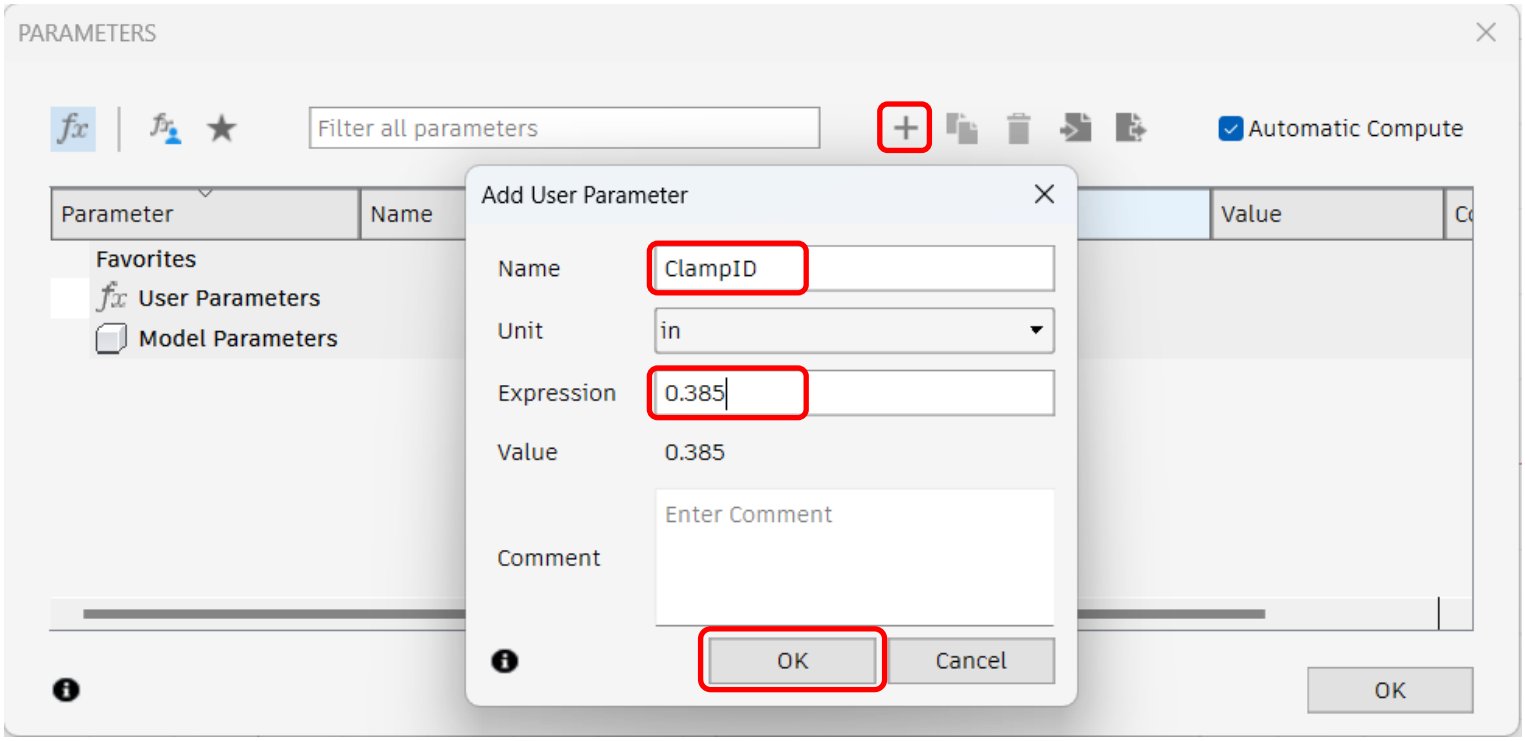
Setting User Parameters

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

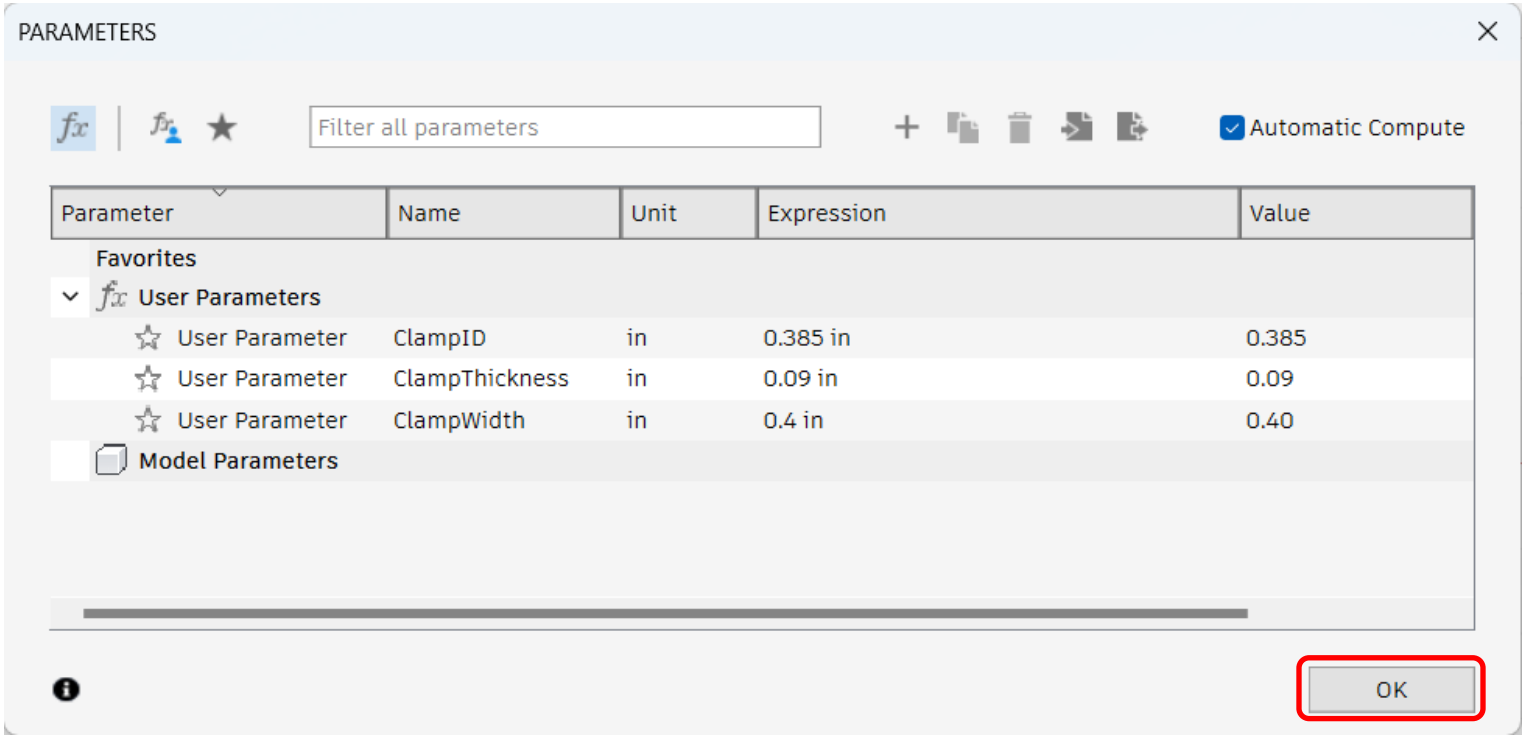


- click on the top **+** icon
- set the **Name** to **ClampID**, the **Expression** to **0.385**, and click **OK**

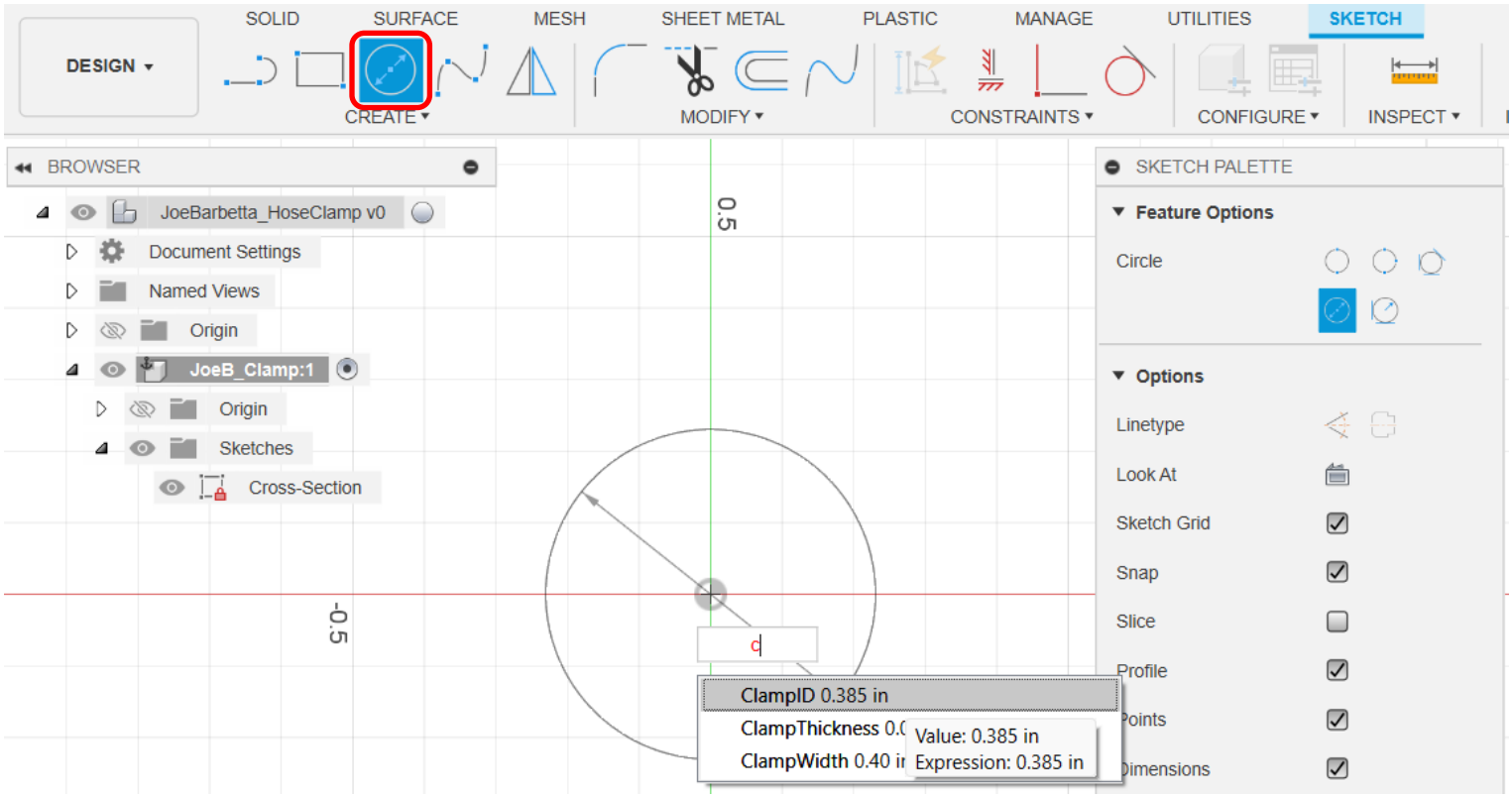
ID stands for Inner Diameter, which should be slightly larger than the OD (Outer Diameter) of the tube that this clamp would attach to. The value of 0.385 is 0.01” larger than a 3/8” OD tube.



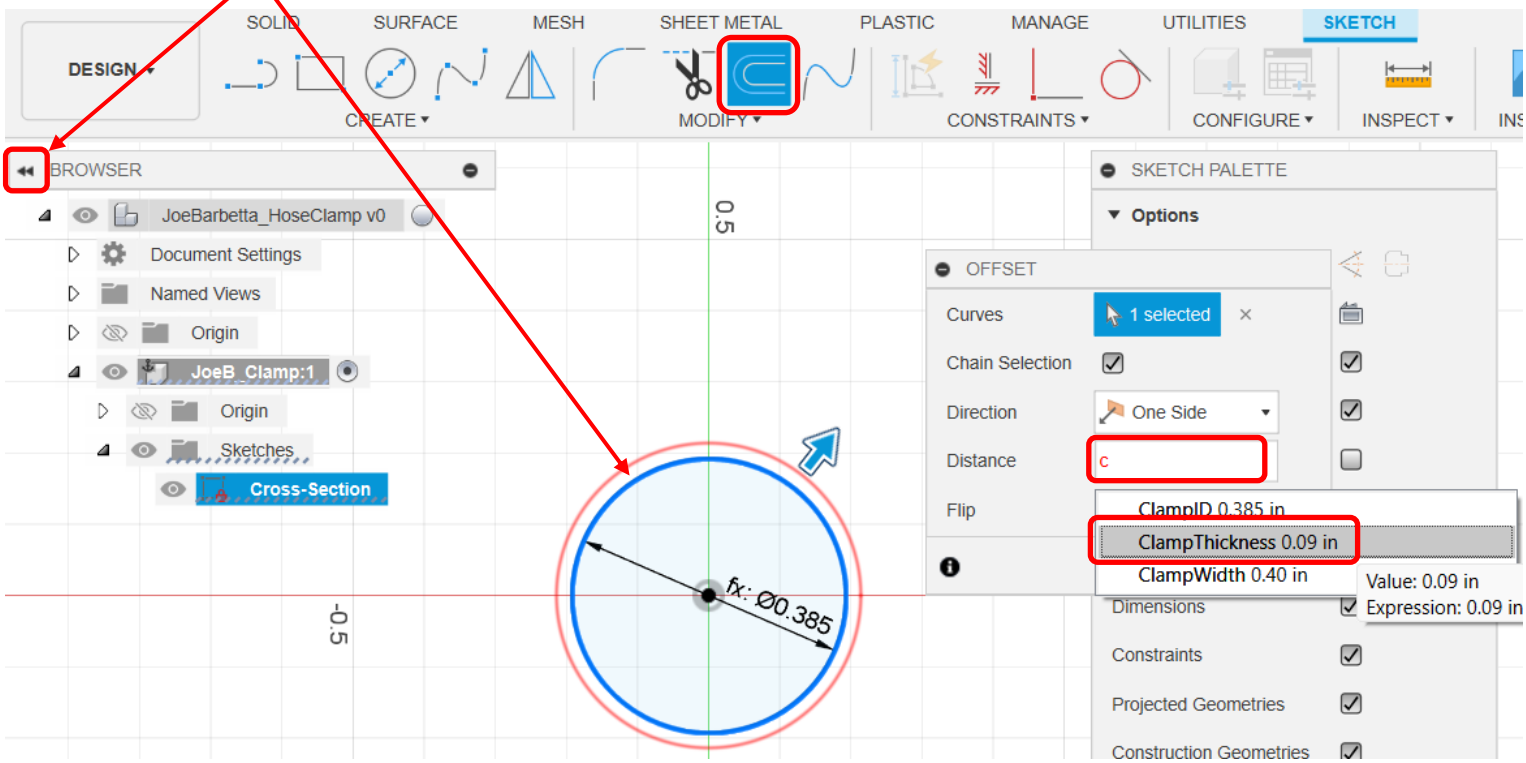
- use the **+** icon again to set 2 more parameters: **ClampThickness 0.09** and **ClampWidth 0.4**
- click **OK**



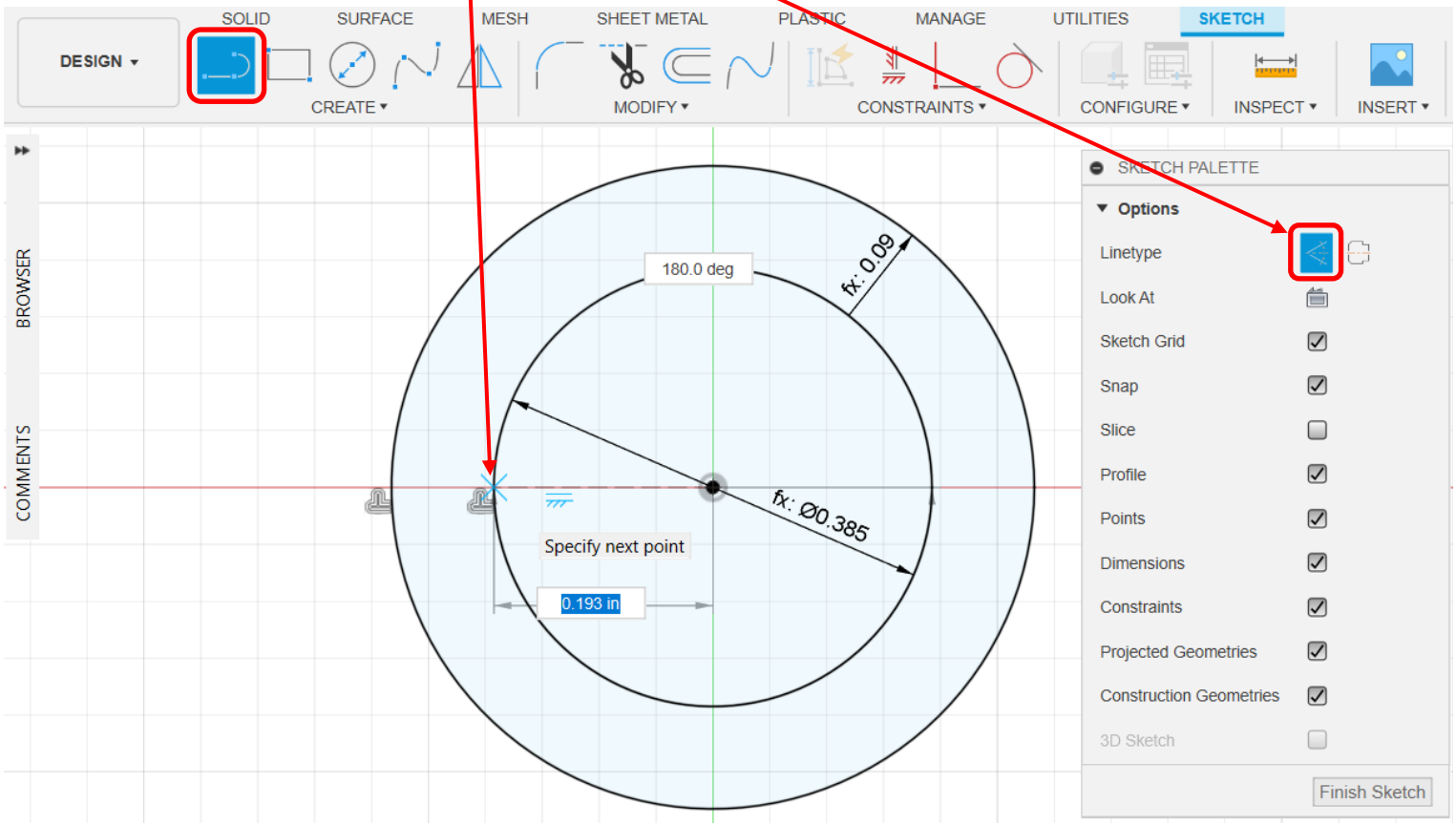
- select the **Center Diameter Circle** tool. If it is not visible, find it in the **CREATE** menu.
- click on the **Origin**, extend the circle outward, type **c**, select **ClampID**, and press the **Enter Key**



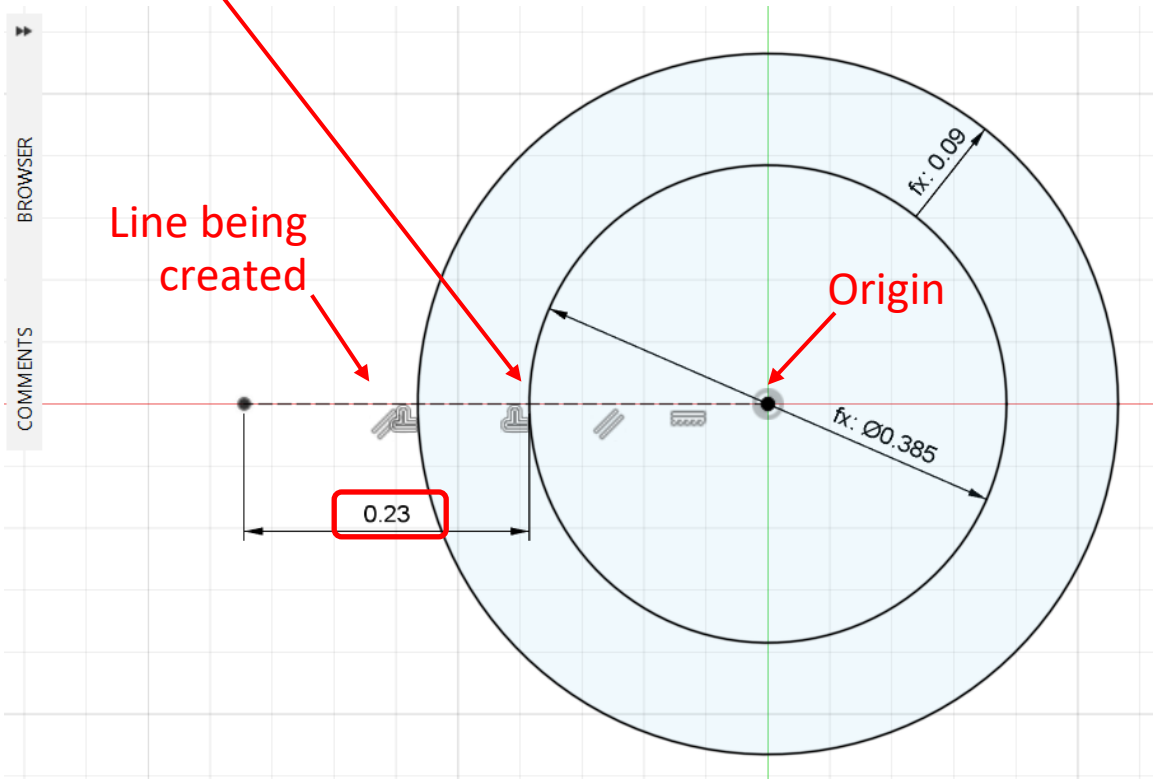
- select the **Offset tool**. If it is not visible, find it in the **MODIFY** menu.
- click on the **circle just created** and extend the red circle outward
- in the **Distance** box type **c**, select **ClampThickness**, and click **OK**
- click on the **double arrow** for the **BROWSER** to hide it



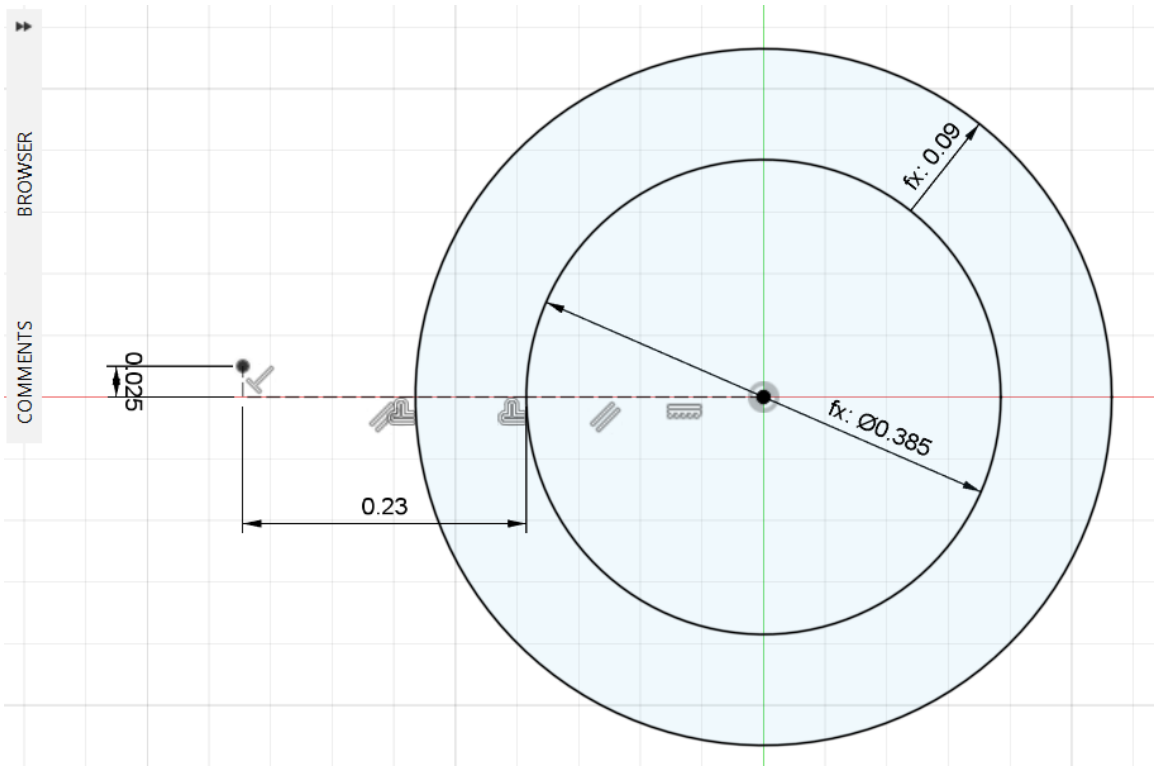
- click on the **Construction icon** to highlight it blue
- select the **Line** tool, click on the **Origin**, and **extend the line to the left**
- when it reaches the inner circle, a **blue x** will show. Click on that **point**. No value needs to be entered.



