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

Did you lose the top to your snake in a can?

Make a new one.

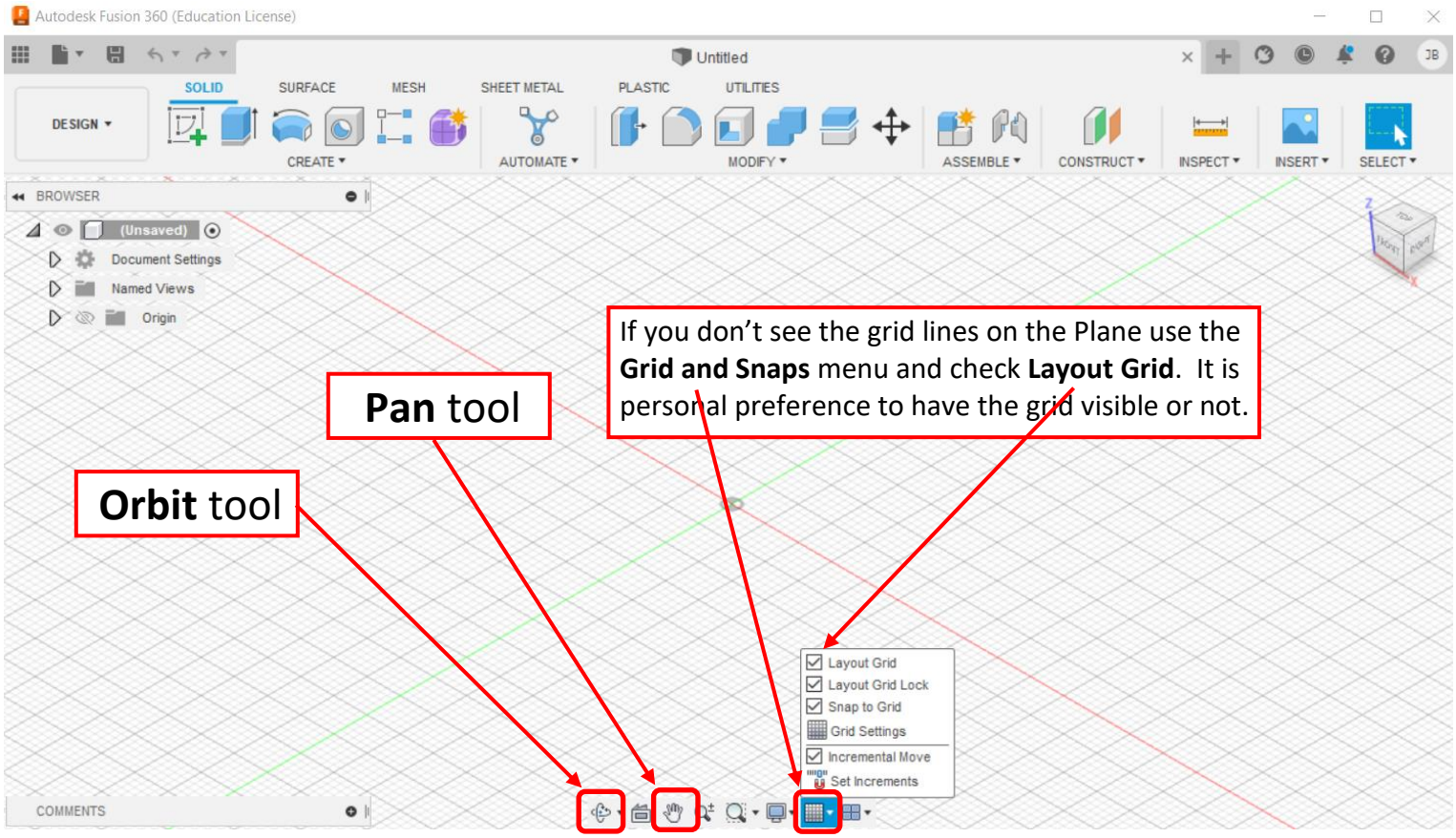
Today's Lesson is Sponsored by Raytheon

Contents

Changing the View of a Design

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

- click on the **Orbit** tool and click somewhere on the **Grid** to practice rotating and changing the angle of the view.
- click on the **Pan** tool and then on the **Grid** to practice moving the view laterally.
- after using the *Orbit* or *Pan* tool one must press the **Esc key** to exit that mode.
- use the **Mouse Wheel** to practice Zooming in and out.