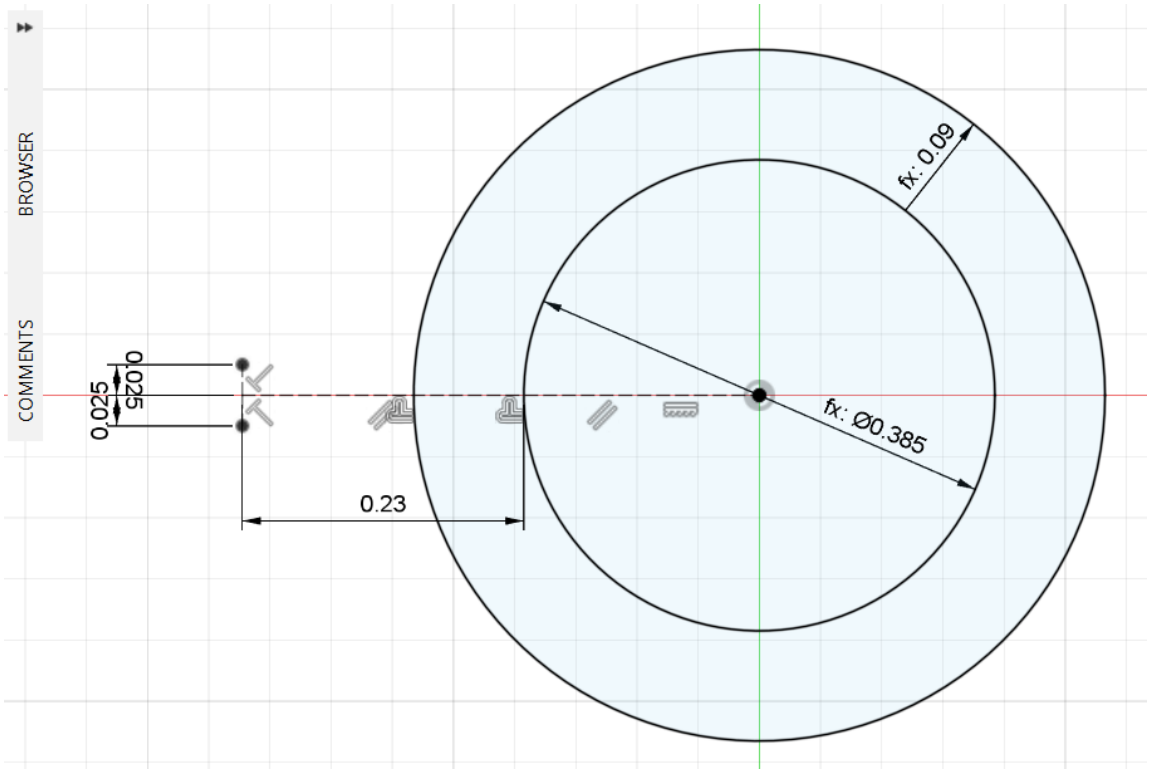
- select the **Line** tool again
- click on the **end of the line** just created and **extend the line to the left**
- type **0.23** and press the **Enter key**



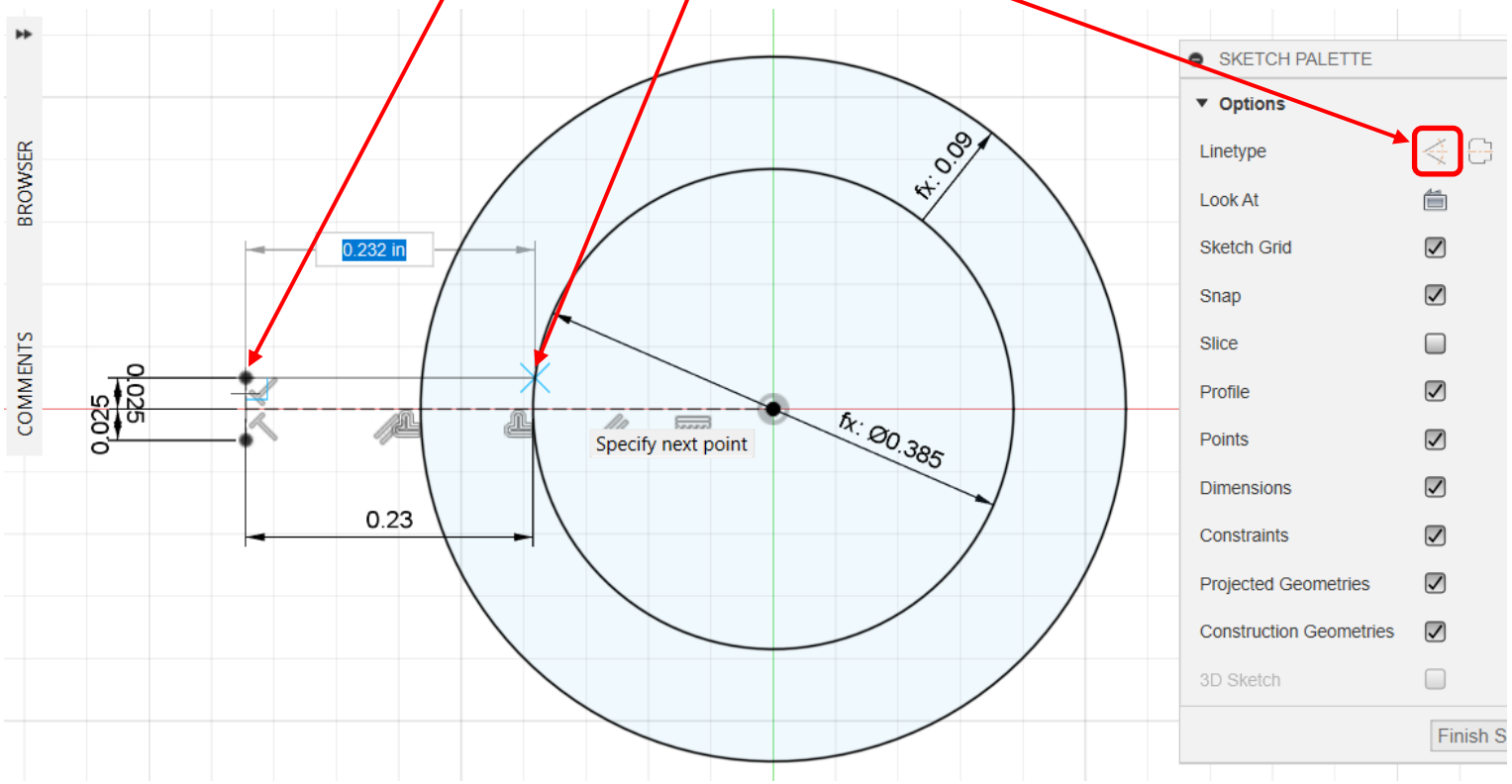
- use the **Line** tool again to start a line at the **end of the horizontal line**
- extend the line upward and enter a value of **0.025**



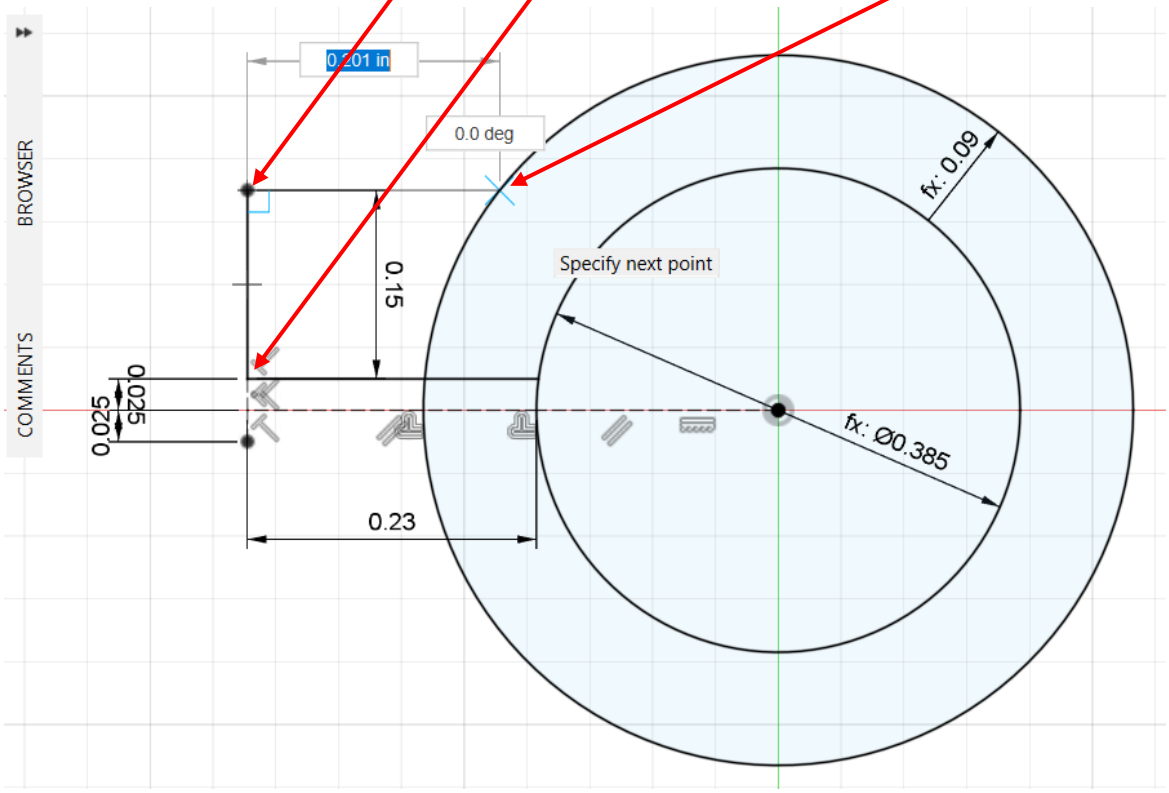
- use the **Line** tool again to start a line at the **end of the horizontal line**
- extend this line downward and enter a value of **0.025**



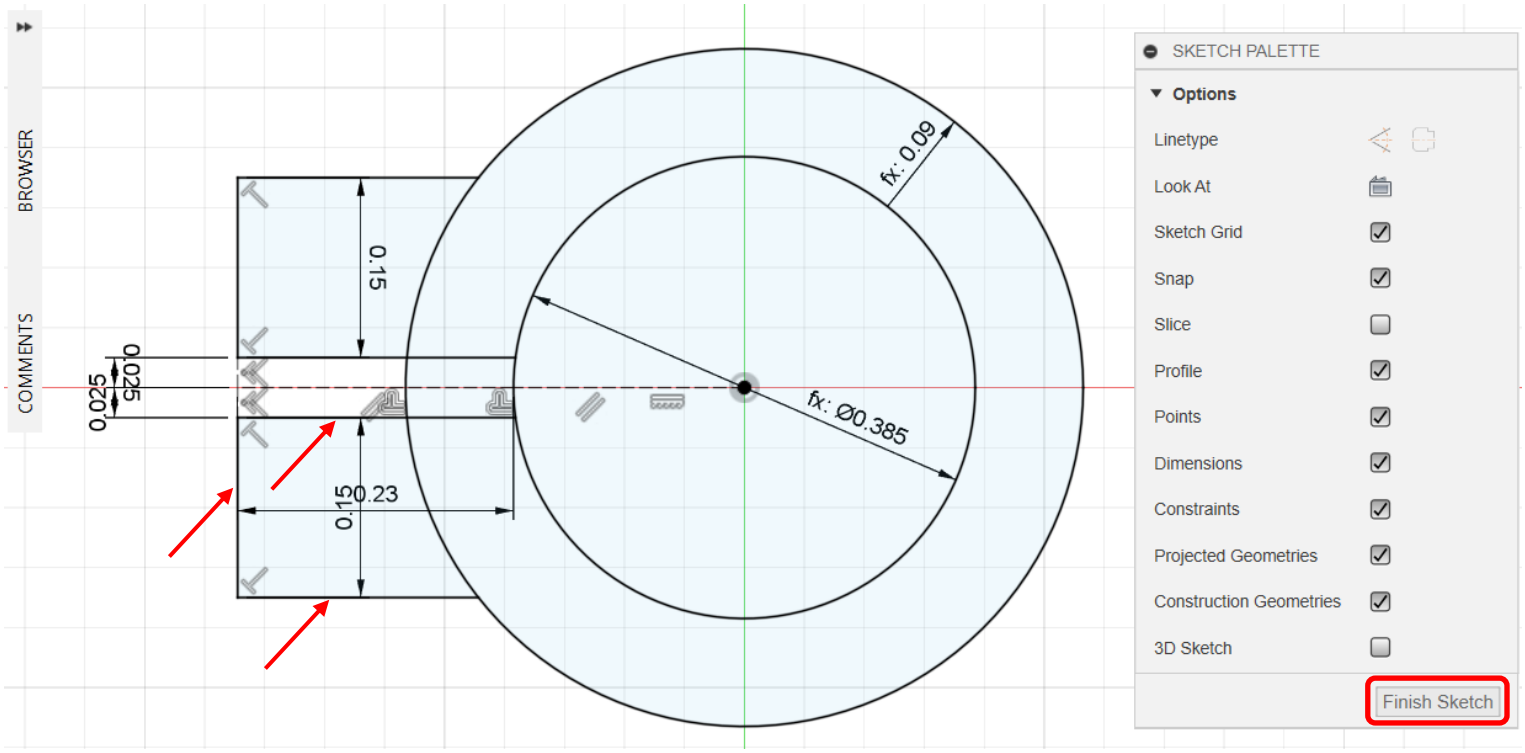
- click on the **Construction icon** to remove the blue highlighting. Lines now drawn should be solid.
- select the **Line** tool and click on the **top of the short vertical line**
- extend the line **to the right** to the inner circle where a **blue x** should appear and **click on that point**. No value will be entered.



- create another line from the **top of the short vertical line**, extend it upward and enter a value of **0.15**
- create another line from that **last point**, extend it to the outer circle, where a **blue x** should appear, and **click at that point**. No value needs to be entered.

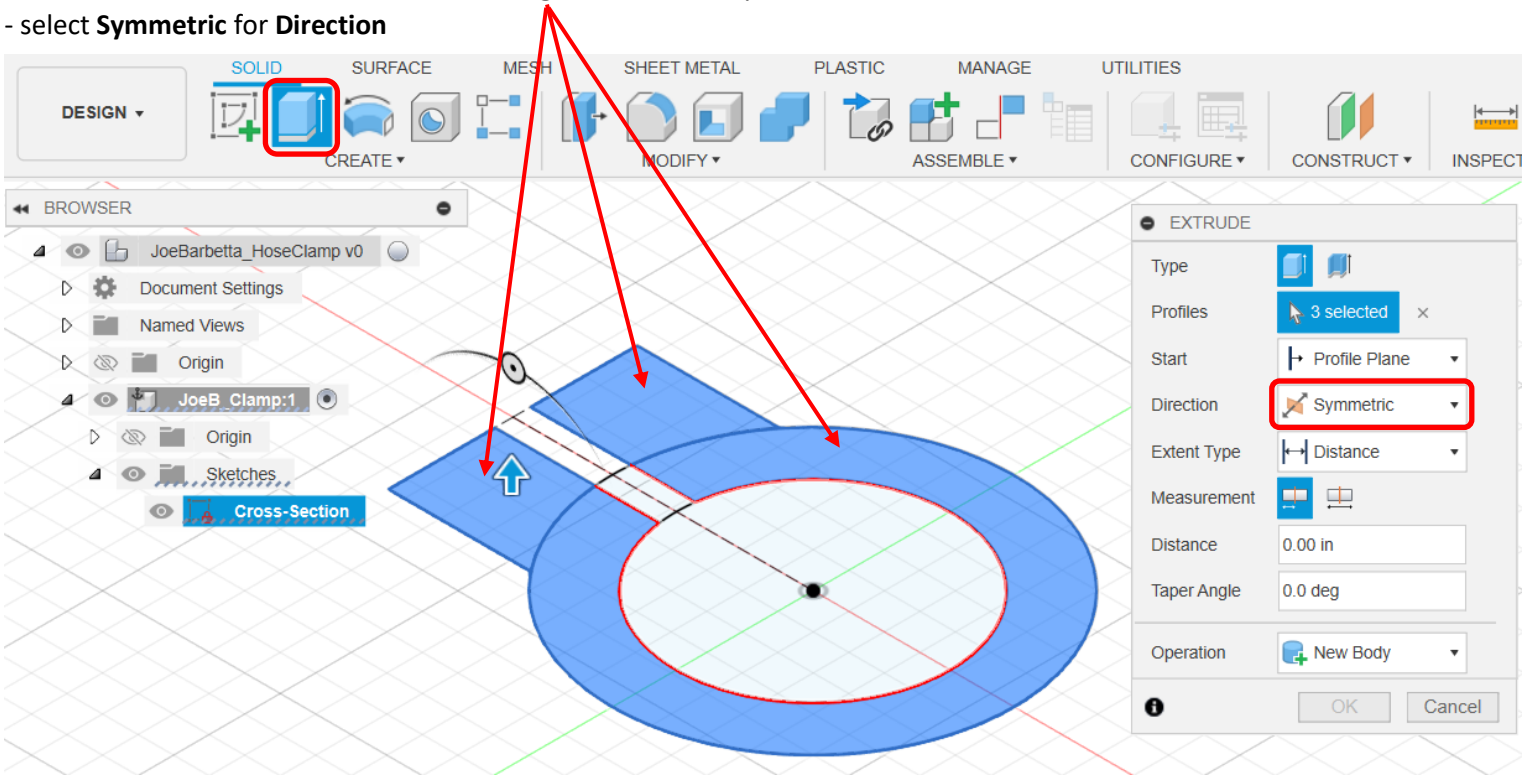


- create the **same 3 lines** in the bottom section
- click **Finish Sketch**

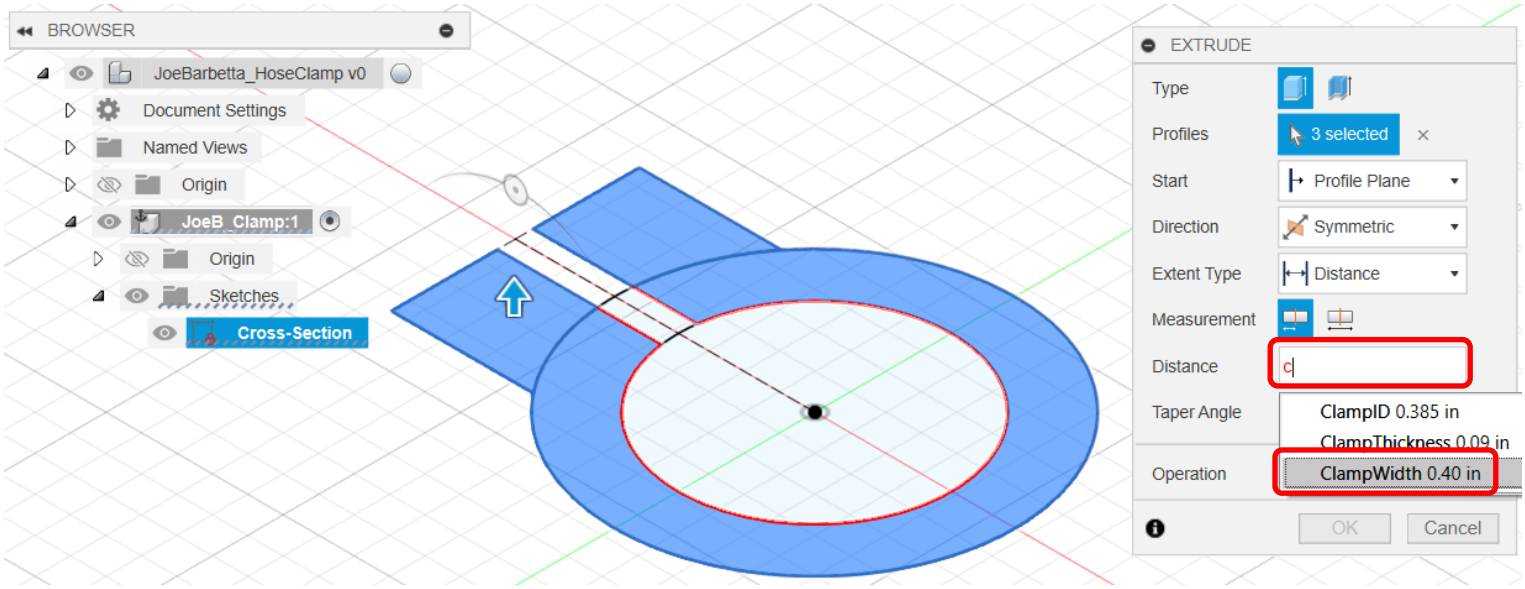


Using a Symmetric Extrude

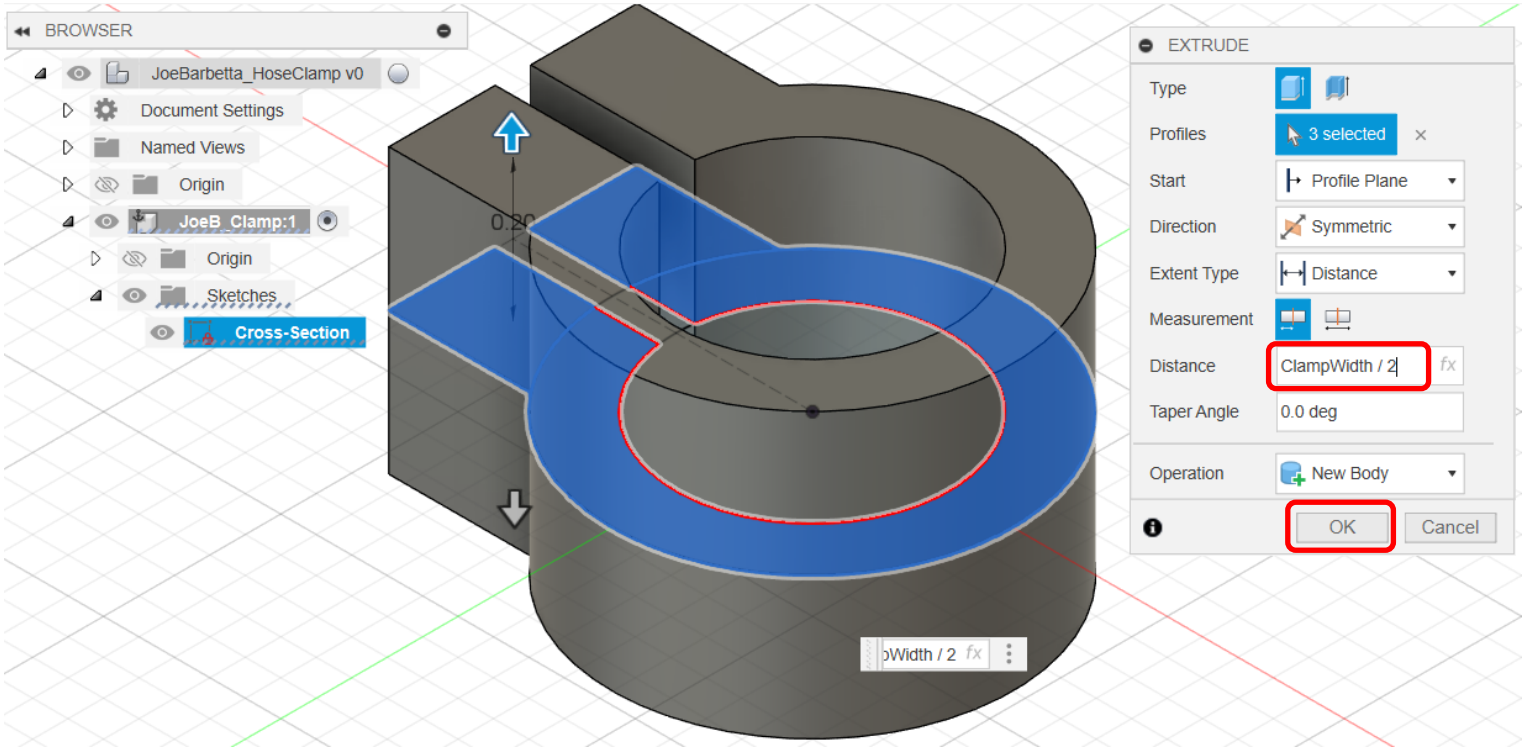
- click on the **Home** icon at the **View Cube**
- click on the **double arrows** to reopen the BROWSER
- select the **Extrude** tool and click on the **3 regions** indicated by the arrows
- select **Symmetric** for **Direction**



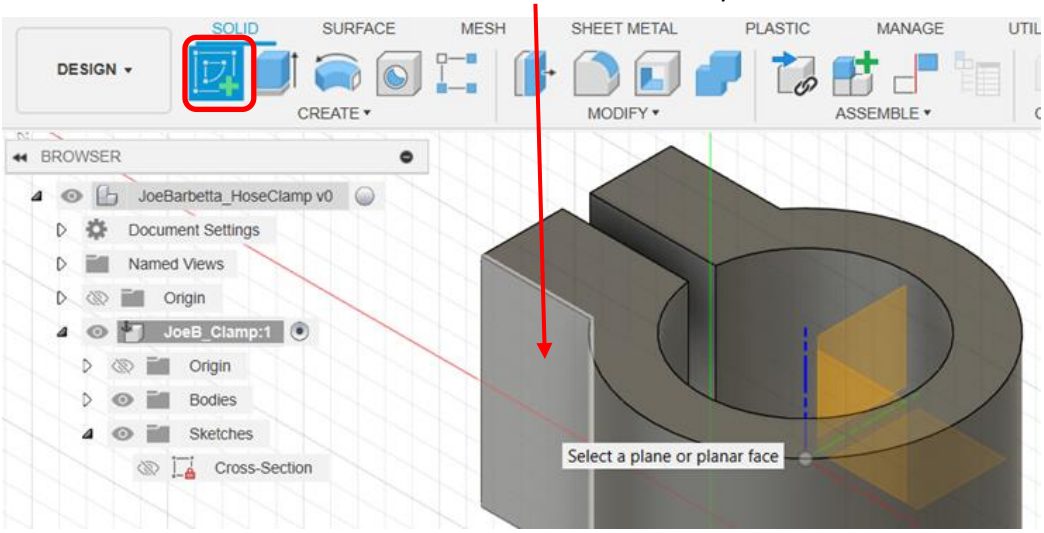
- in the **Distance** box type **c** and select **ClampWidth**



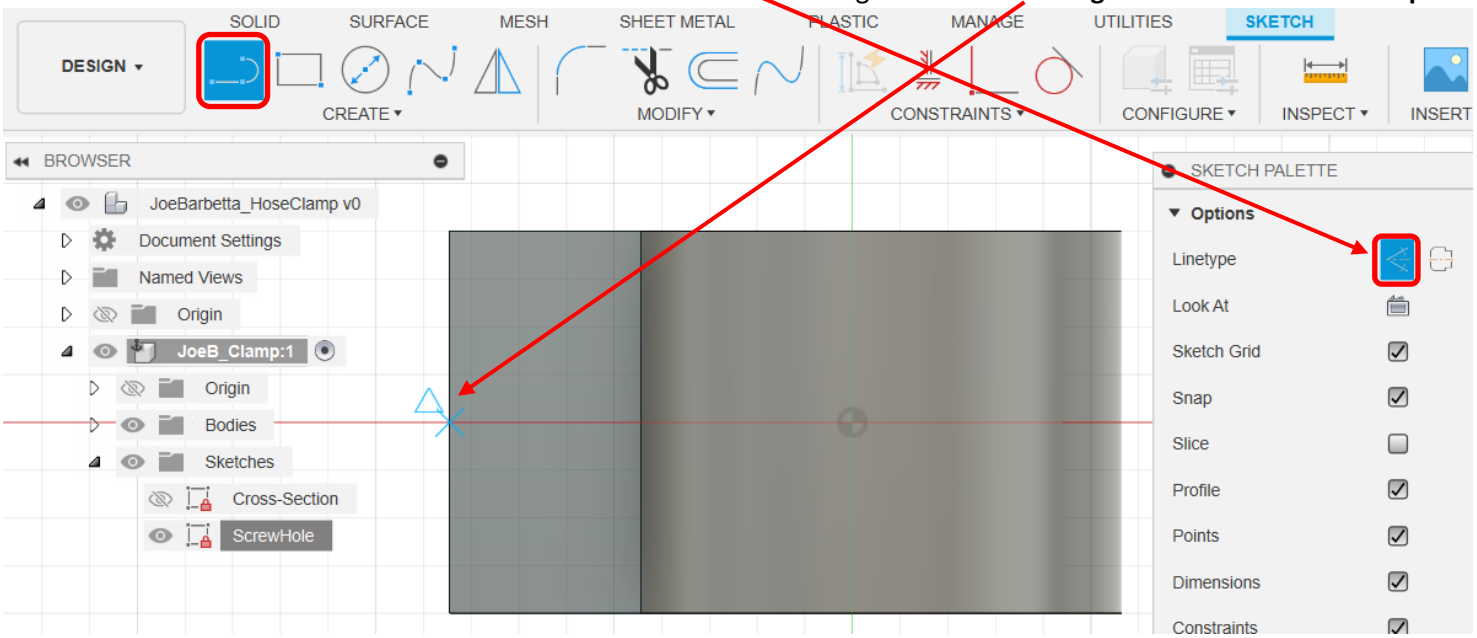
- in the **Distance** box type **/ 2** after **ClampWidth**
- click **OK**



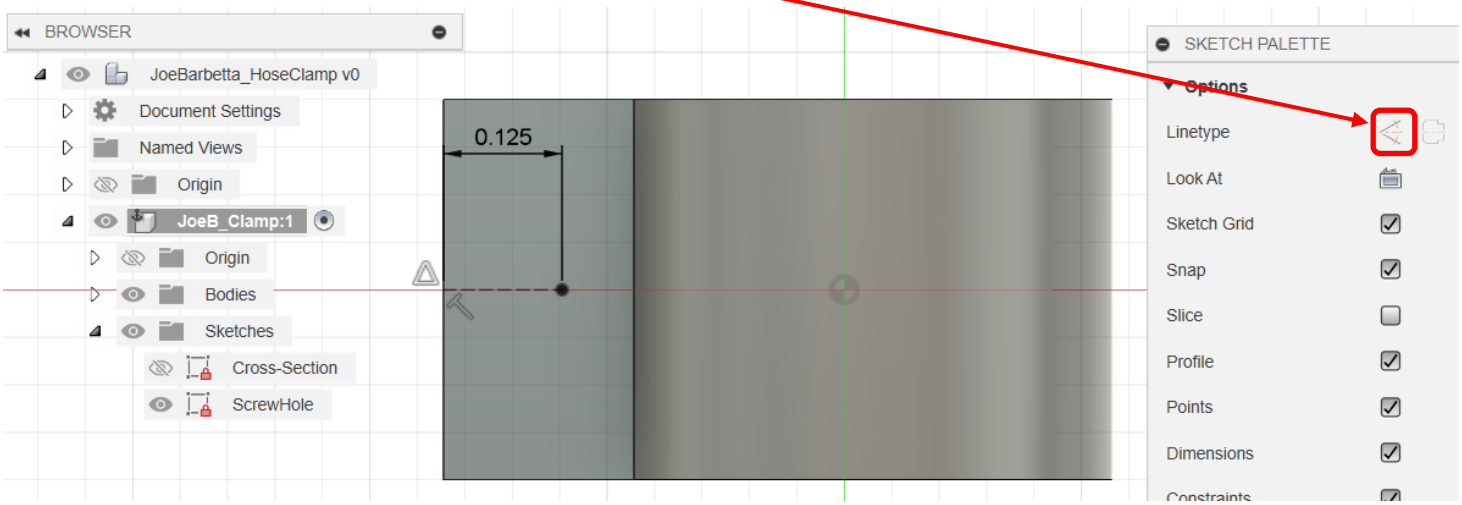
- select the **Extrude** tool and click on the **surface** indicated by the arrow



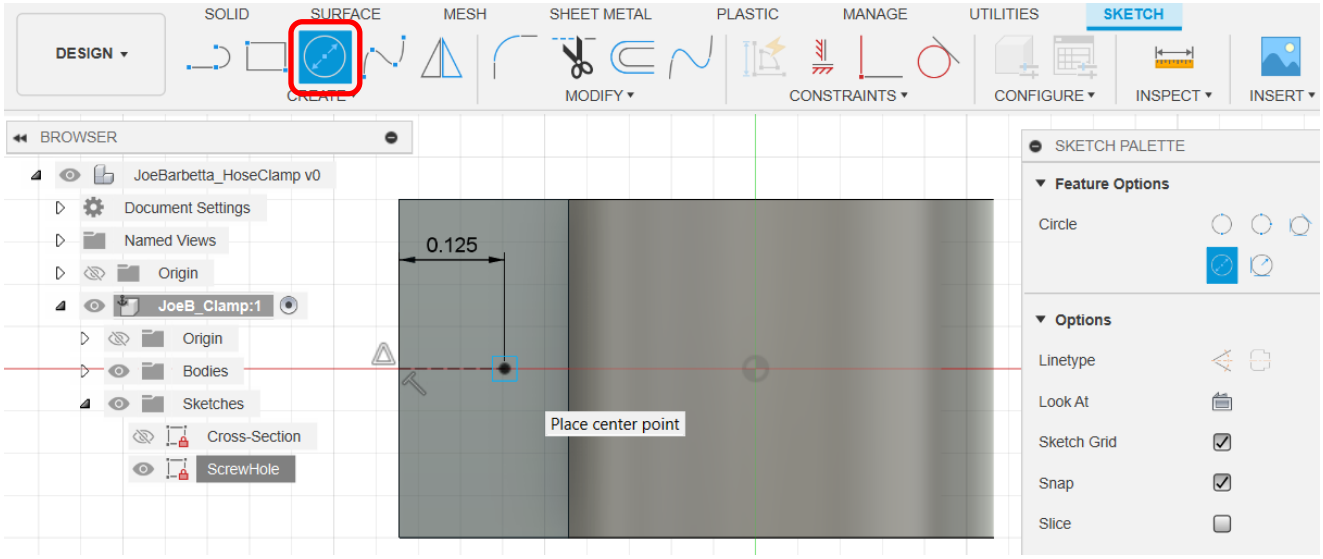
- **rename** the Sketch to **ScrewHole**
- click on the **Construction** line icon to highlight it blue
- select the **Line** tool and move the mouse over the center of the left edge until a **blue triangle** shows and **click at this point**



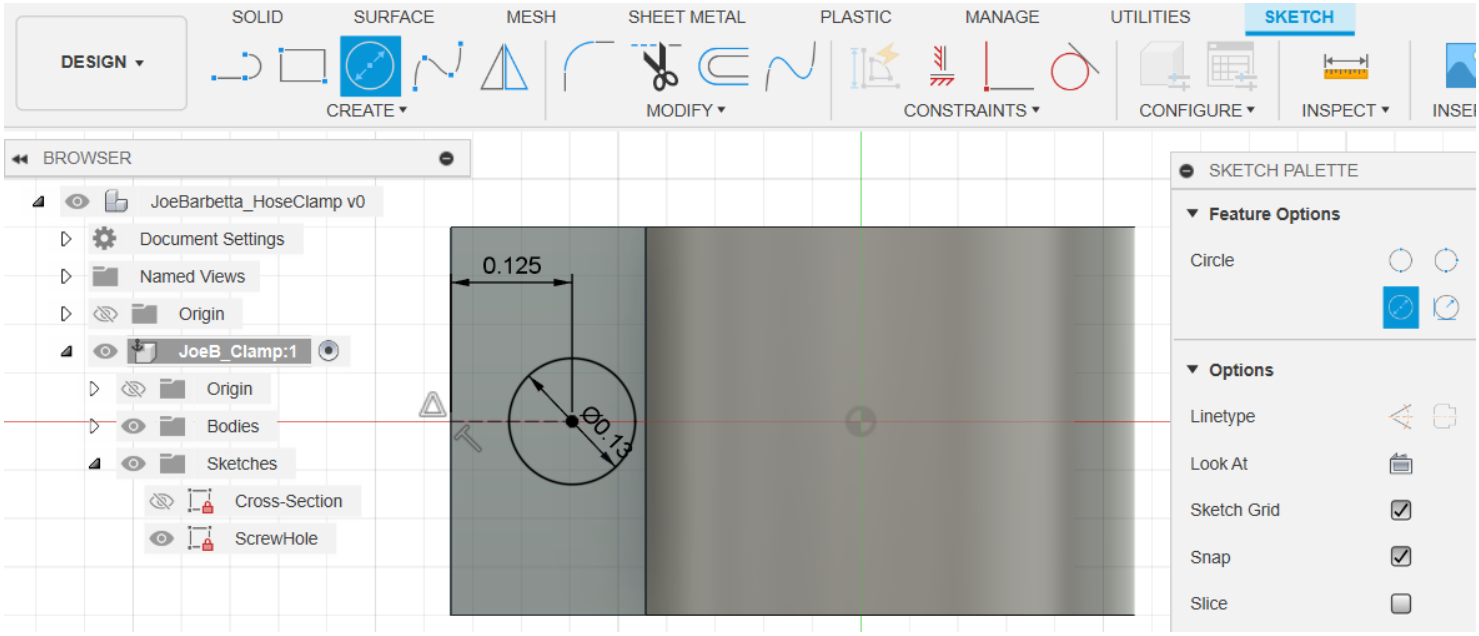
- extend the line **to the right** and enter 0.**125**
- click on the **Construction** icon to remove the highlighting



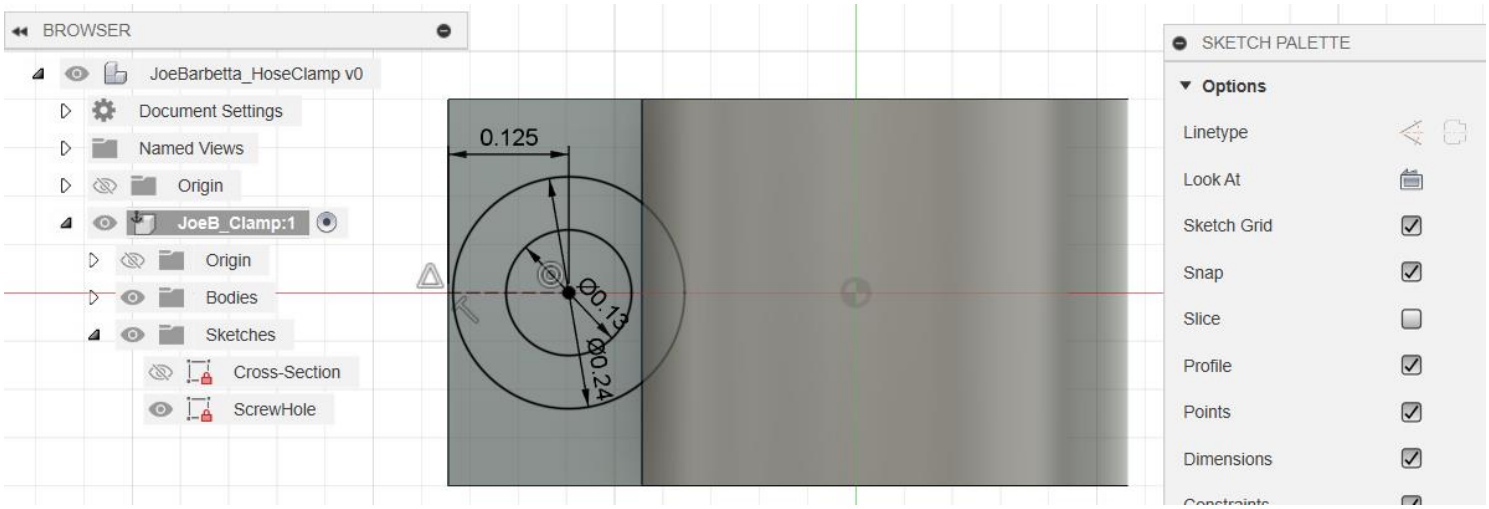
- select the **Center Diameter Circle** tool and click on the **right end** of the line



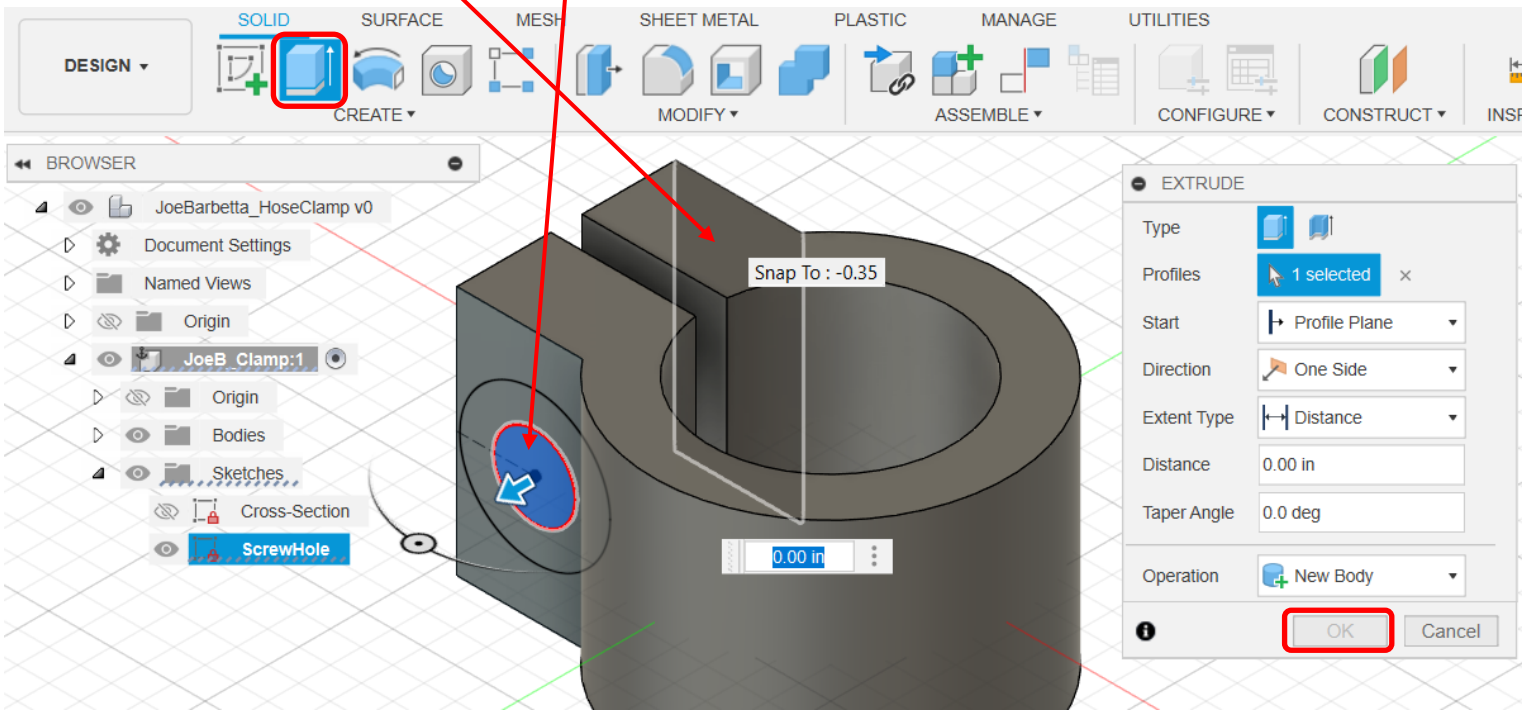
- extend the circle outward and enter a value of **0.13**



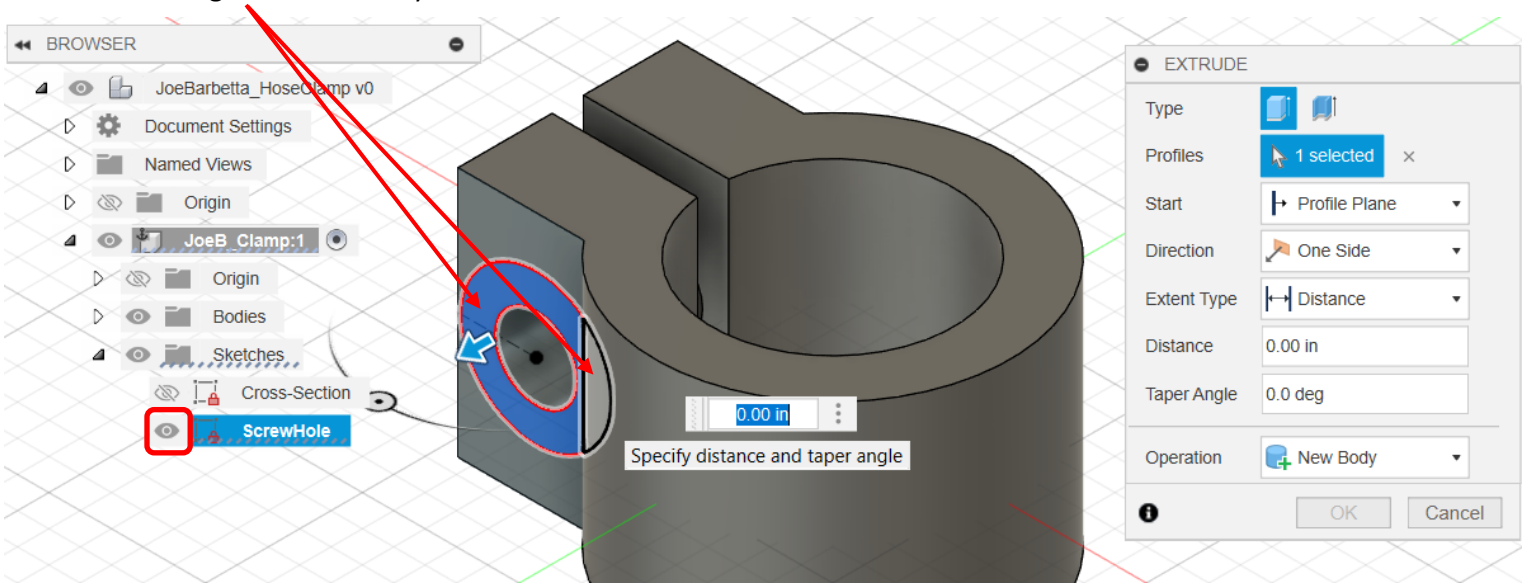
- use the **Center Diameter Circle** tool again to create a circle with a diameter of **0.24**
- click **Finish Sketch**



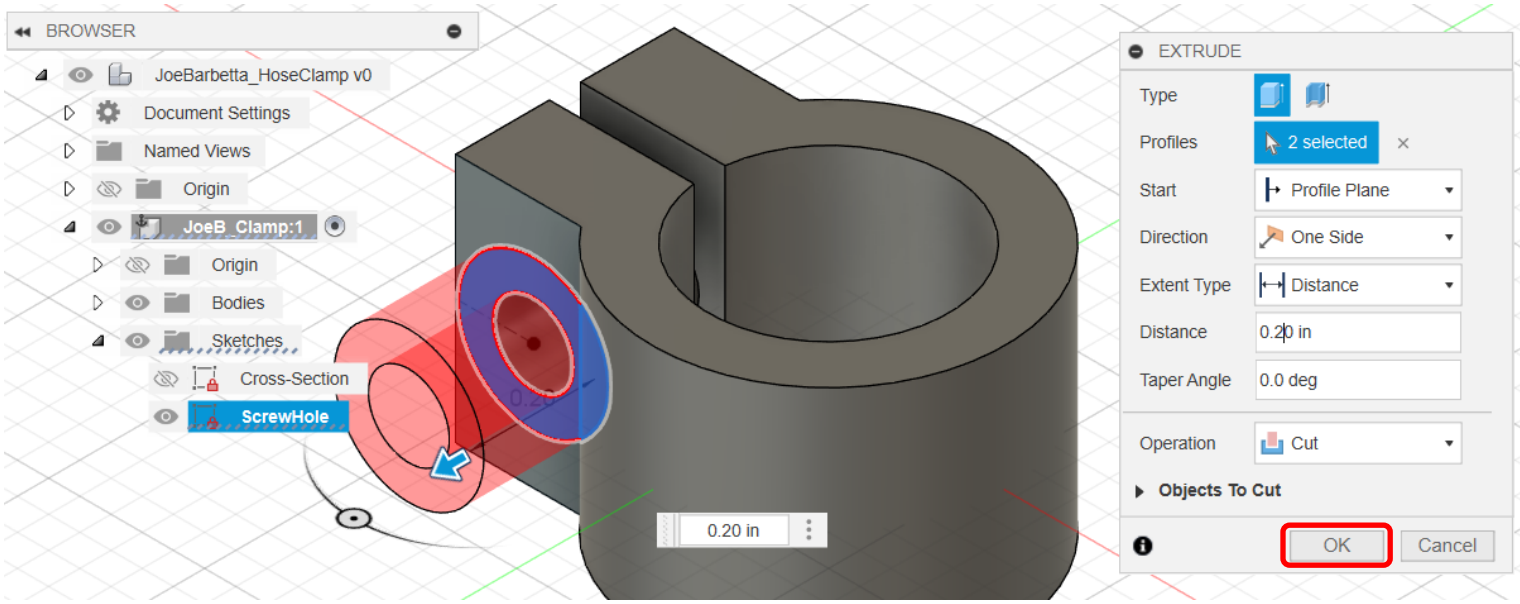
- click on the **Home** icon at the **View Cube**
- select the **Extrude** tool and click on the **center circle** to highlight it blue
- click in the area indicated by the arrow to select the hidden rear face and click **OK**



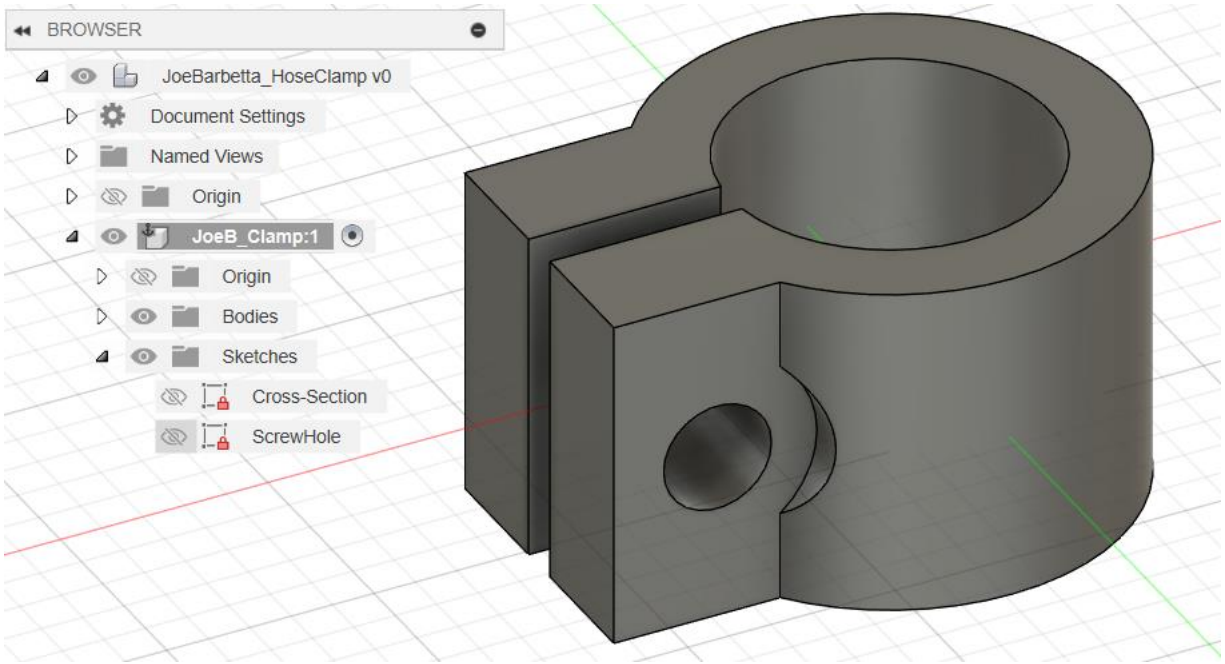
- click on the **eye** icon for the **ScrewHole** Sketch to make it visible again
- click on the **2 regions** indicated by the arrows