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

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

The View Cube now resembles that on the right.

- click on the **Curved Arrows** at the upper right of the View Cube and practice Rotating the View.
- click on the **Arrows** at the sides of the View Cube to practice jumping to various Views.
- click on the **Home** icon to the upper left of the View Cube. This can always be used to reset the view to the Home View

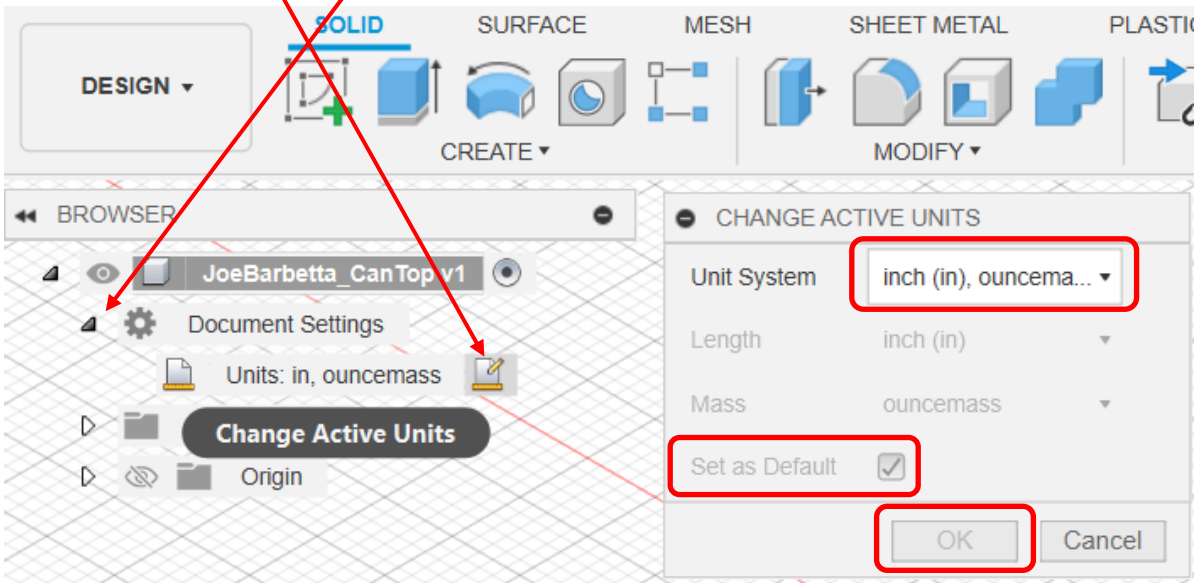
Starting a Design in Fusion (START HERE)

- 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 **_CanTop** e.g. **JoeBarbetta_CanTop** (note the use of the underscore)

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

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

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



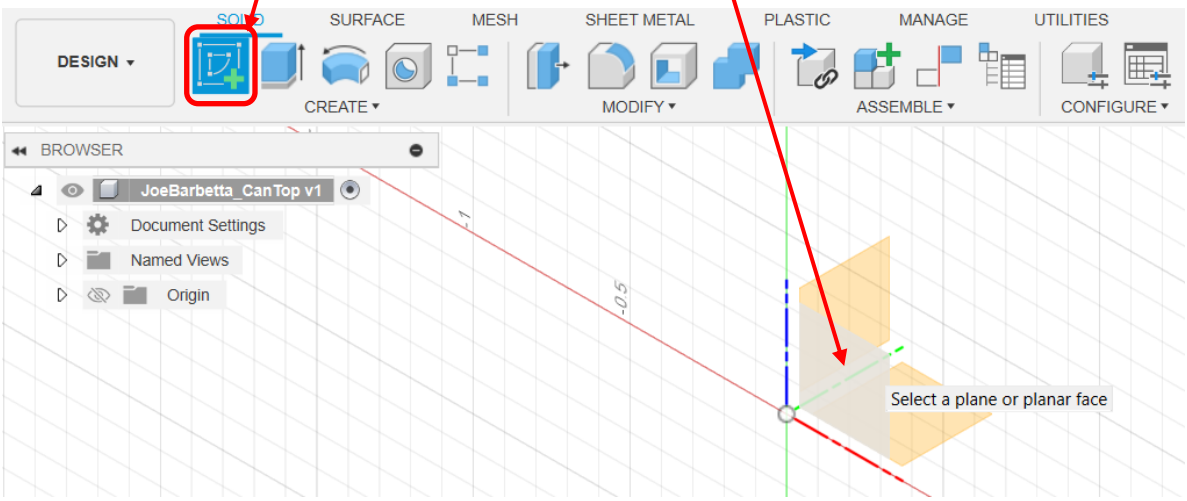
Note that the default units are in mm, which we just changed to inches.