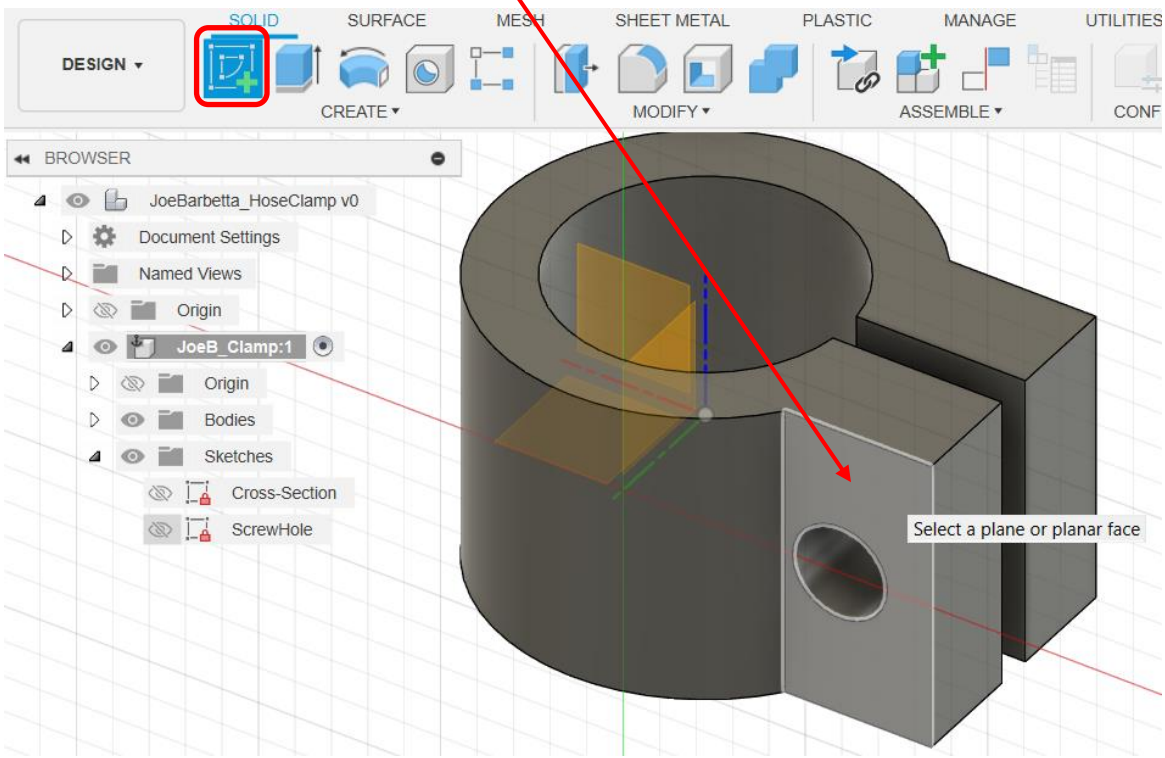
- enter **0.20** for **Distance** and click **OK**



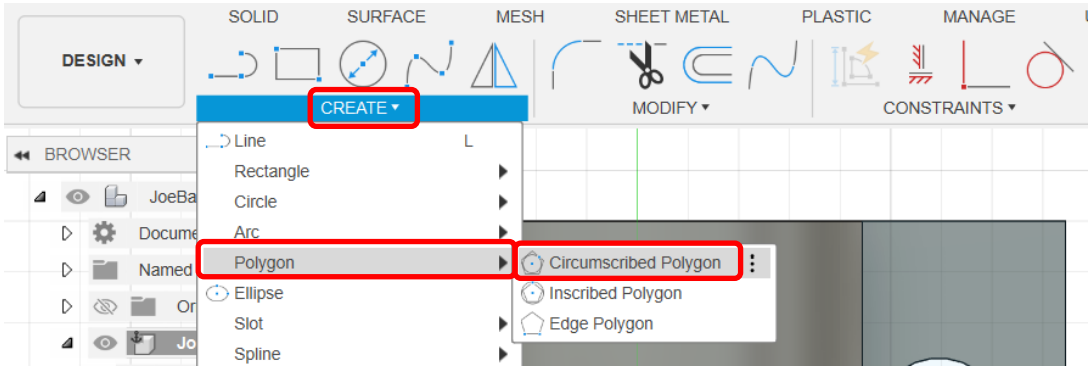
- use the **View Cube** to adjust the view to see the circular cutout, which will allow the screw head to sit on the surface



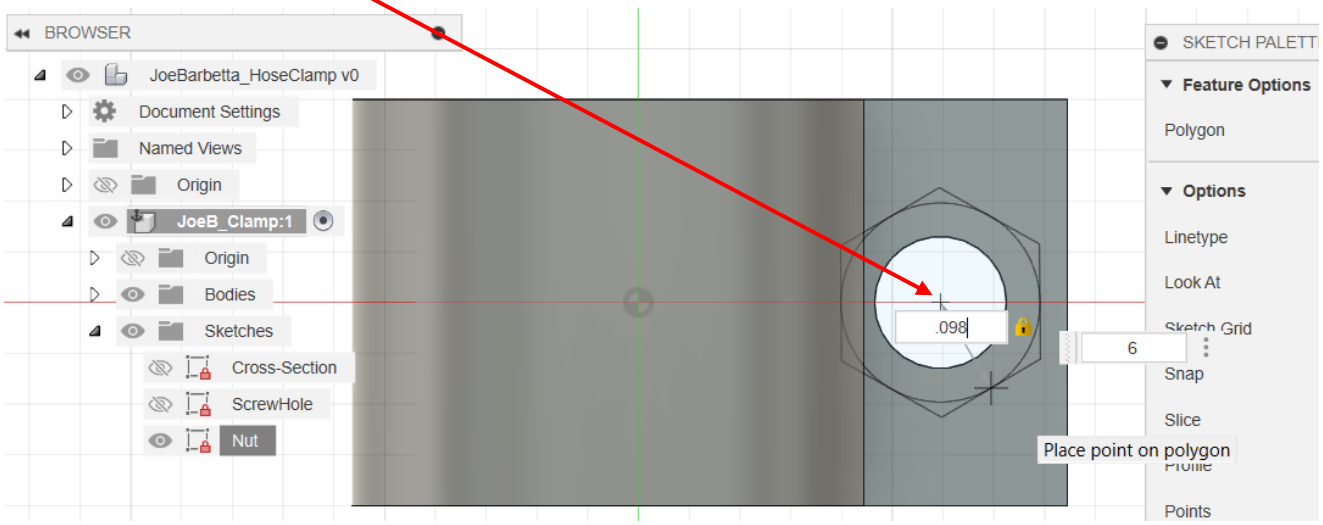
- use the **View Cube** to adjust the view to access the other end of the hole
- select **Create Sketch** and click on the surface indicated by the arrow. Rename the Sketch to **Nut**.



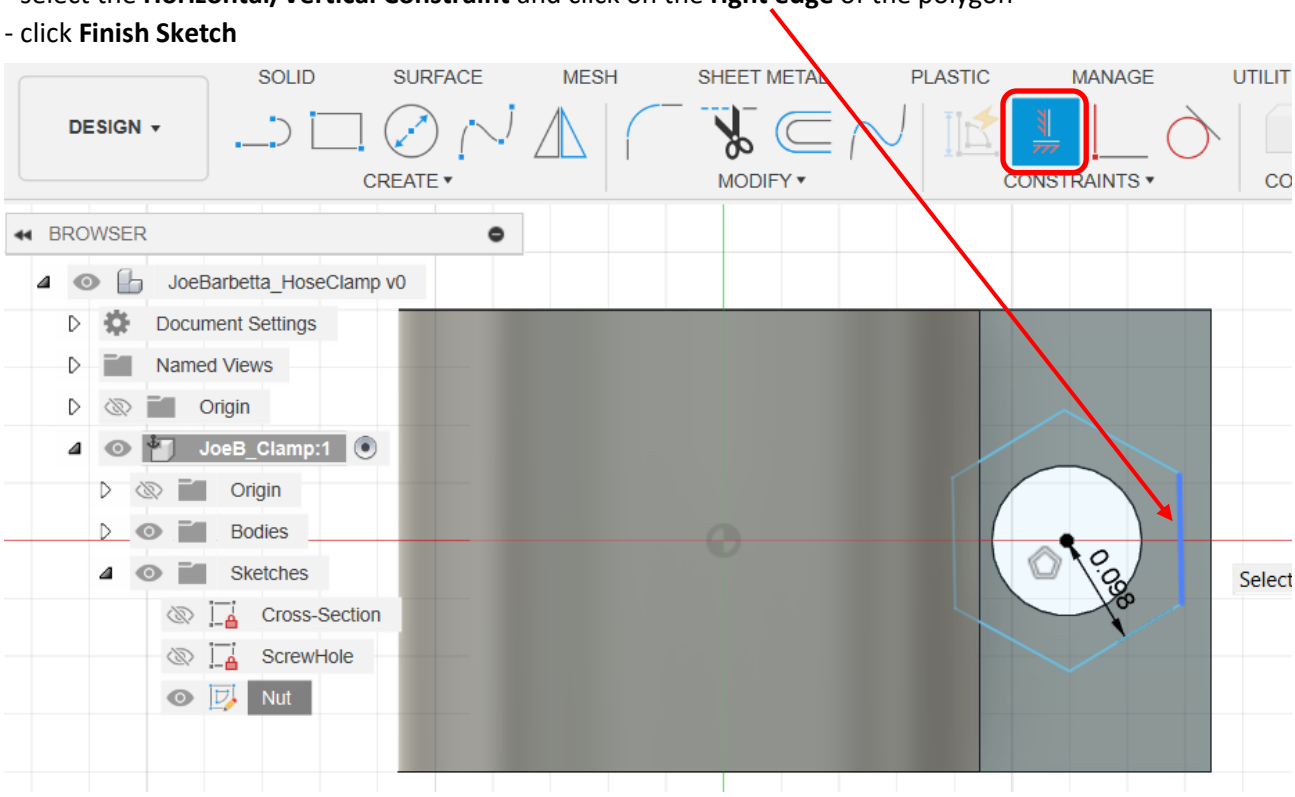
- from the **CREATE** menu select **Polygon** and **Circumscribed Polygon**



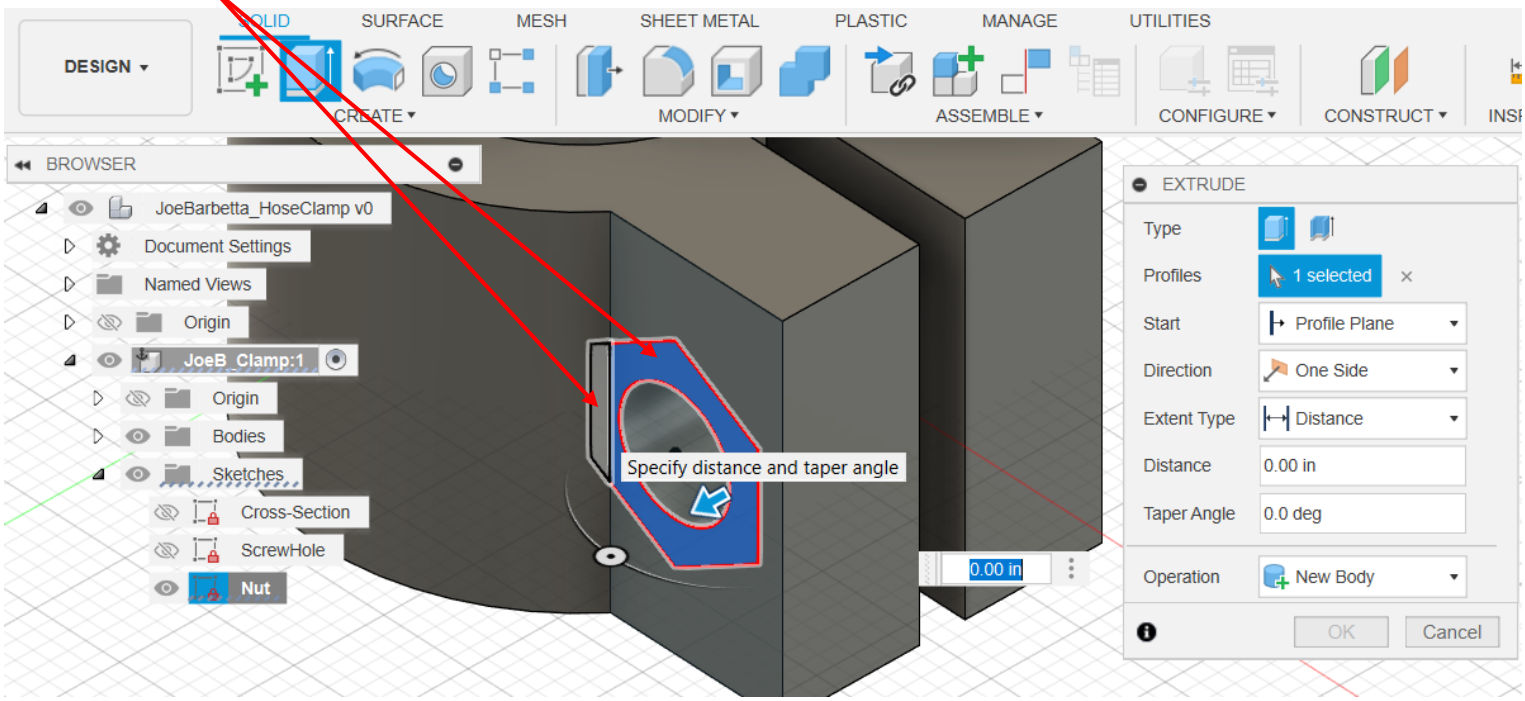
- click on the **center of the hole**, **extend the polygon outward**, and enter a value of **0.098**



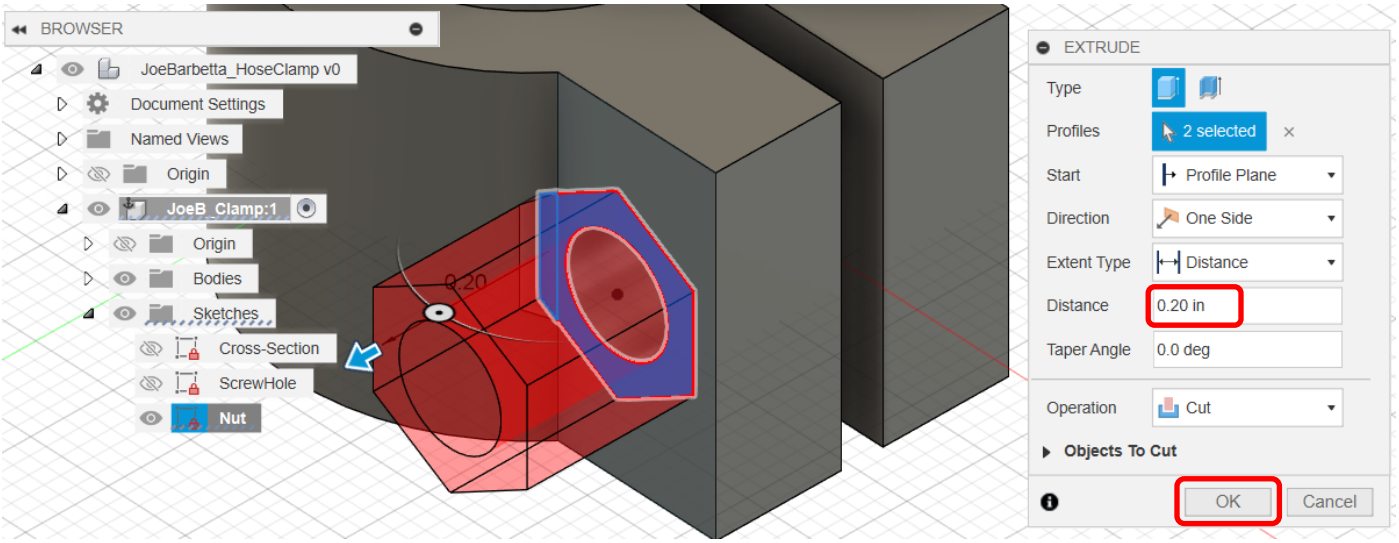
- select the **Horizontal/Vertical Constraint** and click on the **right edge** of the polygon
- click **Finish Sketch**



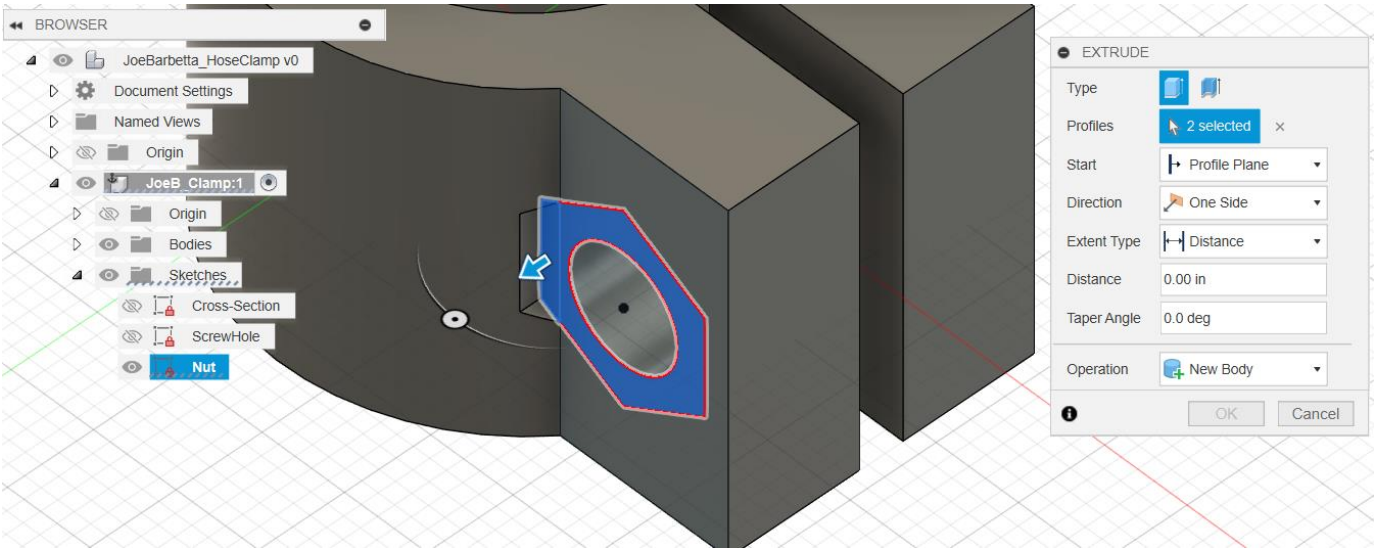
- click the **Home** icon at the **View Cube** and zoom into the Sketch just created
- select the **Extrude** tool
- click on the **2 regions** indicated by the arrows to highlight them blue



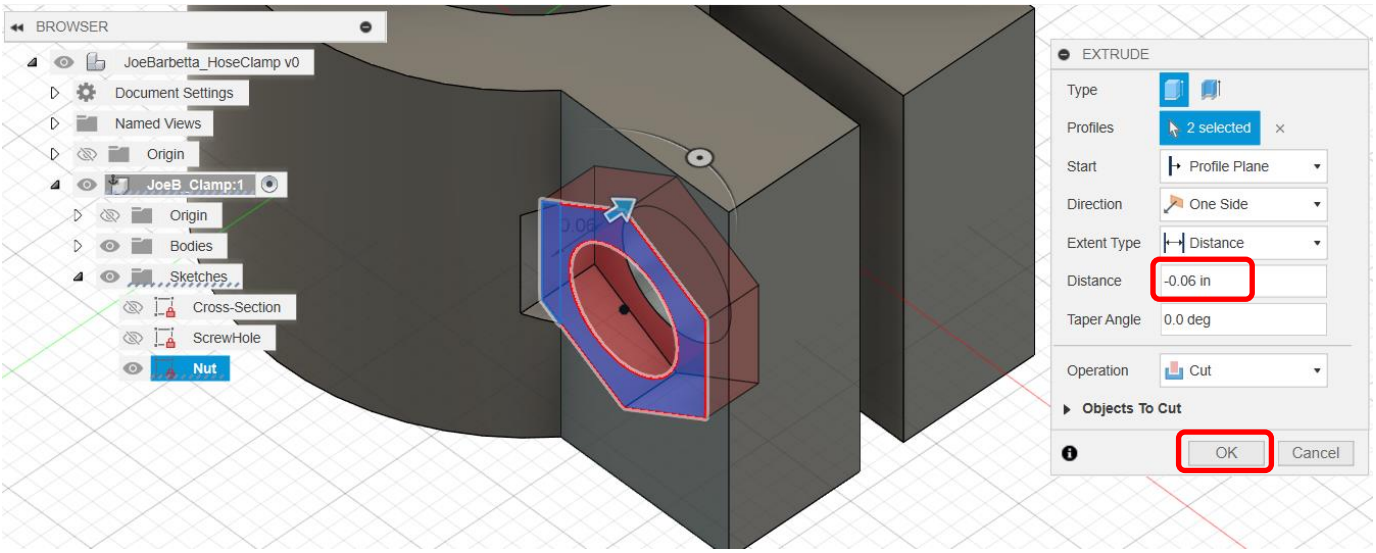
- enter a value of **0.20** for **Distance** and click **OK**



- click on the **eye** icon for the **Nut** Sketch to make it visible again
- select the **Extrude** tool again and select the **2 regions** again

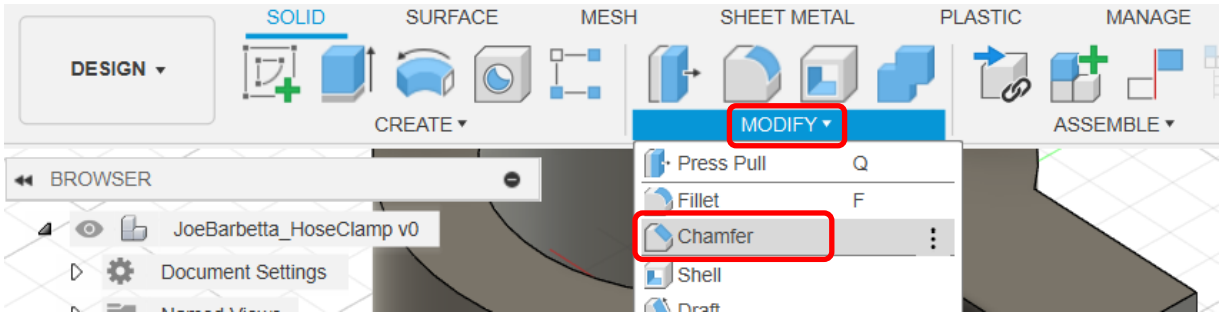


- enter a value of **-0.06** (note the minus sign) for **Distance** and click **OK**