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

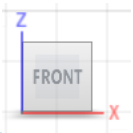
Creating the First Sketch

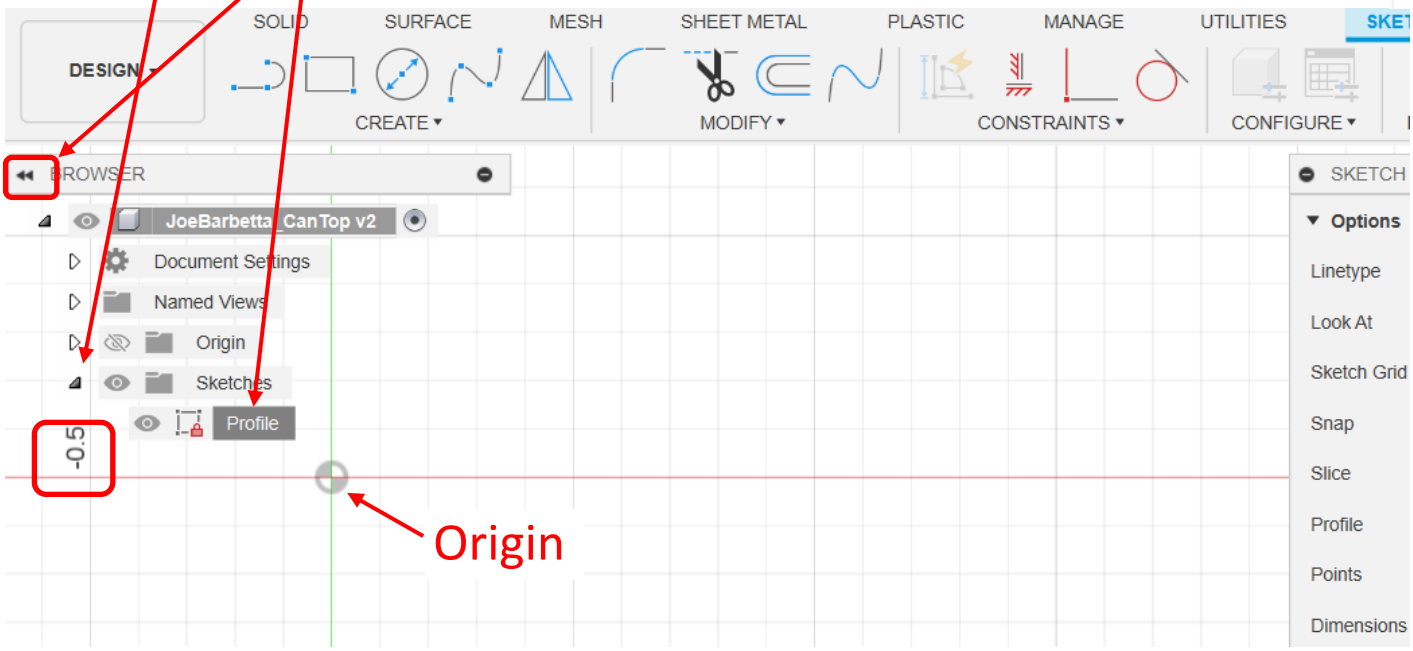
Note that a Fusion expert may tell you to create a Component first. Just say **"Dude. I'm just making a can top."**

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

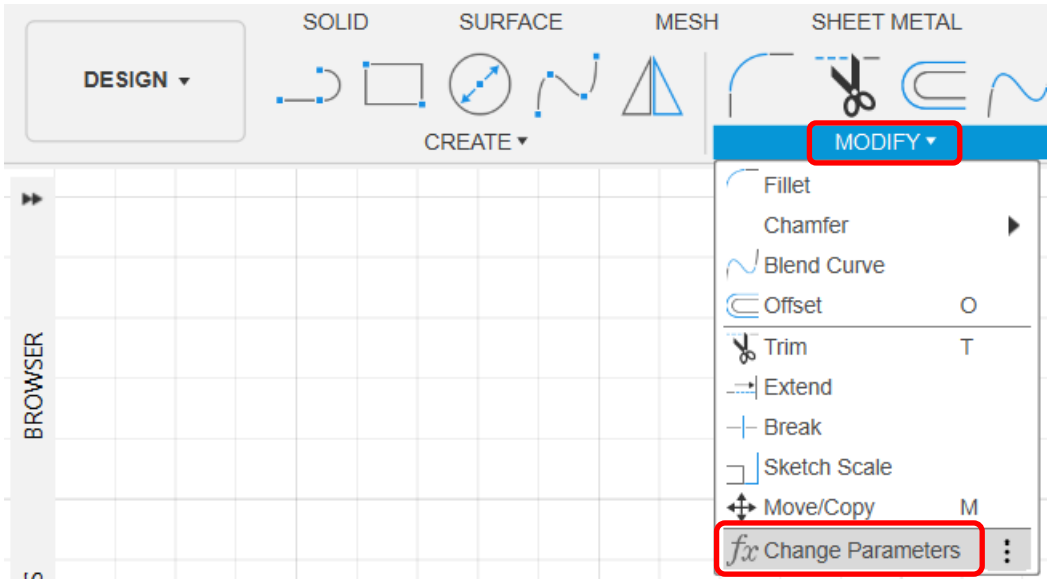


- zoom in as shown below. The scale labels can give an idea of how far one is zoomed in. The **View Cube** should indicate you are sketching on the **FRONT X-Z Plane**.
- click on the **arrow** next to the Sketches folder to open it
- right click on the **Sketch name**, select **Rename** from the menu and rename the Sketch to **Profile**
- click on the **double-arrows** for the BROWSER to hide it to make more room for your sketch