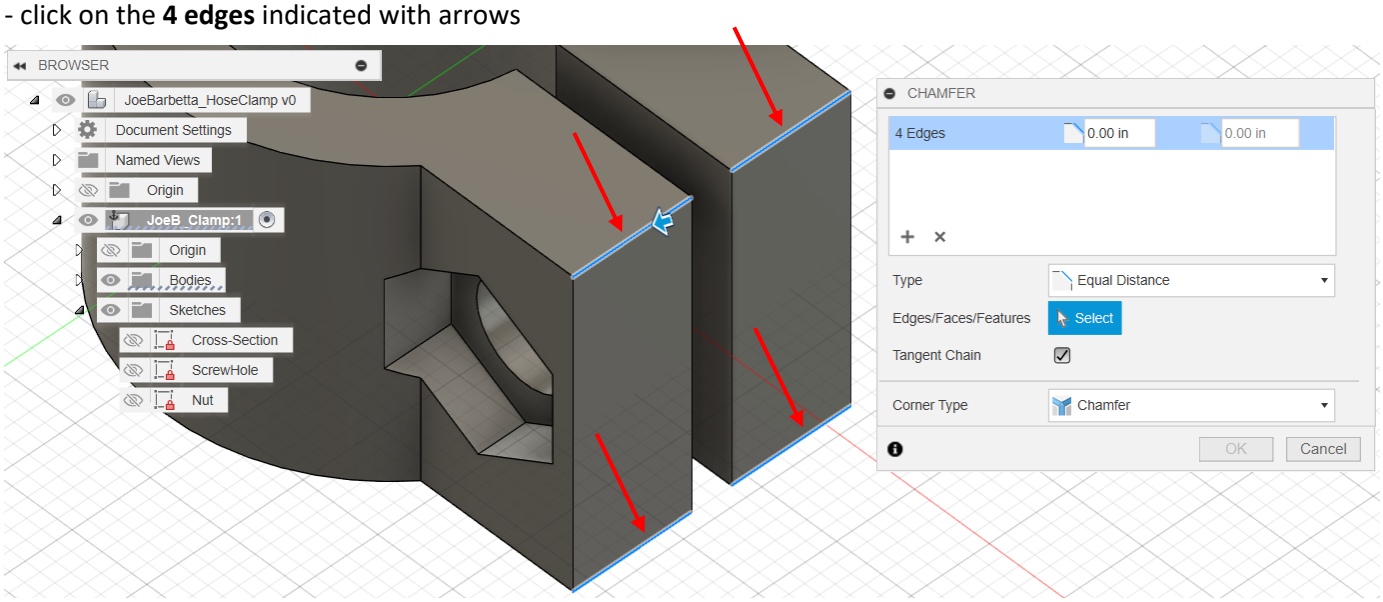


Adding Chamfers

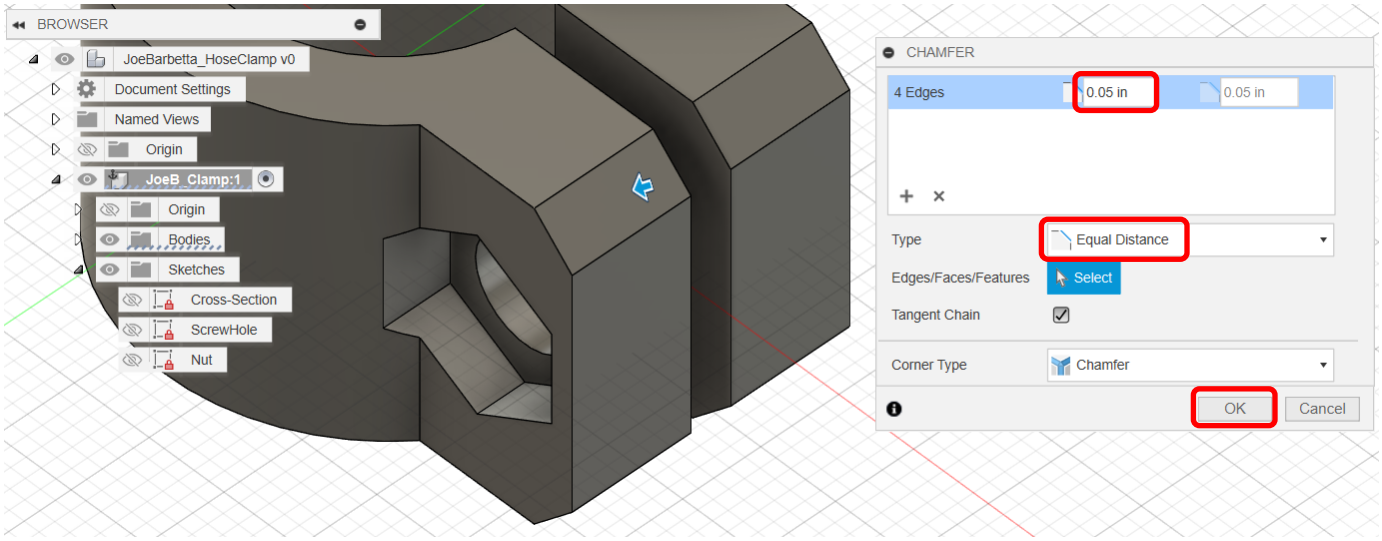
- from the **MODIFY** menu select **Chamfer**



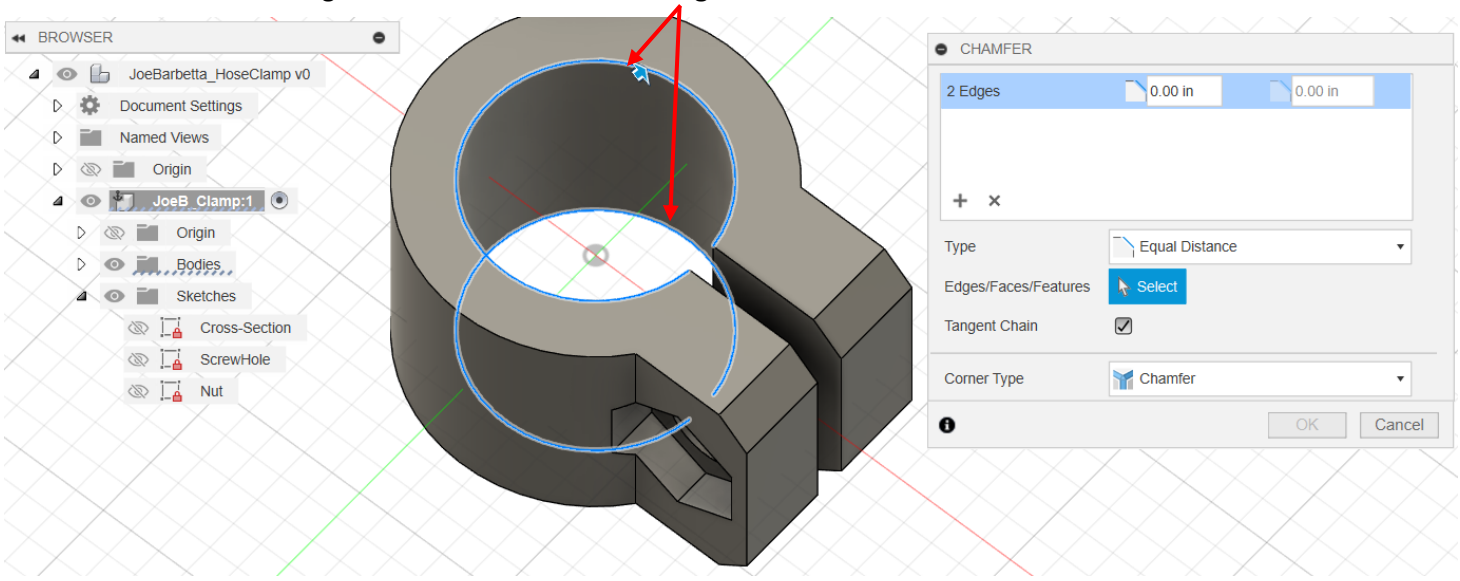
- click on the **4 edges** indicated with arrows



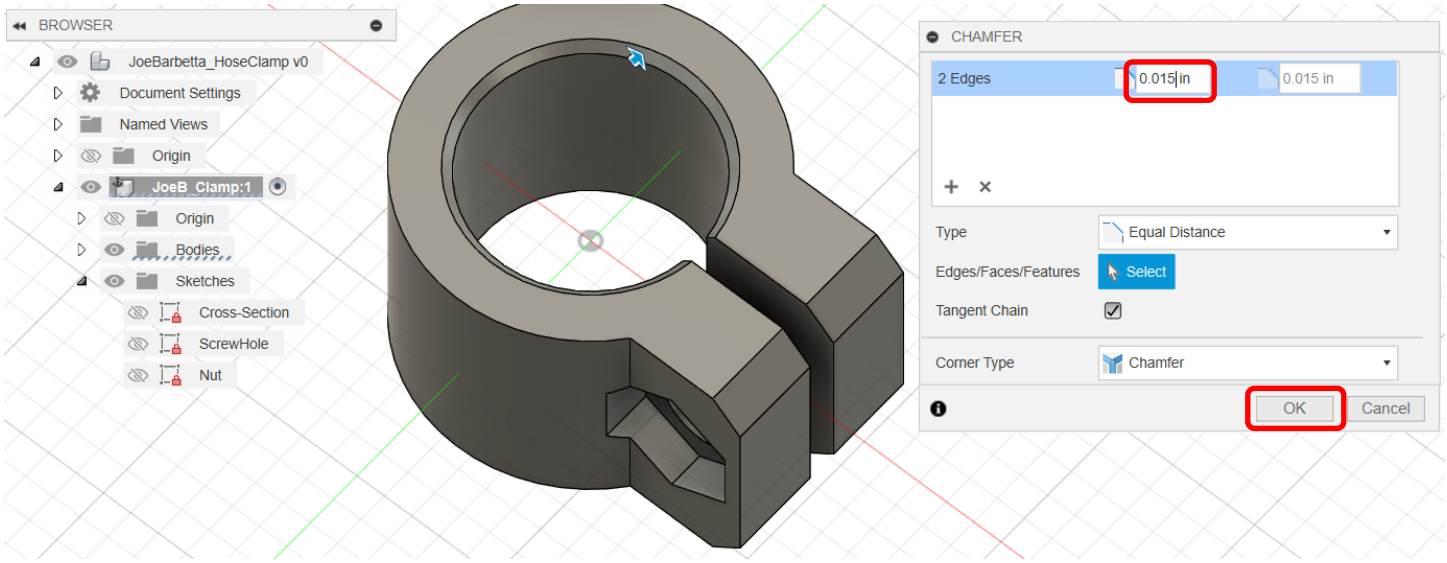
- ensure that **Type** is set to **Equal Distance**
- enter a value of **0.05** and click **OK**



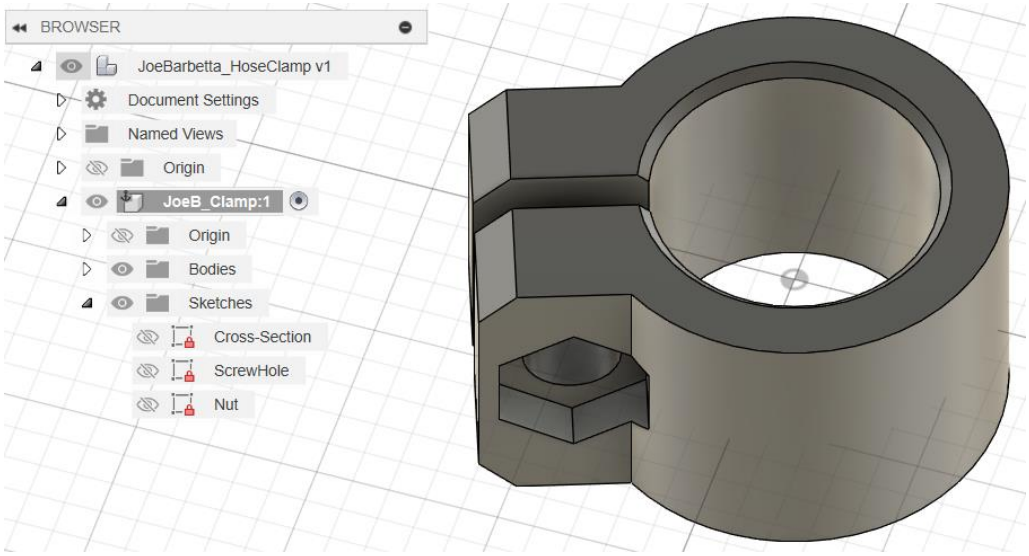
- select the **Chamfer** tool again and click on the **2 inner edges** as shown



- enter a value of **0.015** and click **OK**

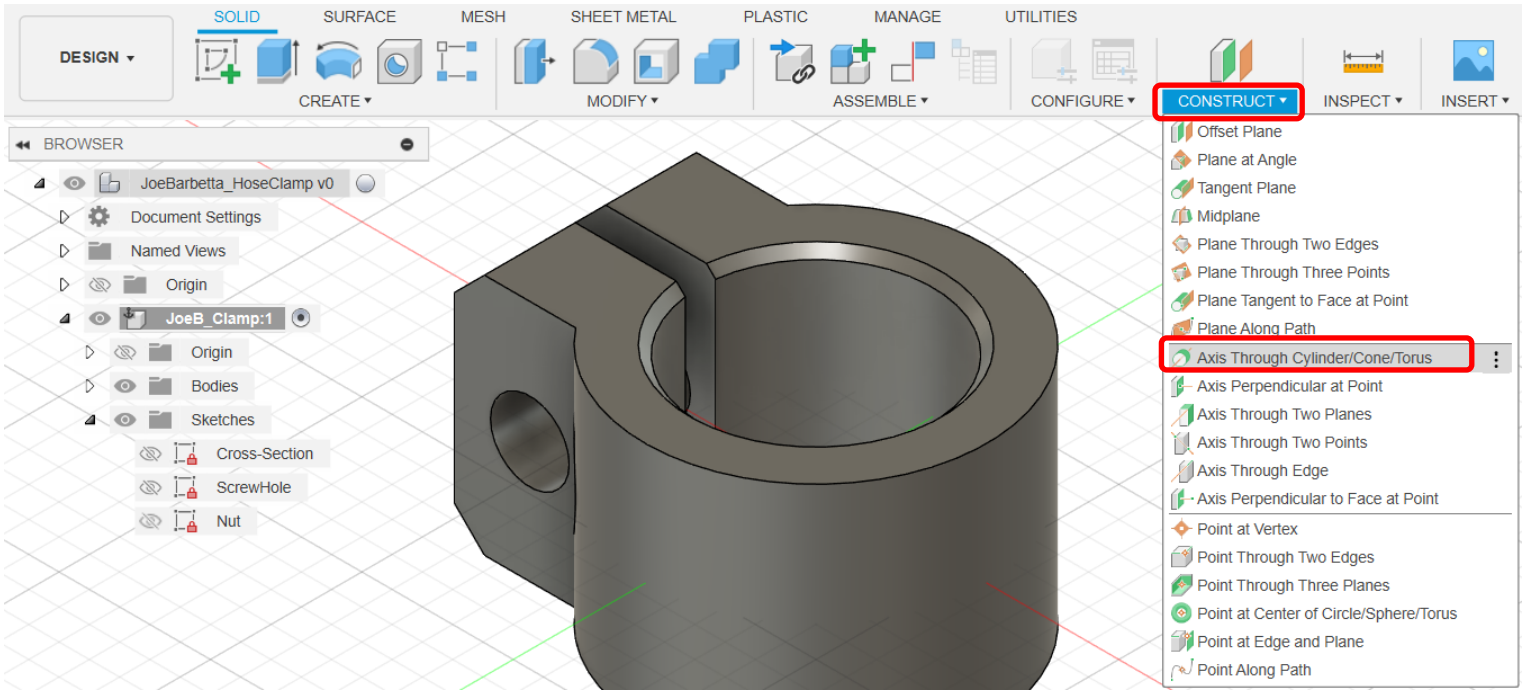


- use the **View Cube** to rotate the view to verify that the underside of the hole has a chamfer

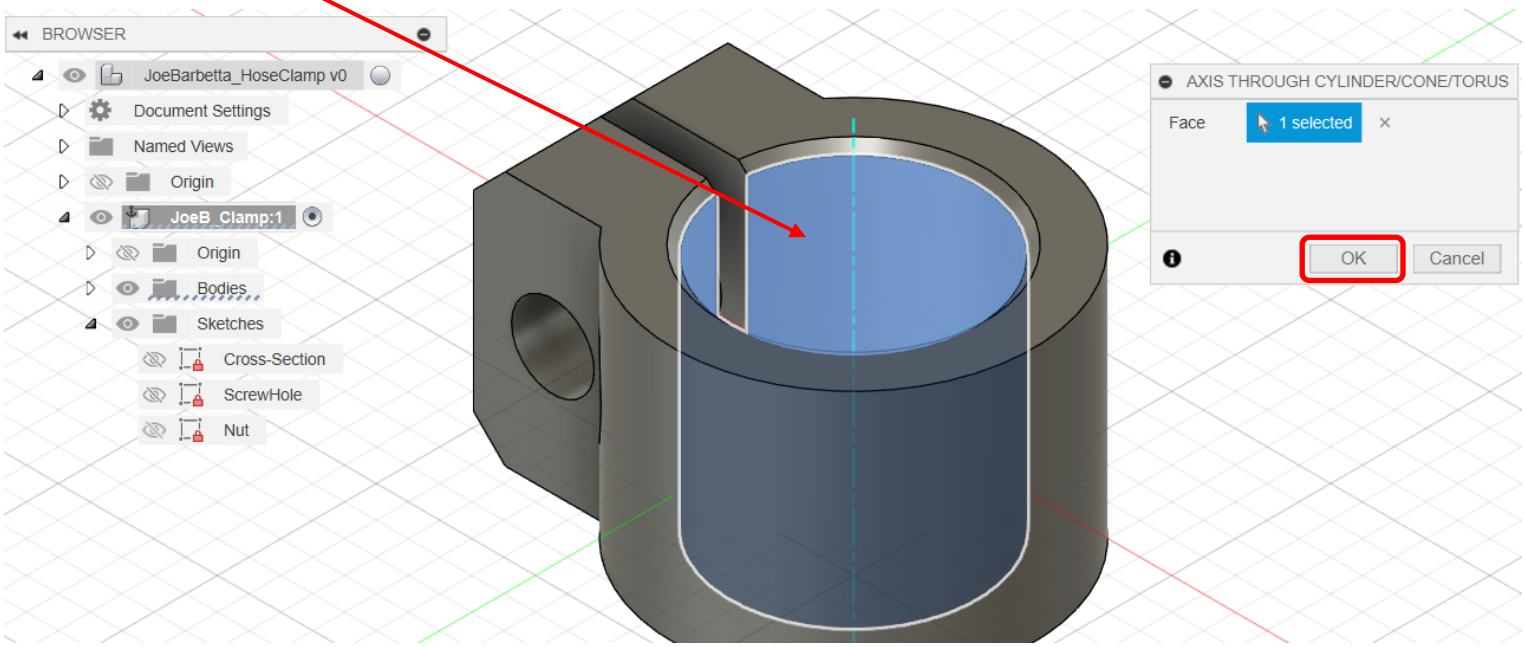


Adding Text

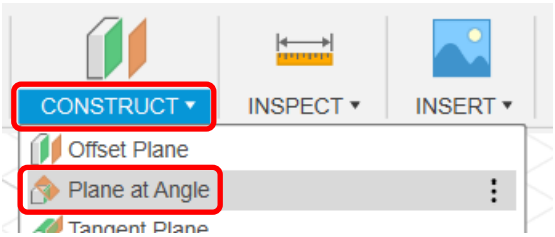
- from the **CONSTRUCT** menu select **Axis Through Cylinder/Cone/Torus**



- click on the **inner surface** of the clamp. Note the aqua colored Center Line
- click **OK**

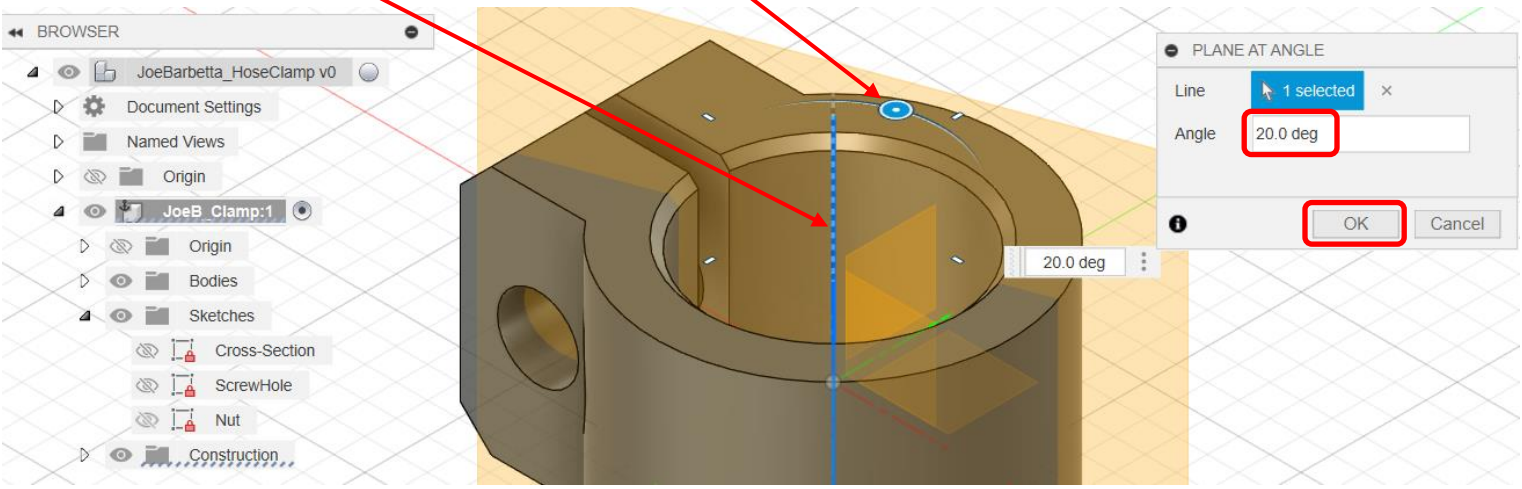


- from the **CONSTRUCT** menu select **Plane at Angle**

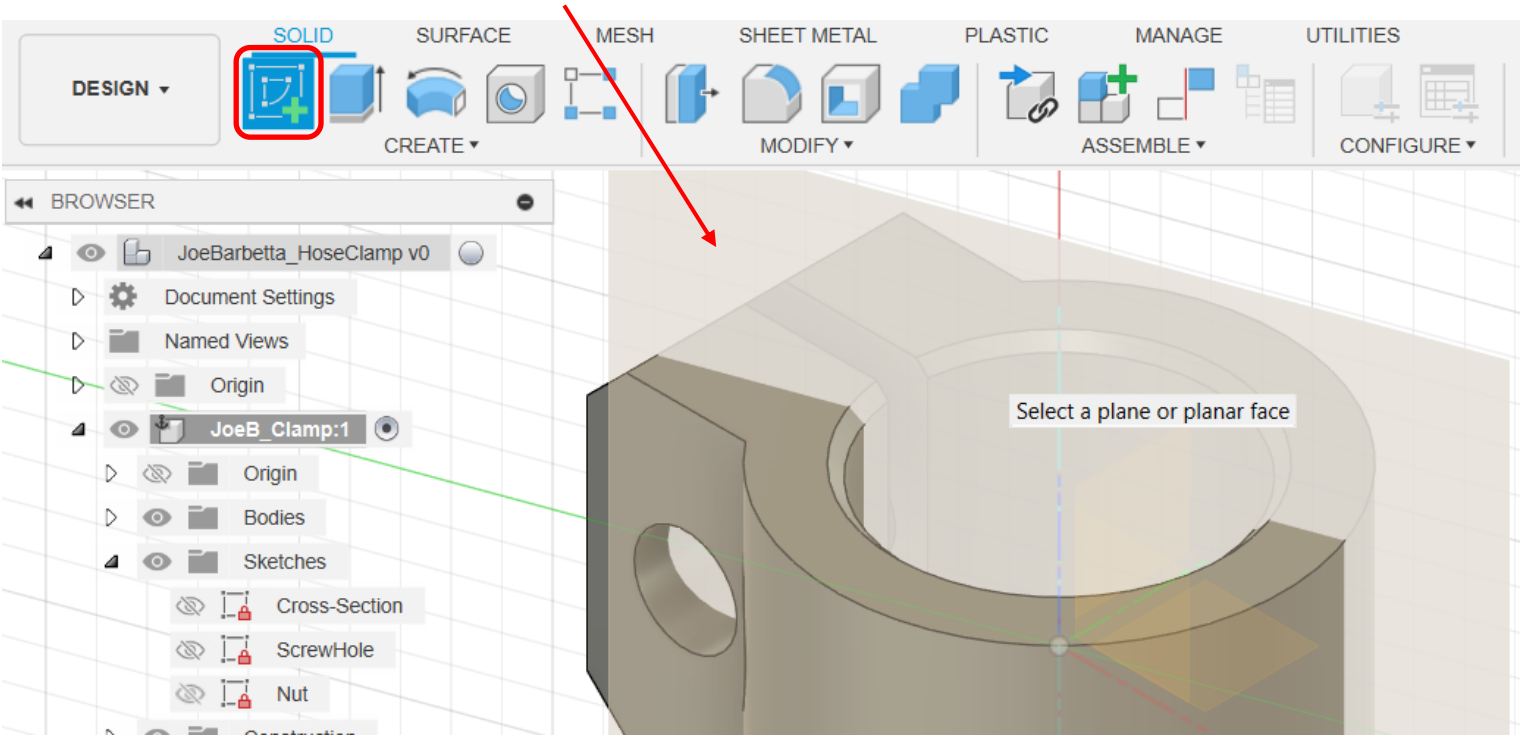


- click on the **Construction Axis** and **turn the rotation circle** until the Angle shows **20.0 deg**. Click **OK**.

The angle is changed to help insure the text doesn't appear on the screw head area.

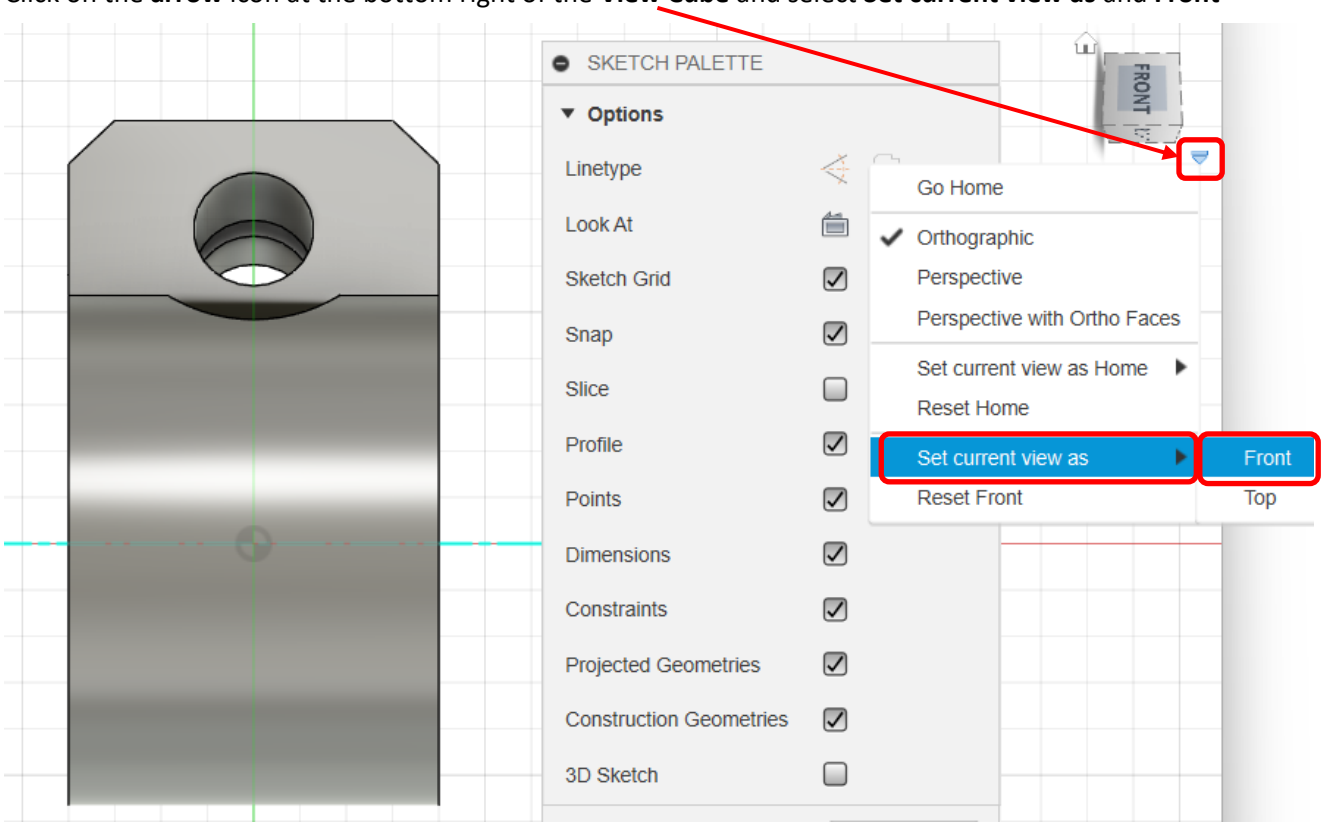


- select **Create Sketch** and click on the **Construction Plane**



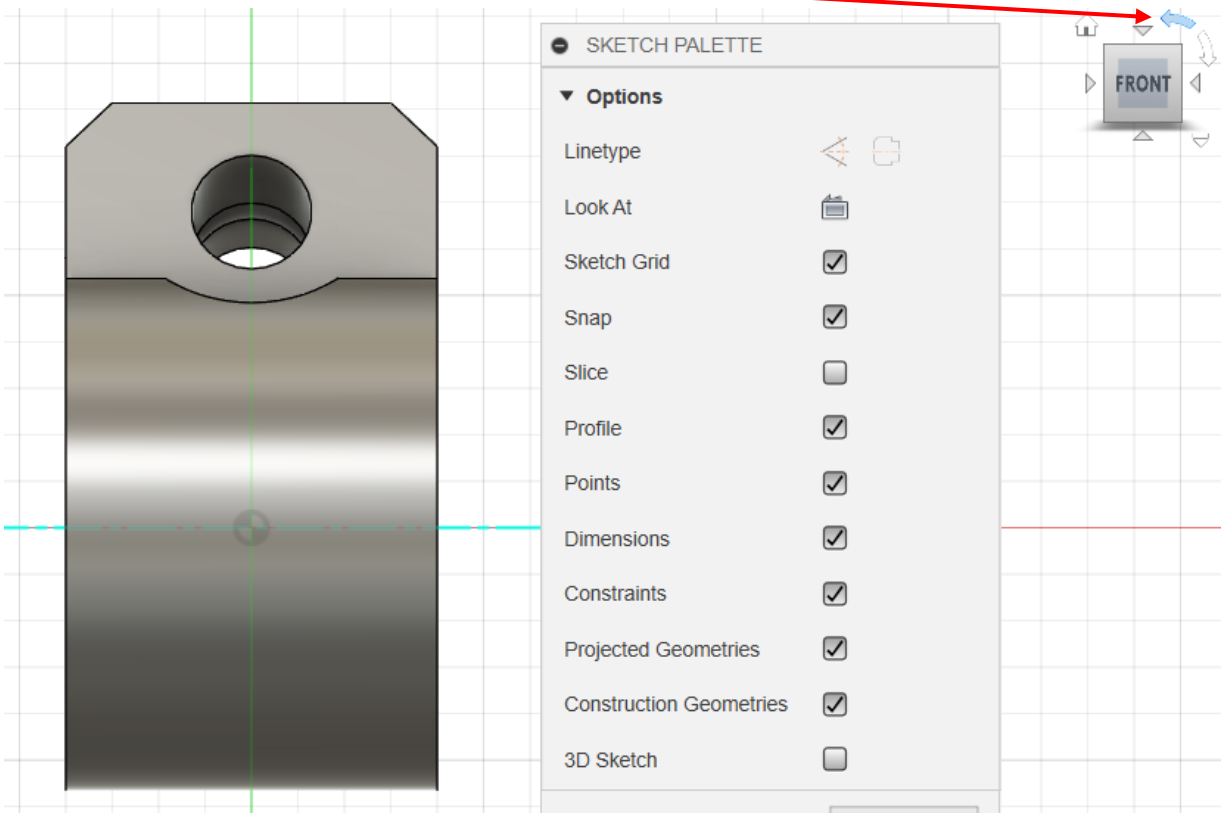
If the Sketch view is rotated, as shown below, it should be rotated so the Z-axis is straight up. Normally, 2 curved arrows will appear at the View Cube when the mouse hovers over it. However, because the Sketch was created on a angled Construction Plane, these arrows are not available.

Click on the **arrow** icon at the bottom right of the **View Cube** and select **Set current view as** and **Front**

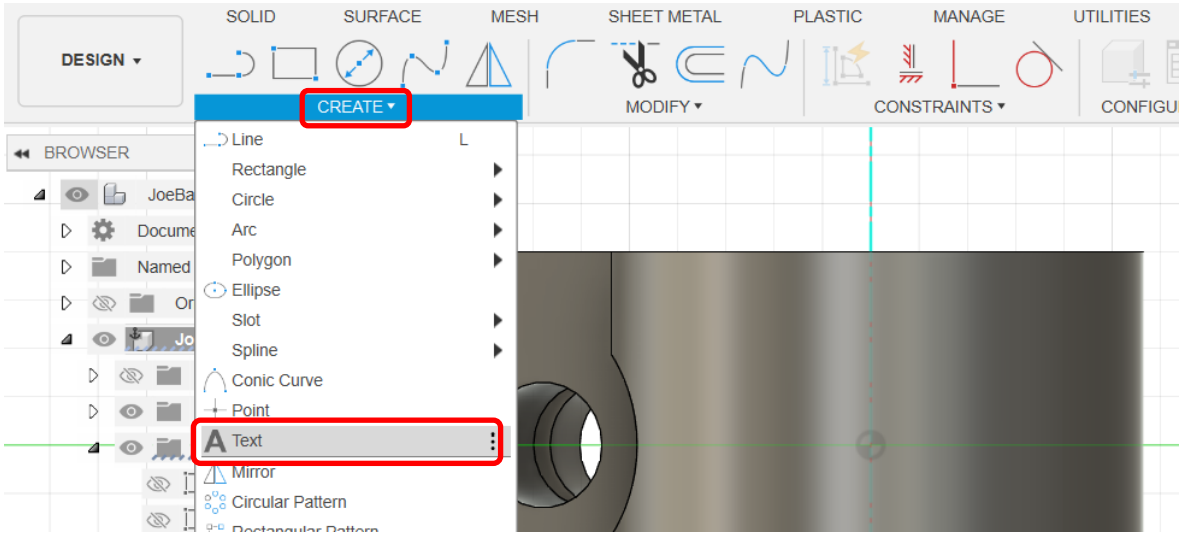


Now the curved arrows are available.

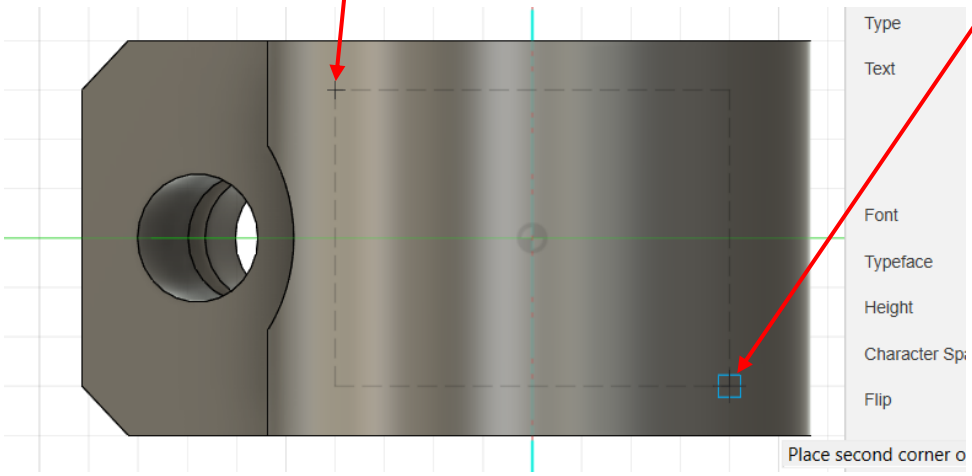
- click on the **Curved Arrow** to rotate the view



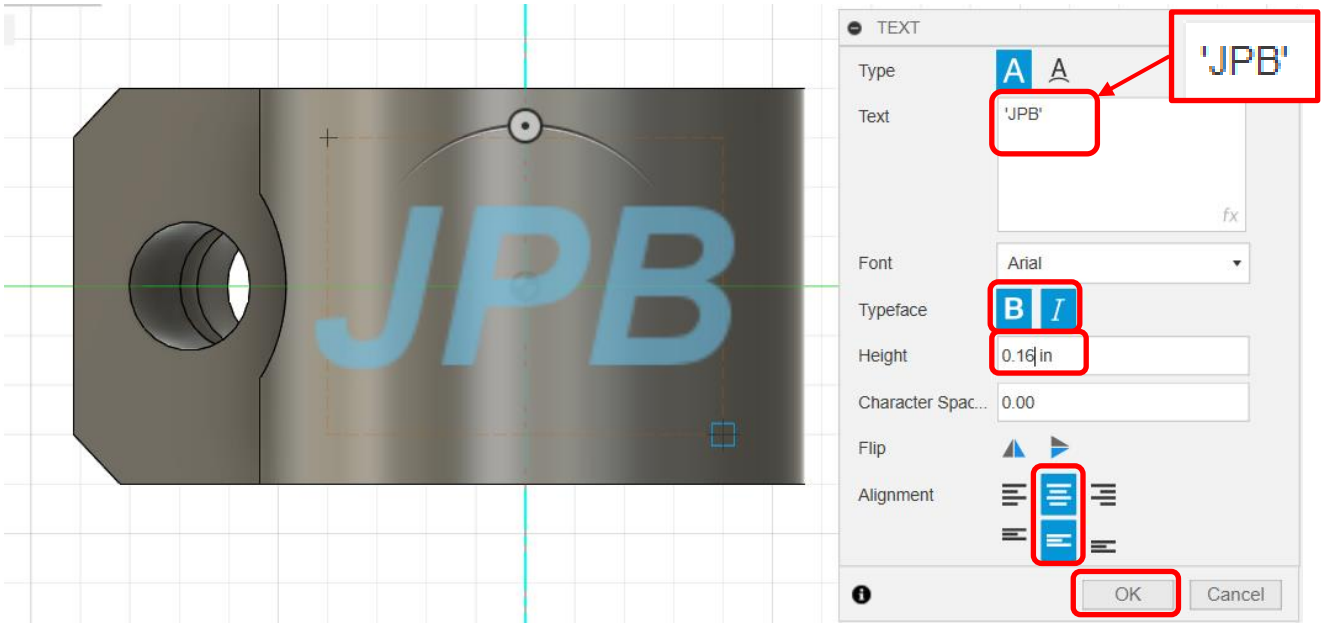
- from the **CREATE** menu, select **Text**



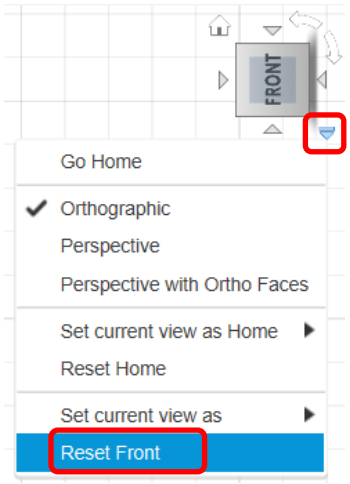
- click on spot near the **top left corner** of the cylindrical region. These positions are not critical.
- extend the rectangle down and to the right and click on a spot near the **bottom right corner**



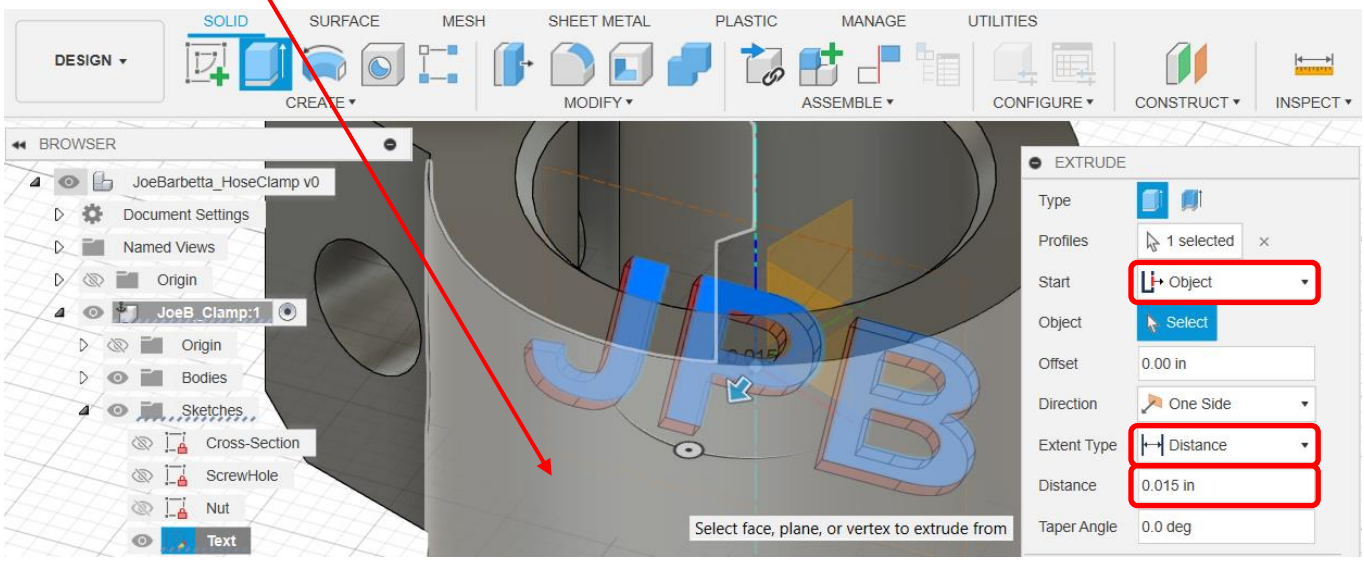
- enter your **3 initials** in the **Text** box **preceeded by and followed by a single quote**, e.g. **'JPB'**
- set the **Height** to **0.16**
- click on the **Bold** and **Italic** icons to highlight them and click on the **2 center Alignment** options, and click **OK**



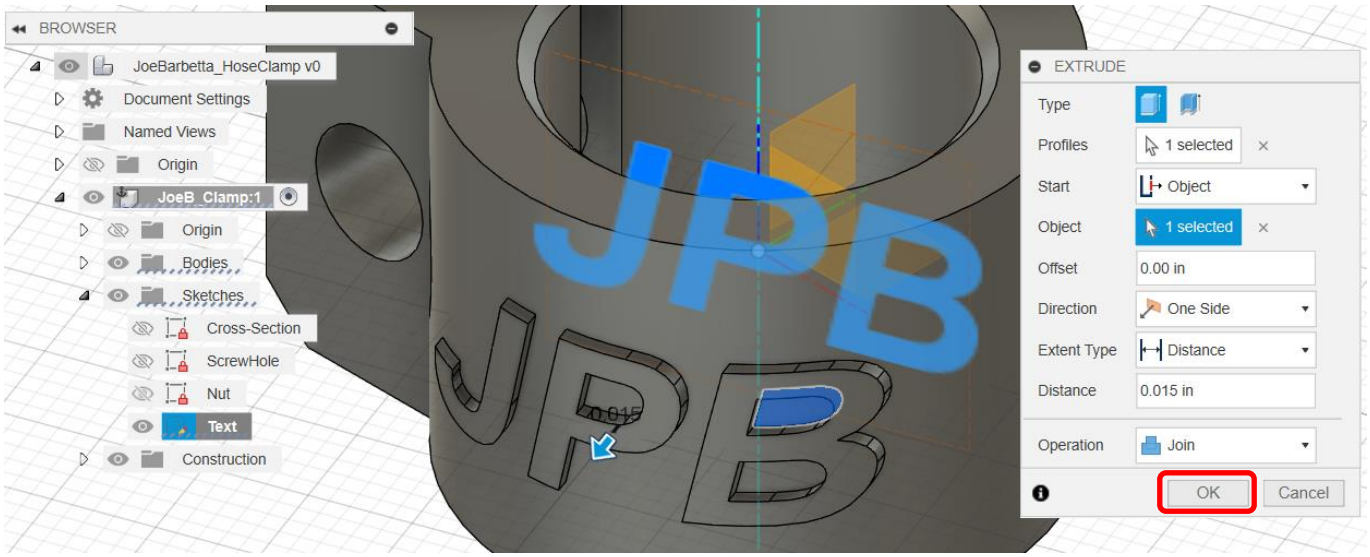
- open the **View Cube** menu and select **Reset Front**
- click **Finish Sketch**



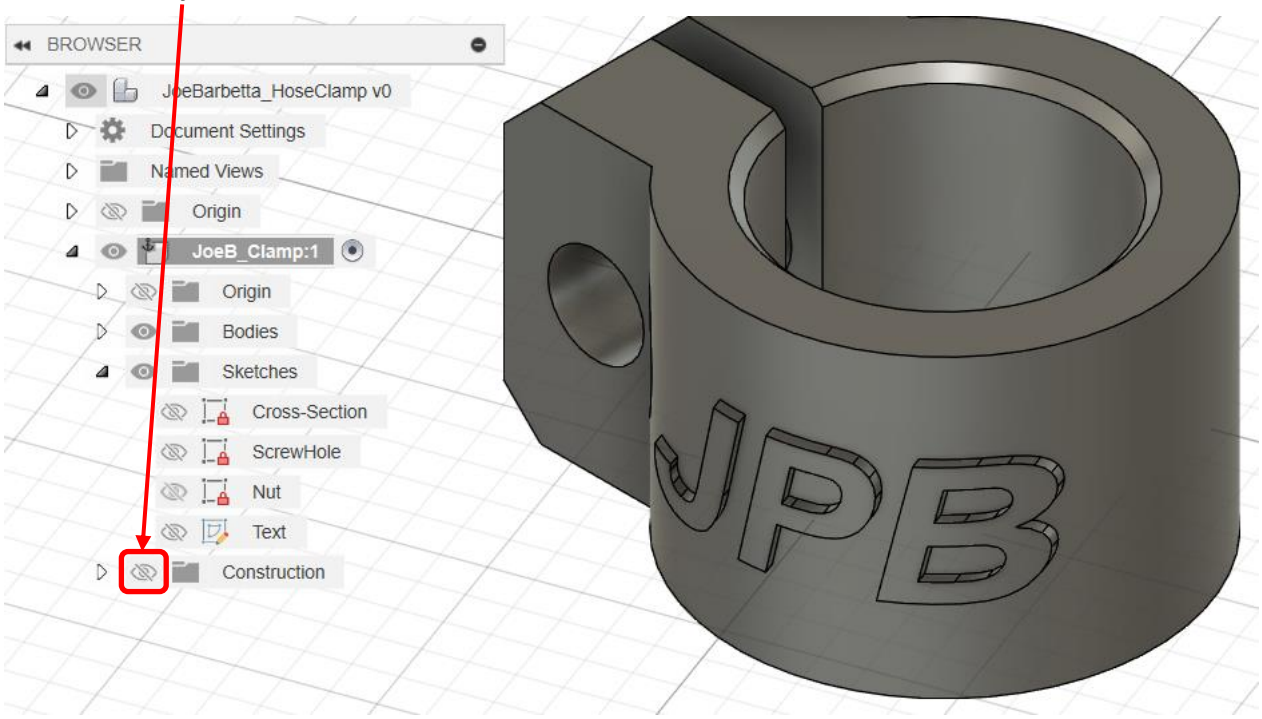
- zoom into the text just created
- select the **Extrude** tool
- select the **Object option** shown for **Start**, **Distance** for **Extent Type**, and **0.015** for **Distance**
- click on the **front curved surface** of the clamp



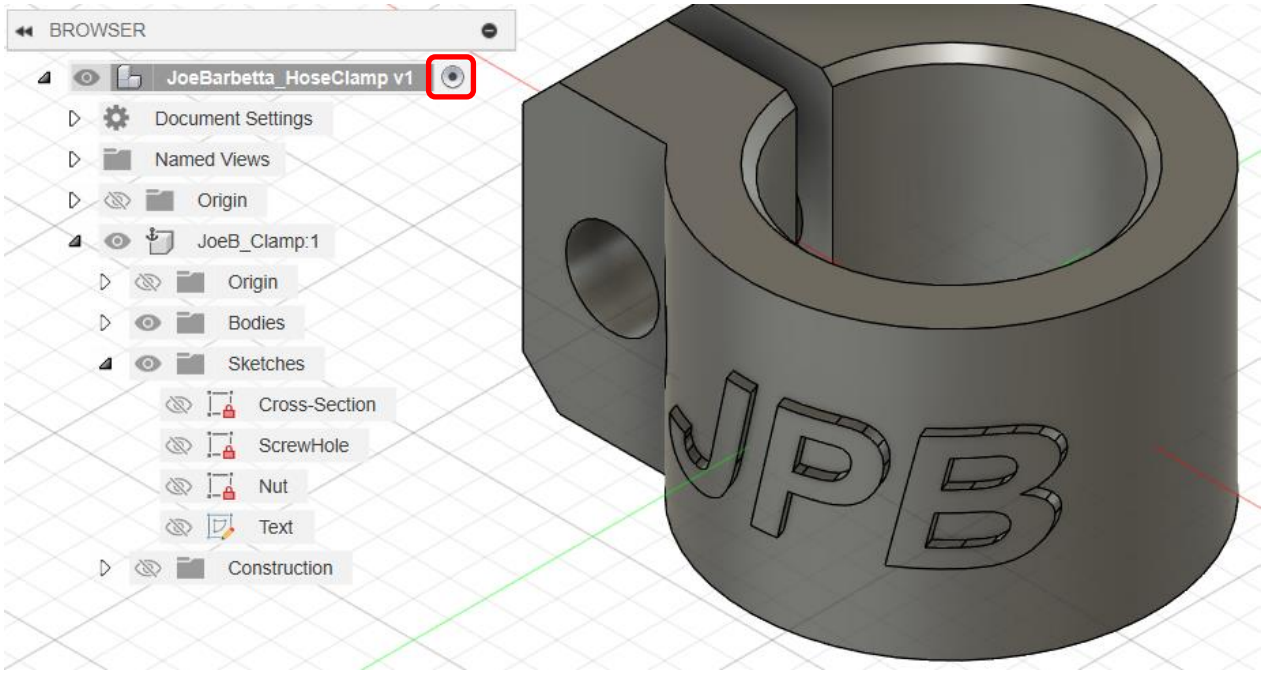
- click **OK**



- click on the **eye** icon for the **Construction** folder to hide the Construction Axis