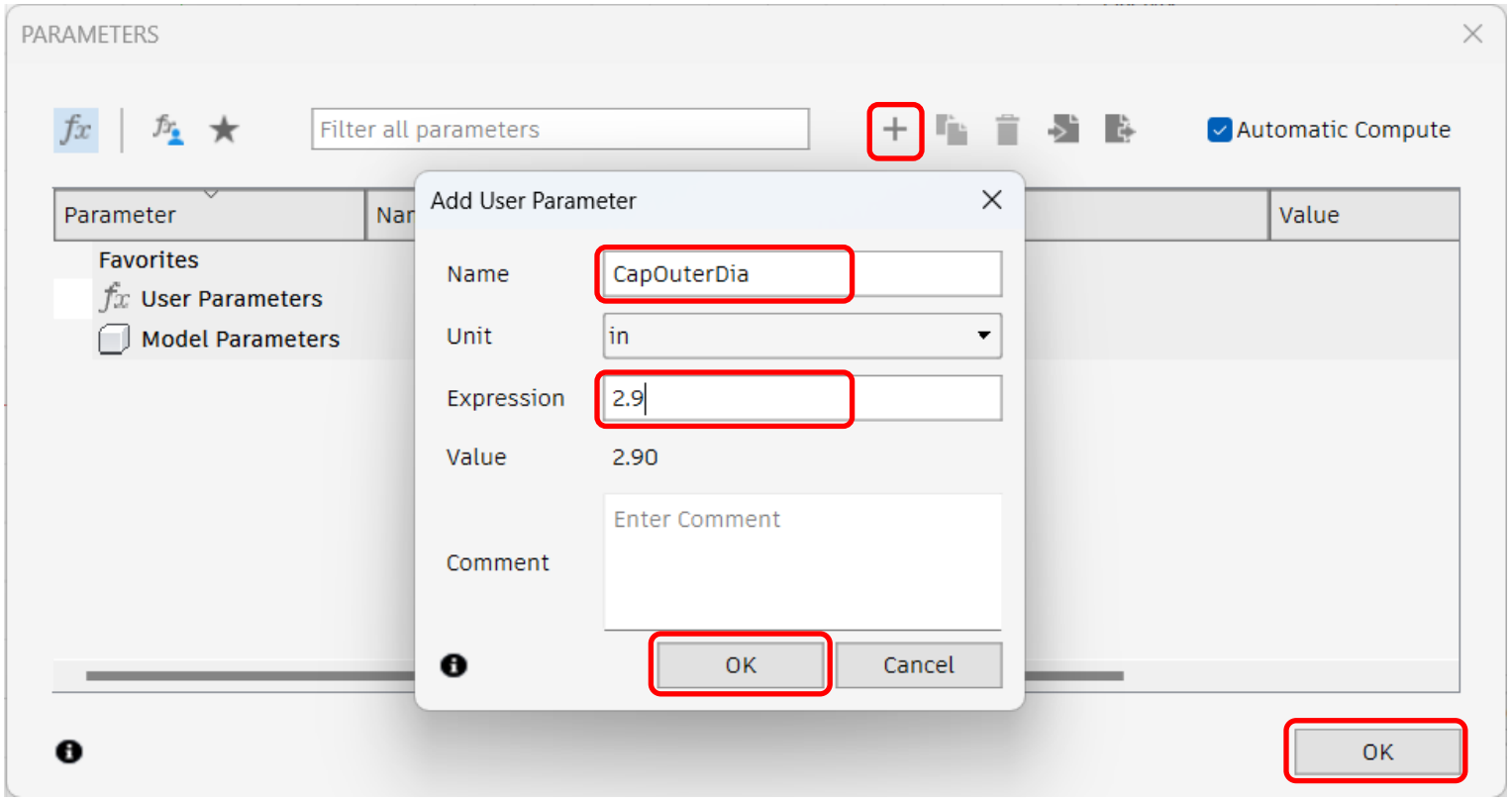




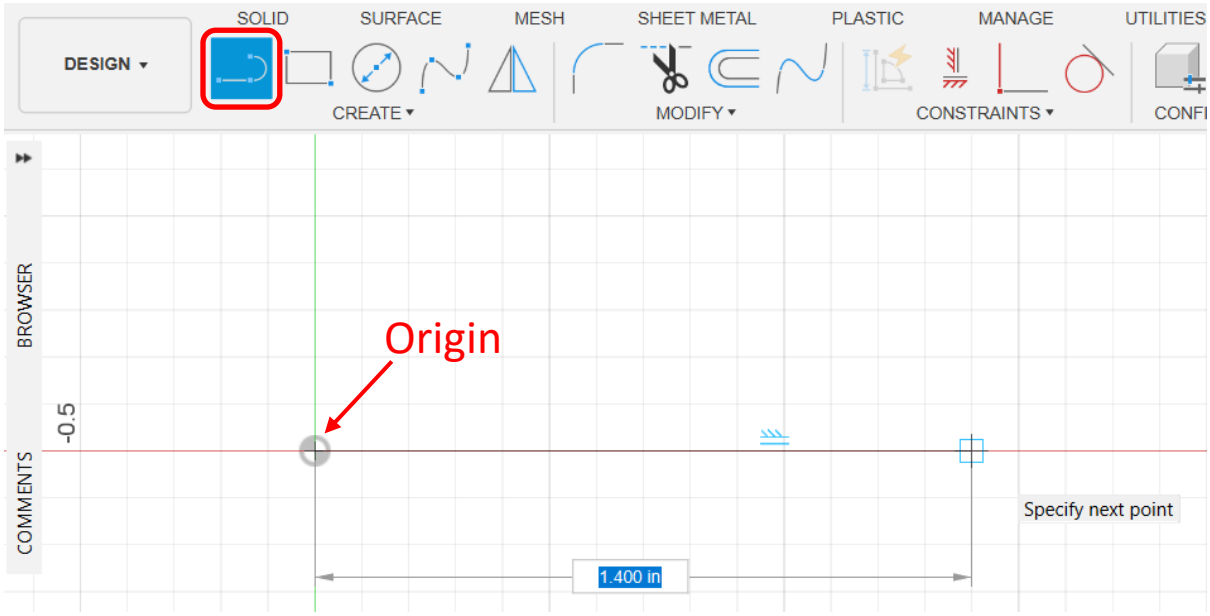
- from the **MODIFY** menu select **Change Parameters**
- if a Parametric Text message window appears, click its **OK** button