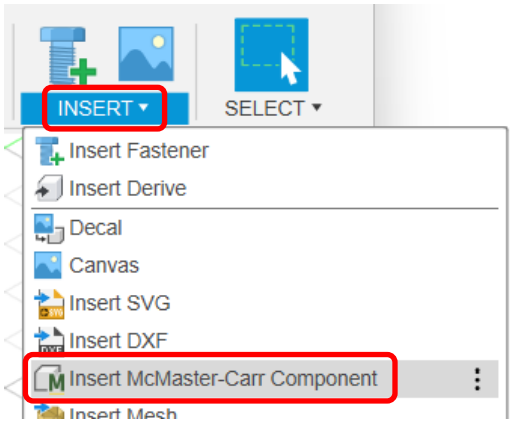


- click on the **circle** for the **Project Name** to Activate it

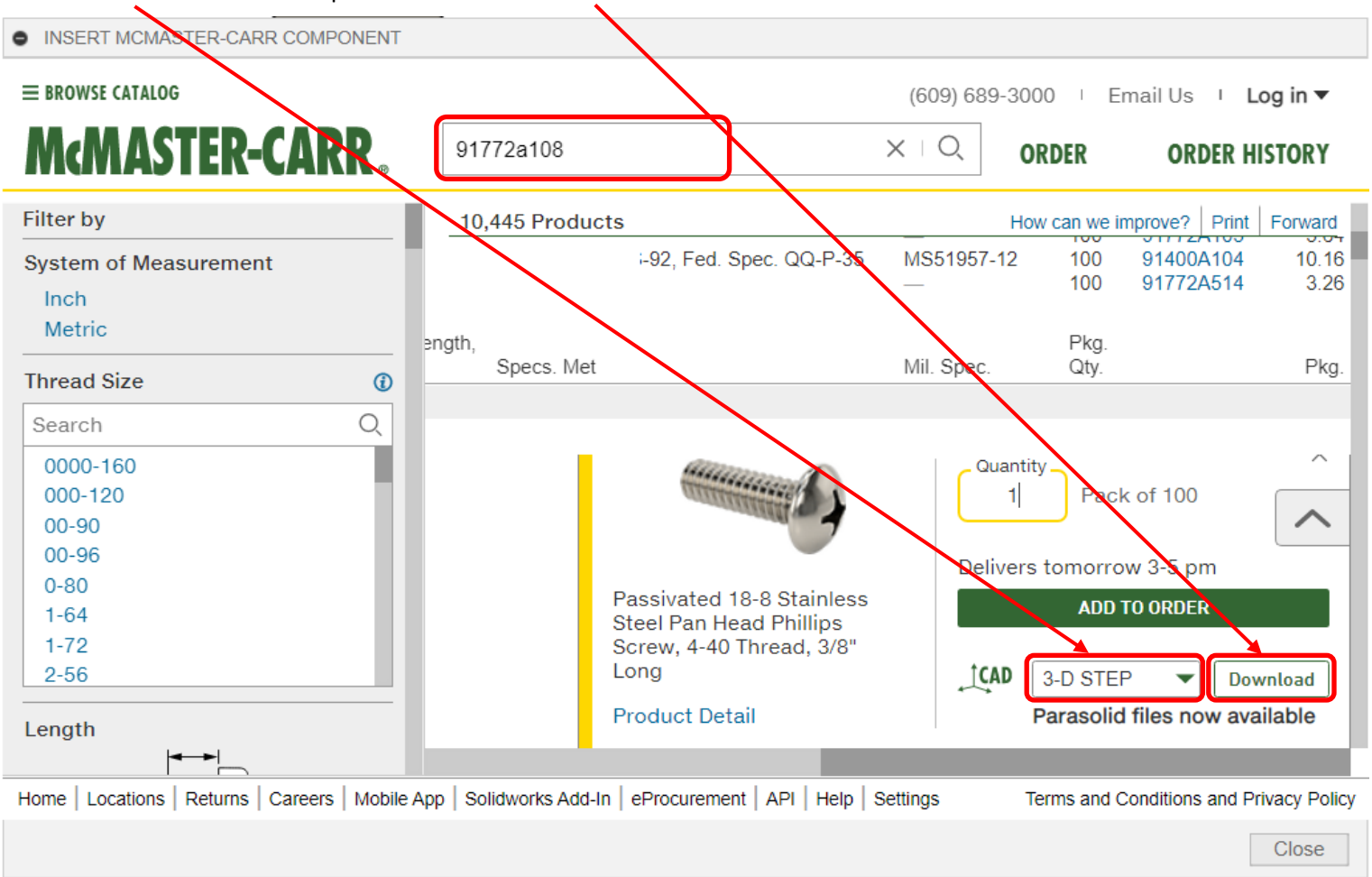


Inserting McMaster-Carr Components

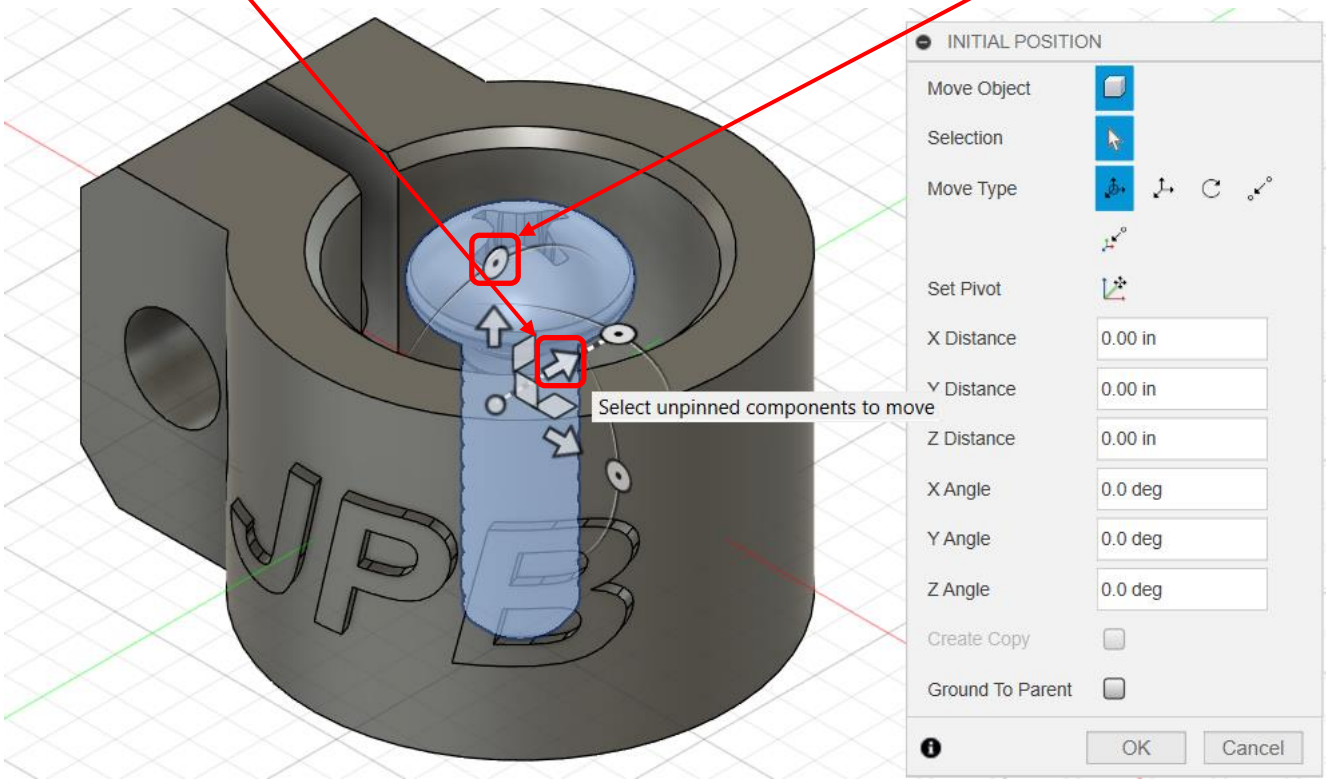
- from the **INSERT** menu at the top right of the Fusion screen and select **Insert McMaster-Carr Component**



- enter the McMaster-Carr part number in the search box **91772A108** and press the **Enter key**
- select **3-D STEP** for the CAD option and click the **Download** button

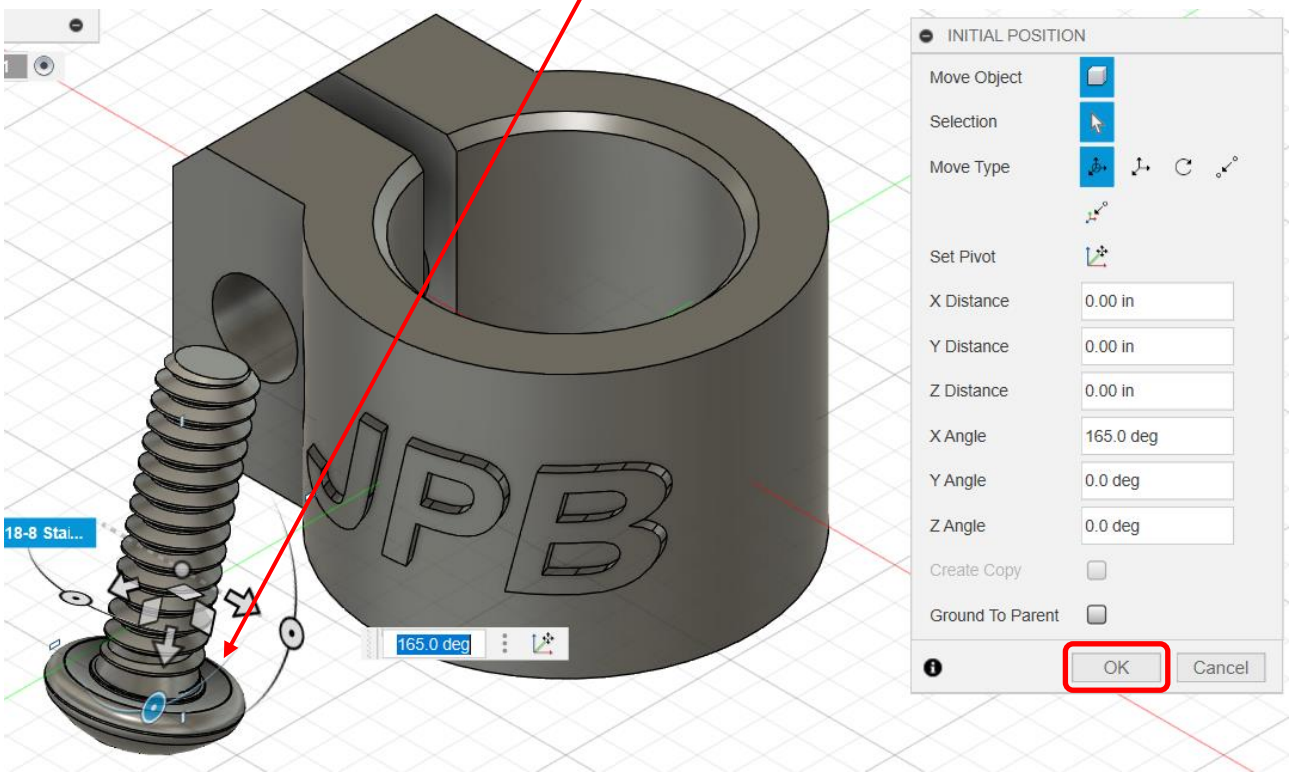


- drag the **move arrow** to move the screw outside of the clamp and then drag the **rotation arc handle** to access the bottom of the screw head, as shown in the next picture.

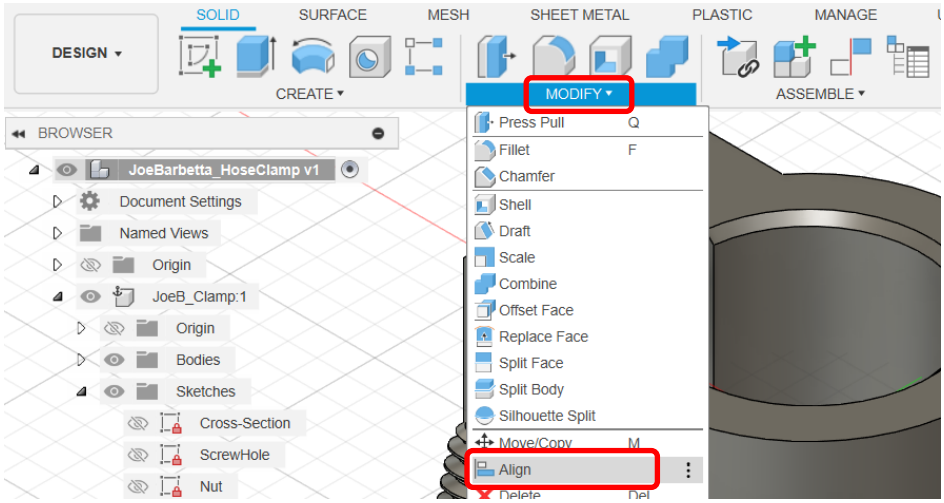


The screw position is not critical. The **underside of the screw head** just needs to be visible.

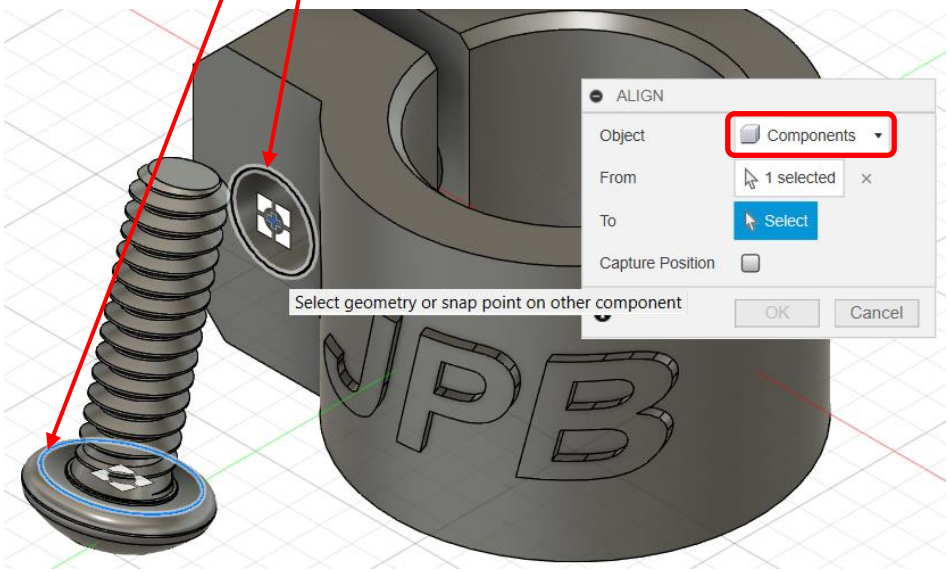
- click **OK**

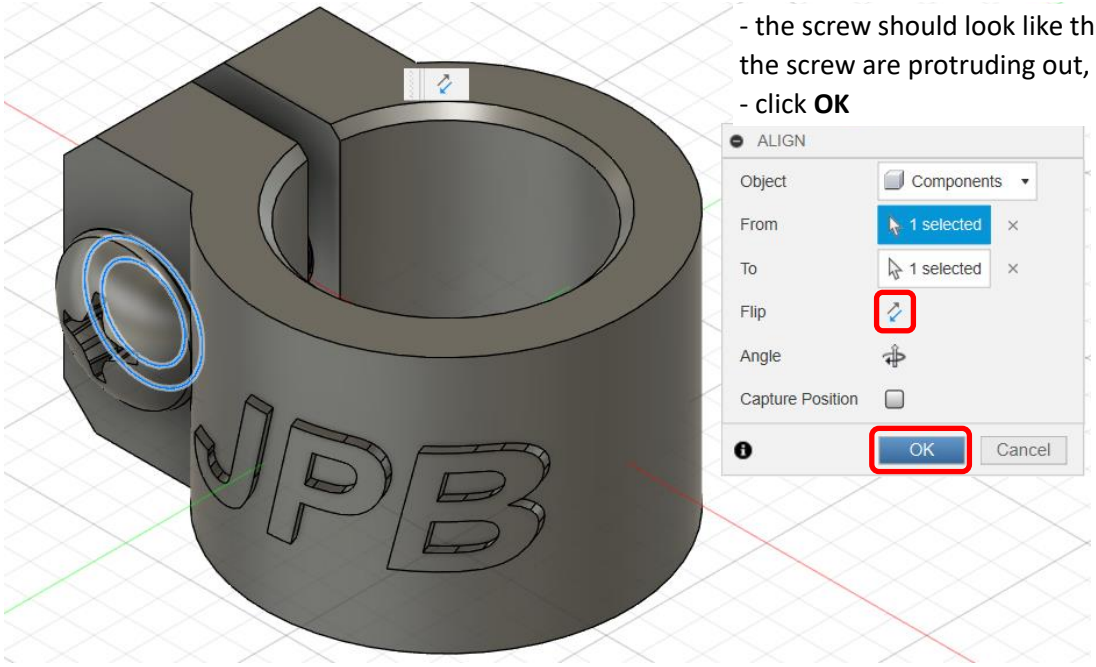


- from the **MODIFY** menu, select **Align**



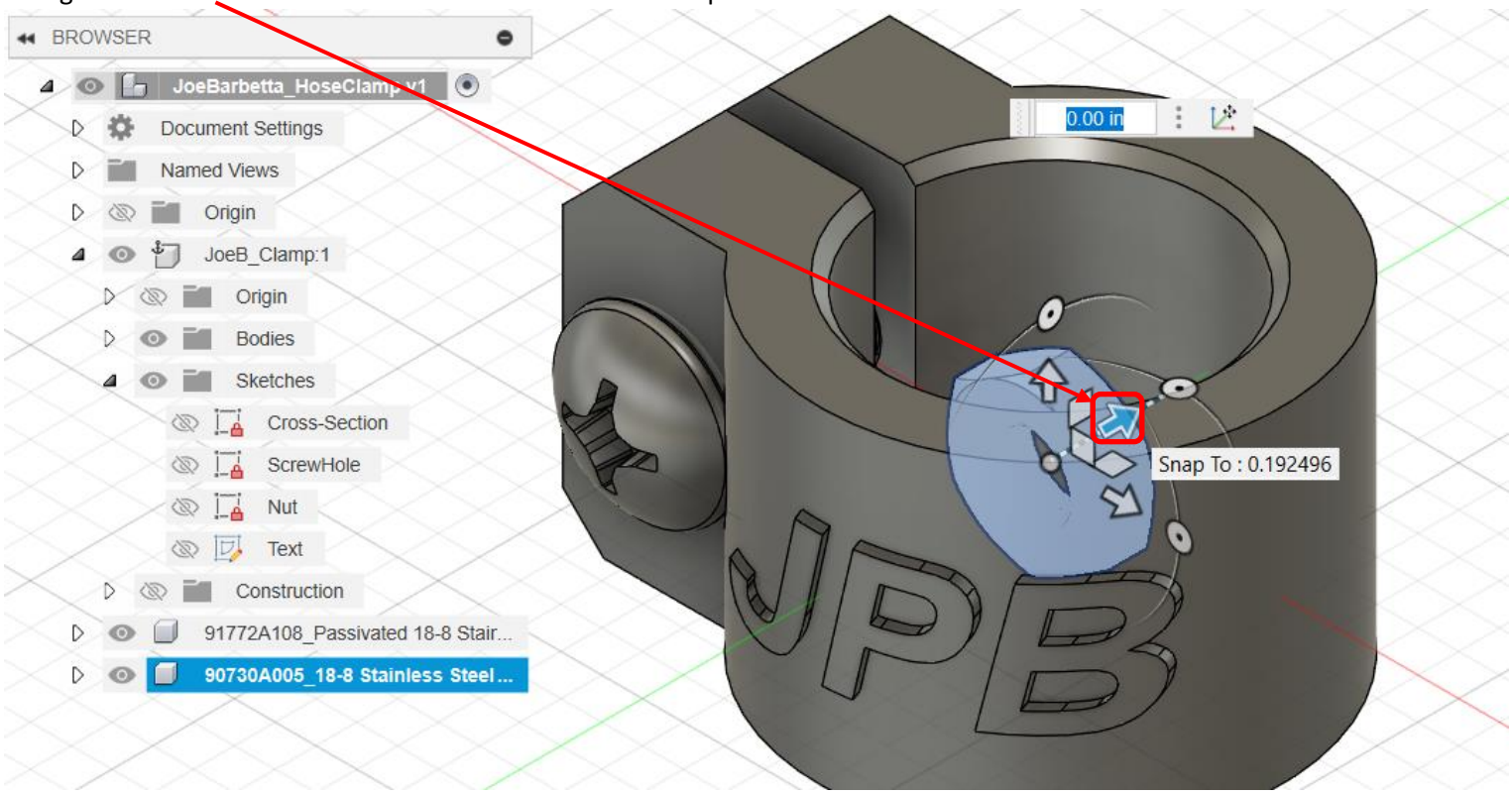
- ensure that **Object** is set to **Components**
- click on the **circular edge** on the bottom of the screw head, which should result in a white target symbol
- click on the **edge of the hole**, which should cause the screw to move into the hole



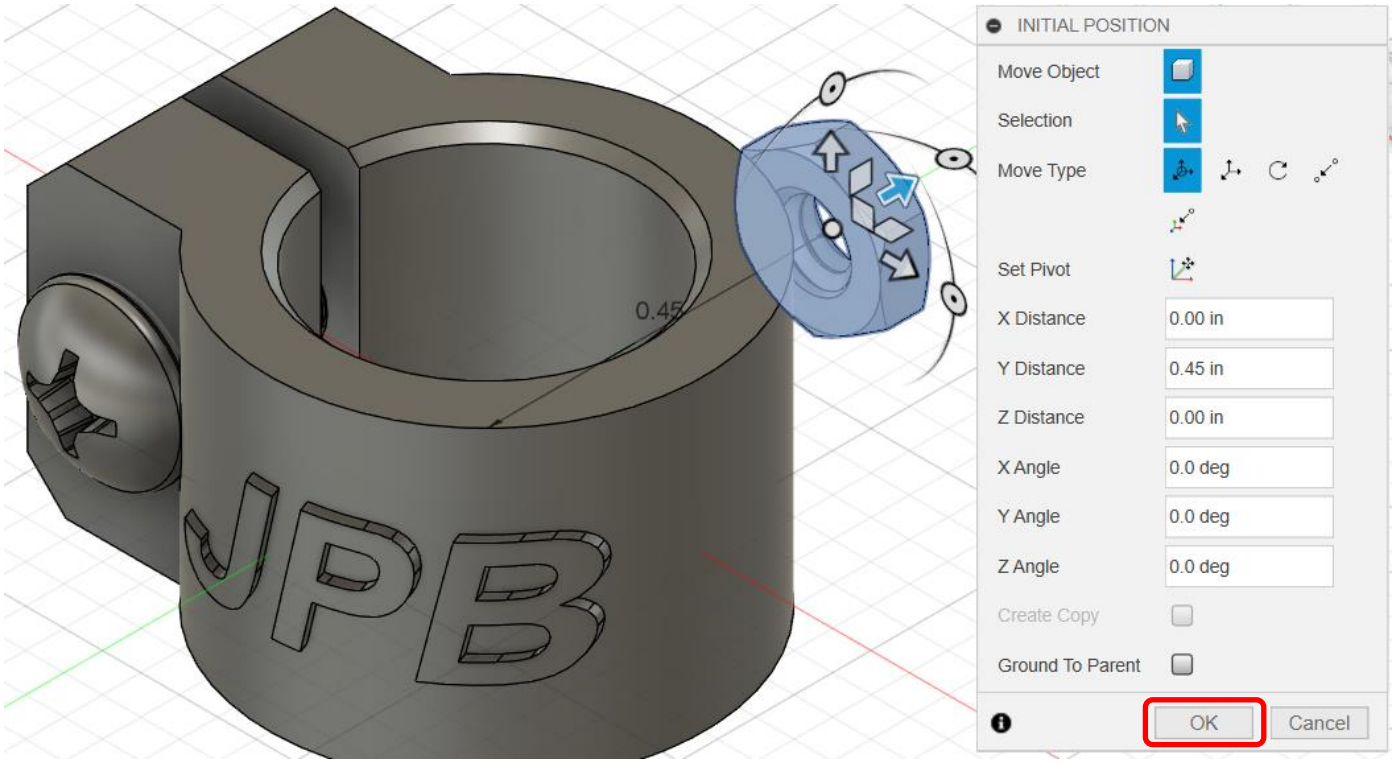


- the screw should look like that on the left, but if the threads of the screw are protruding out, click on the **Flip** icon
- click **OK**

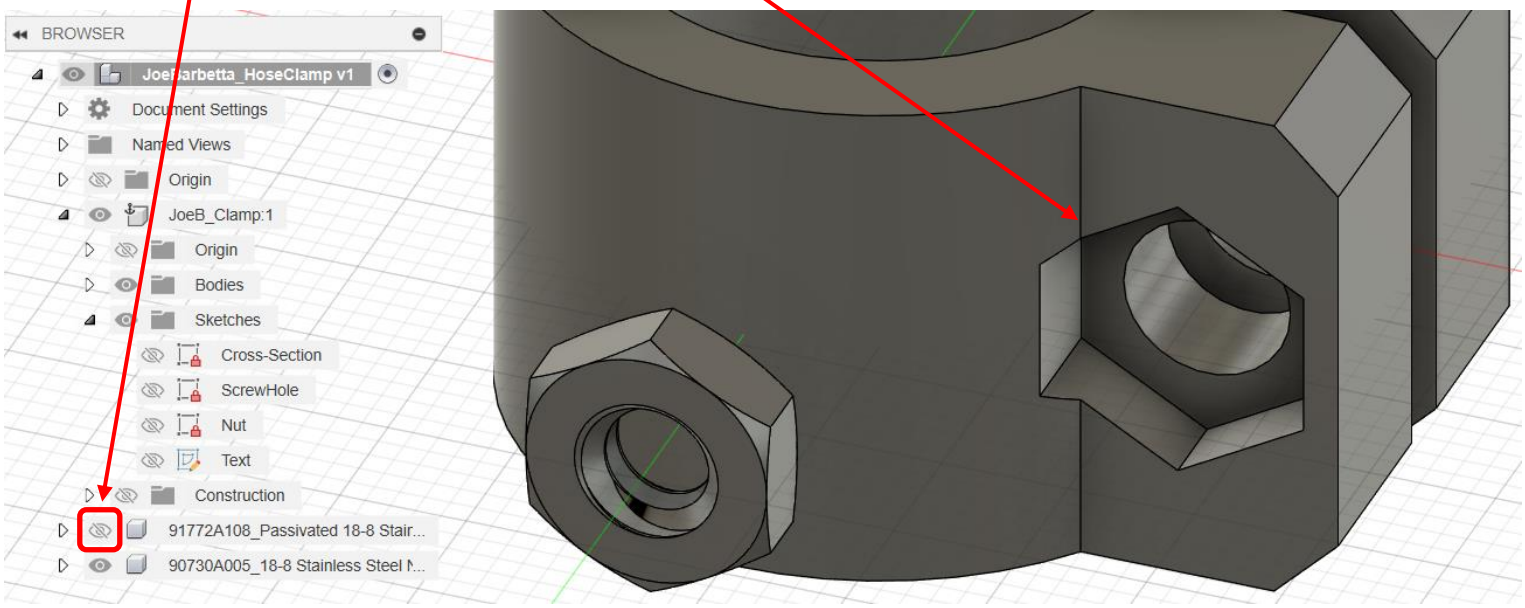
- from the **INSERT** menu at the top right of the Fusion screen and select **Insert McMaster-Carr Component**
- download the 3-D STEP file for the following McMaster-Carr part **90730A005**
- drag the **move arrow** to move the nut outside of the Clamp



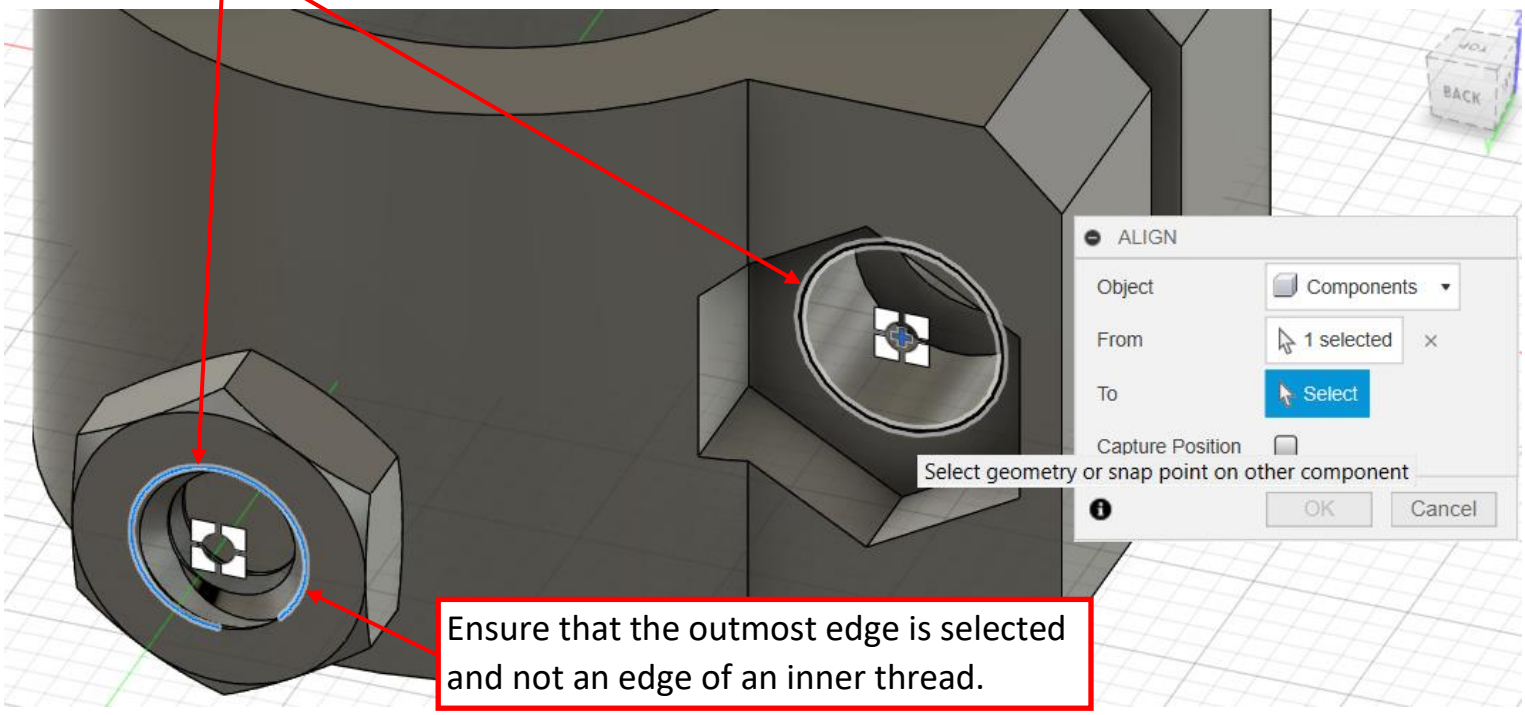
- when the nut is outside of the clamp, click **OK**. The position is not critical.



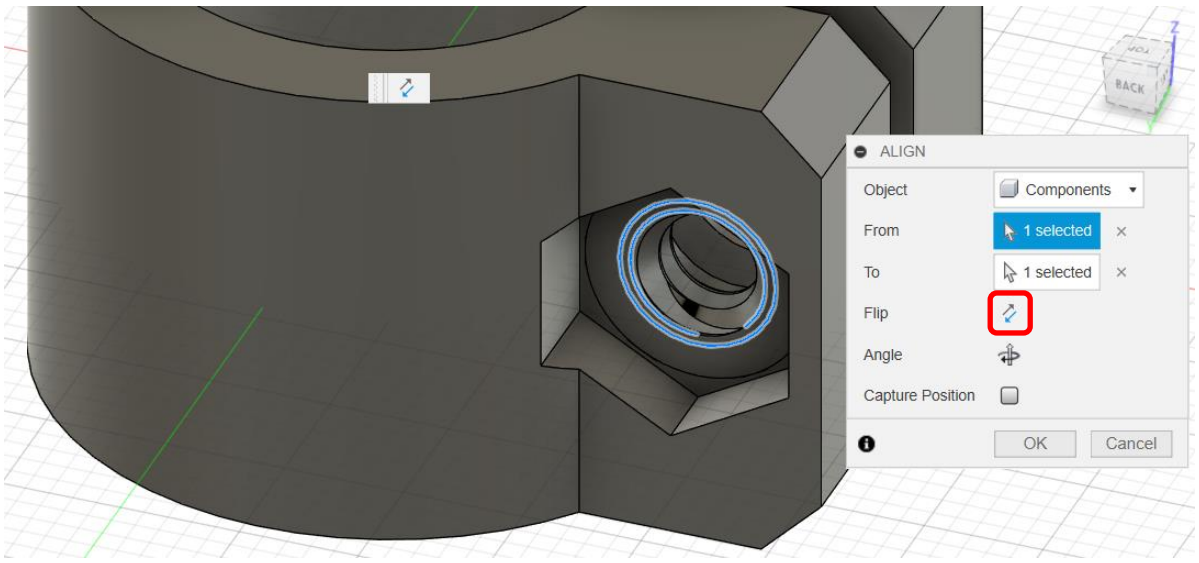
- use the **View Cube** to rotate the view to access the **nut pocket**
- click on the **eye** icon for the screw component to hide it



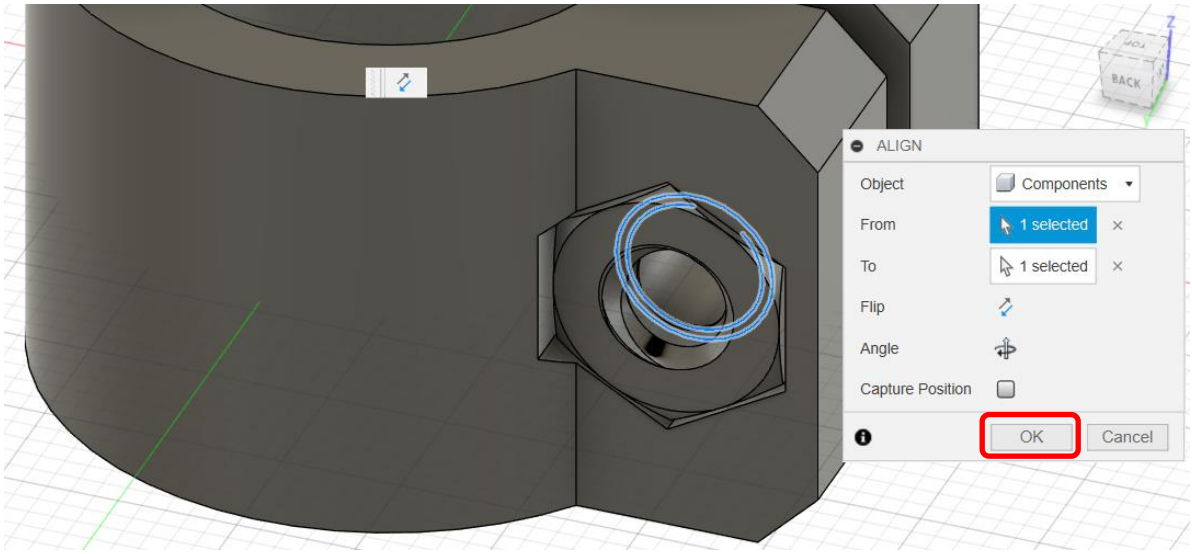
- from the **MODIFY** menu, select **Align**
- click on the **inner edge** of the nut as shown, which should cause the target symbol to appear
- click on the **edge of the hole**, which should cause the nut to jump into the pocket



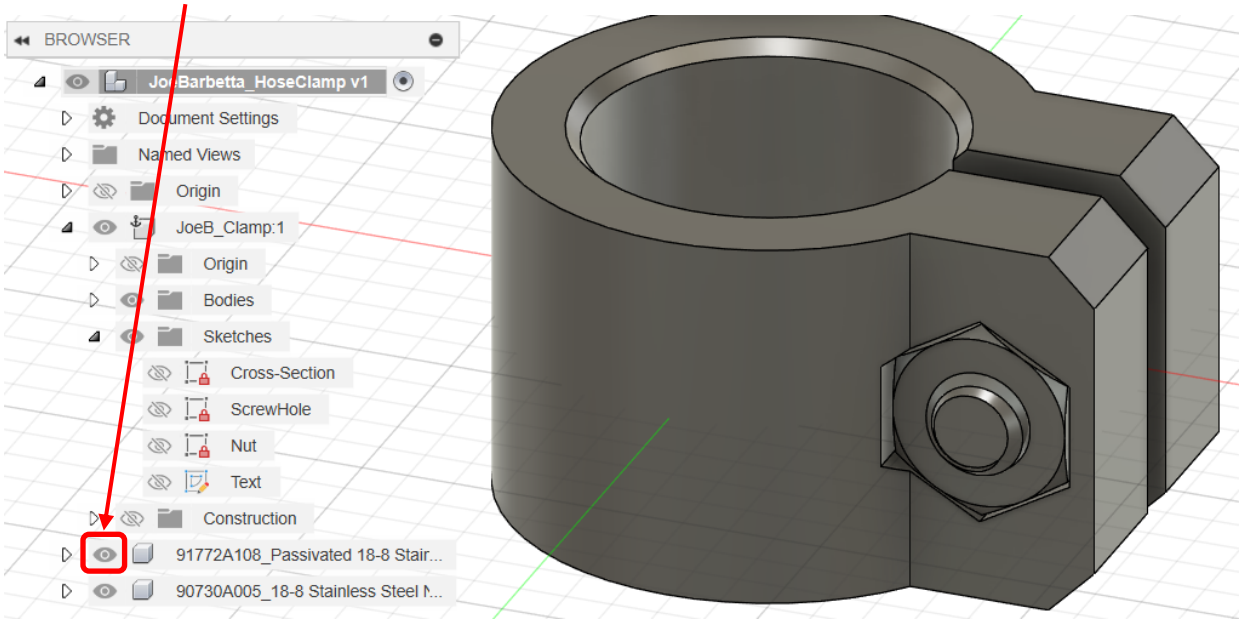
- if the nut is **flipped inward** as this is, click on the **Flip** icon so that it is sitting in the pocket, as the next picture shows



- click **OK**

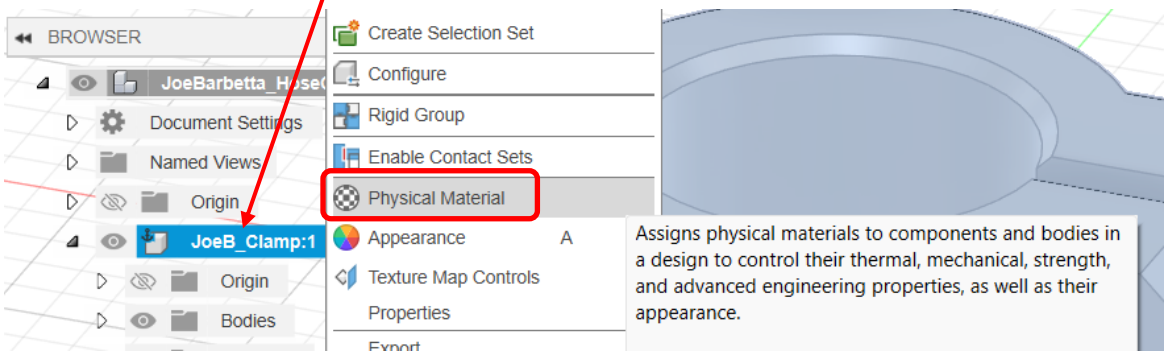


- click on the **eye** icon for the **screw Component** to make it visible again

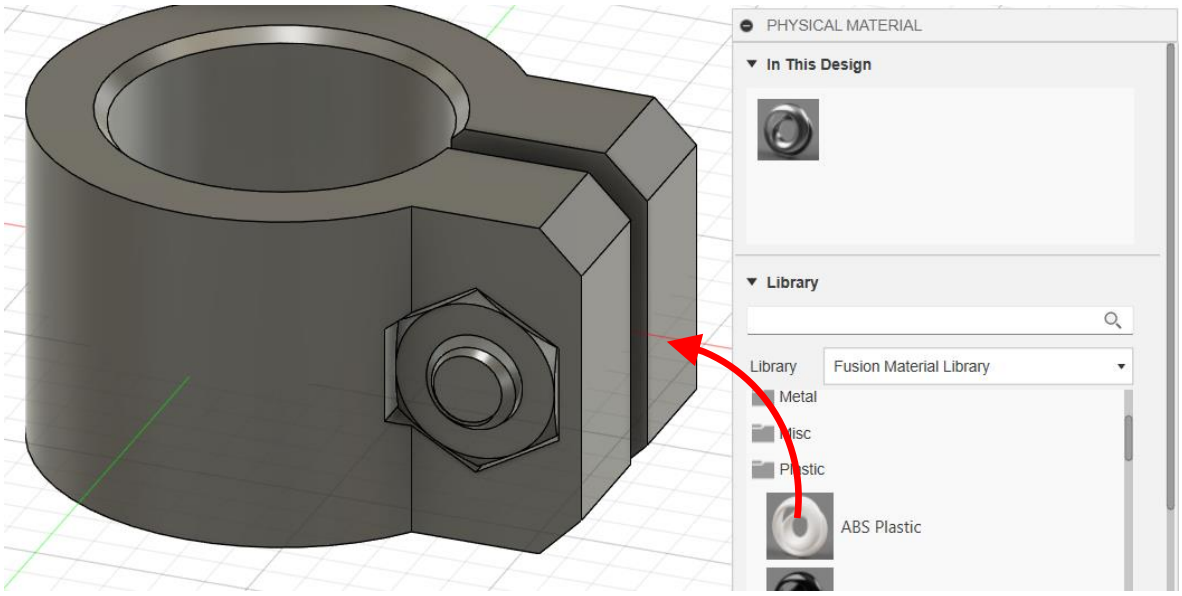


Setting a Material

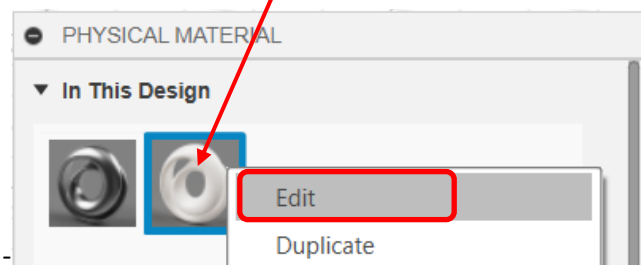
- right-click on the **Component Name** and select **Physical Material**



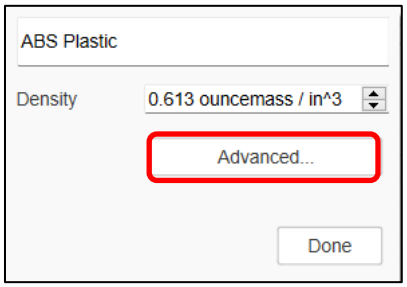
- scroll down to the **Plastic** folder and click on it to access the pastic materials
- drag the **ABS Plastic** icon **onto the clamp body**



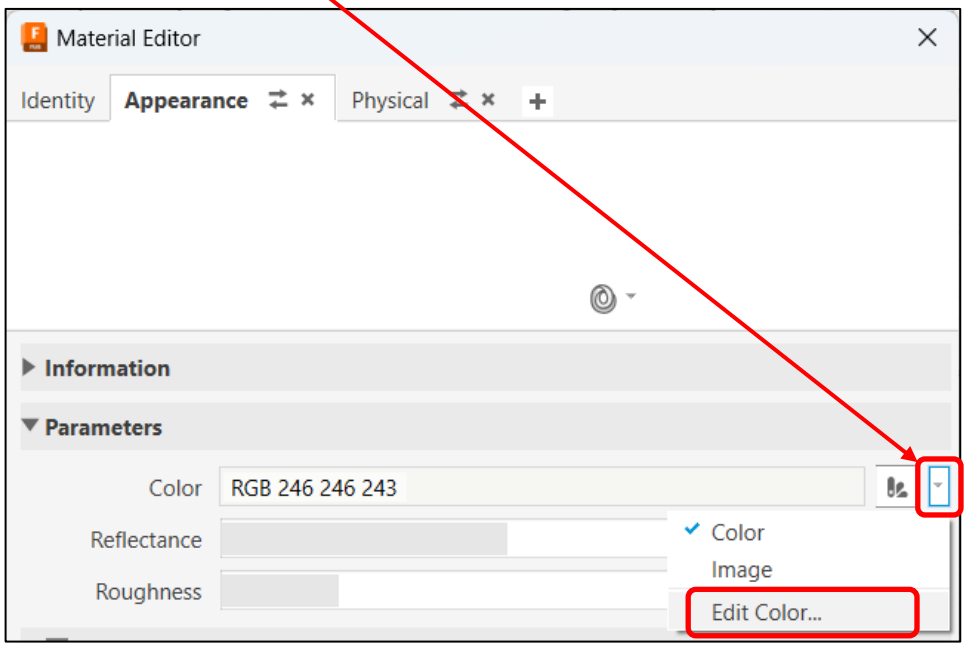
- right-click on the **ABS plastic** icon and select **Edit**



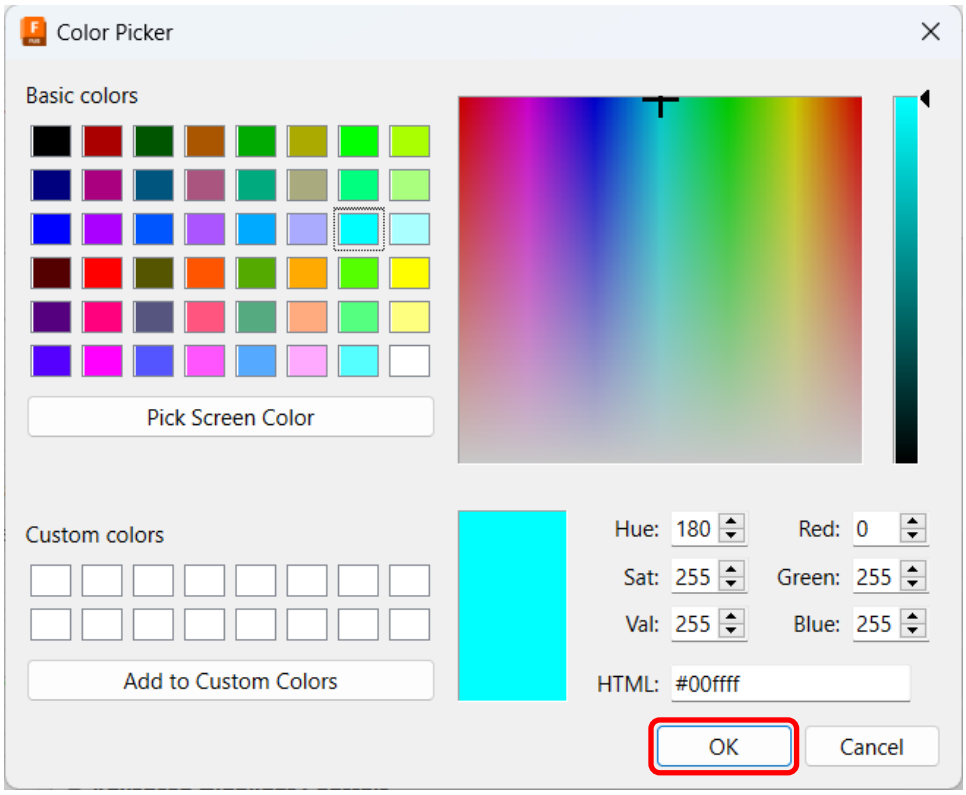
- click on **Advanced**



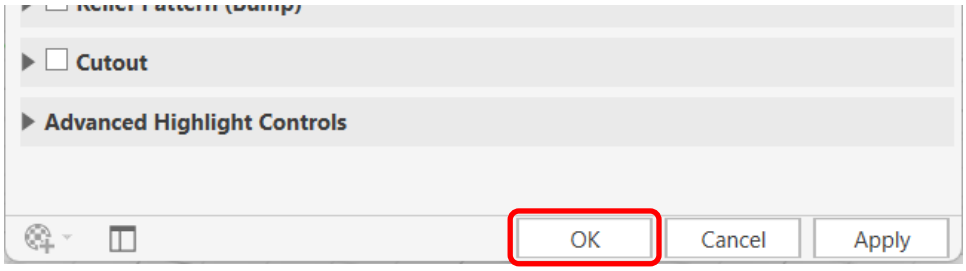
- click on the **drop-down menu button** and select **Edit Color…**



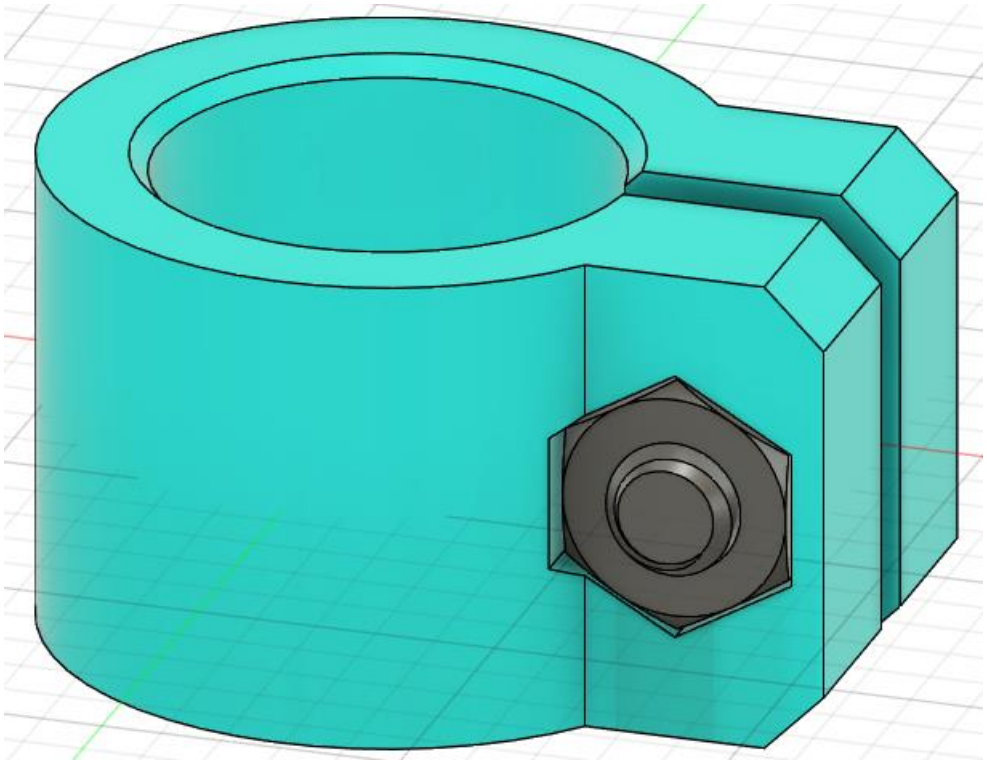
- select your favorite color from either the **Basic colors** boxes or the **color spectrum** box. The right side slider can be used to adjust the brightness. Click **OK**



- click on **OK** at the bottom of the **PHYSICAL MATERIAL** window



The clamp color should change. Note that the color of the screw and nut will not be changed. Fusion uses steel by default. The screw and nut selected from McMaster-Carr are stainless steel.



- click on the **Home** icon at the **View Cube** and admire your creation
- take a **screen shot** of the Fusion screen showing the BROWSER with the **Sketches folder open** and the clamp