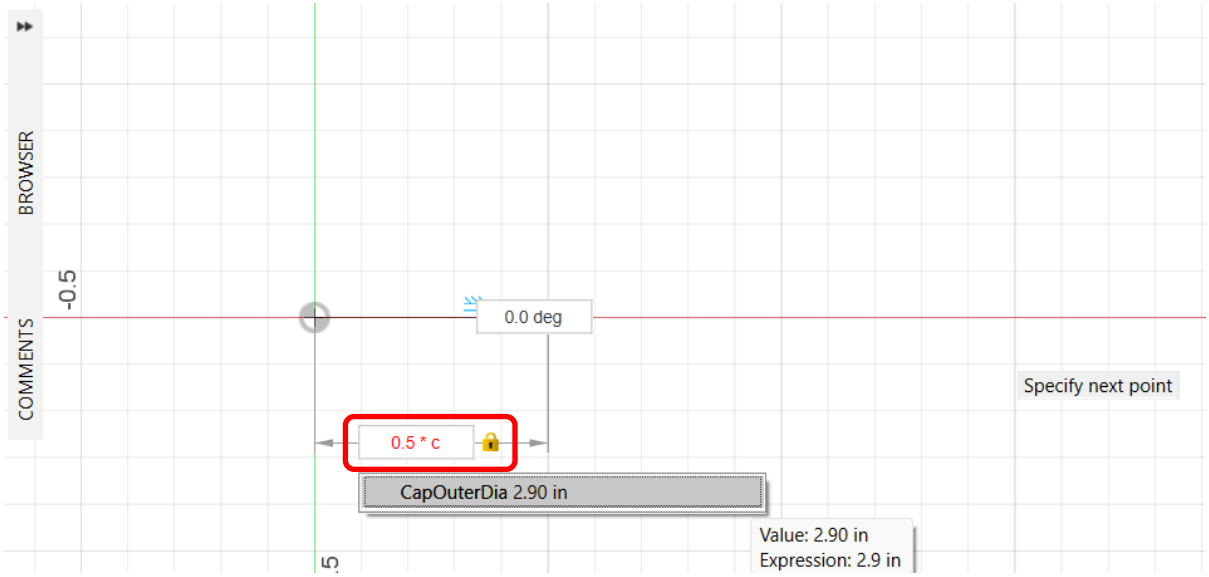
- click on the top **+** icon
- set the **Name** to **CapOuterDia**, the **Expression** to **2.9**, and click **OK**
- click **OK** on the PARAMETERS window also



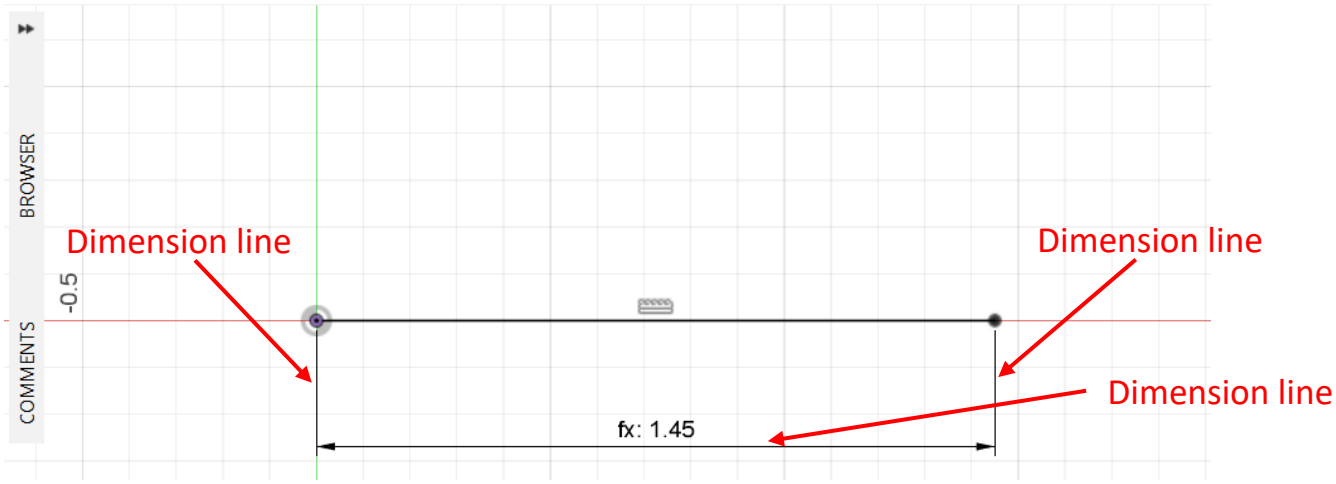
- zoom (using the mouse wheel) and pan (holding the mouse wheel down) to achieve a view similar to that below
- select the **Line** tool
- click on the **Origin** and **extend the line to the right**, but don't click a again



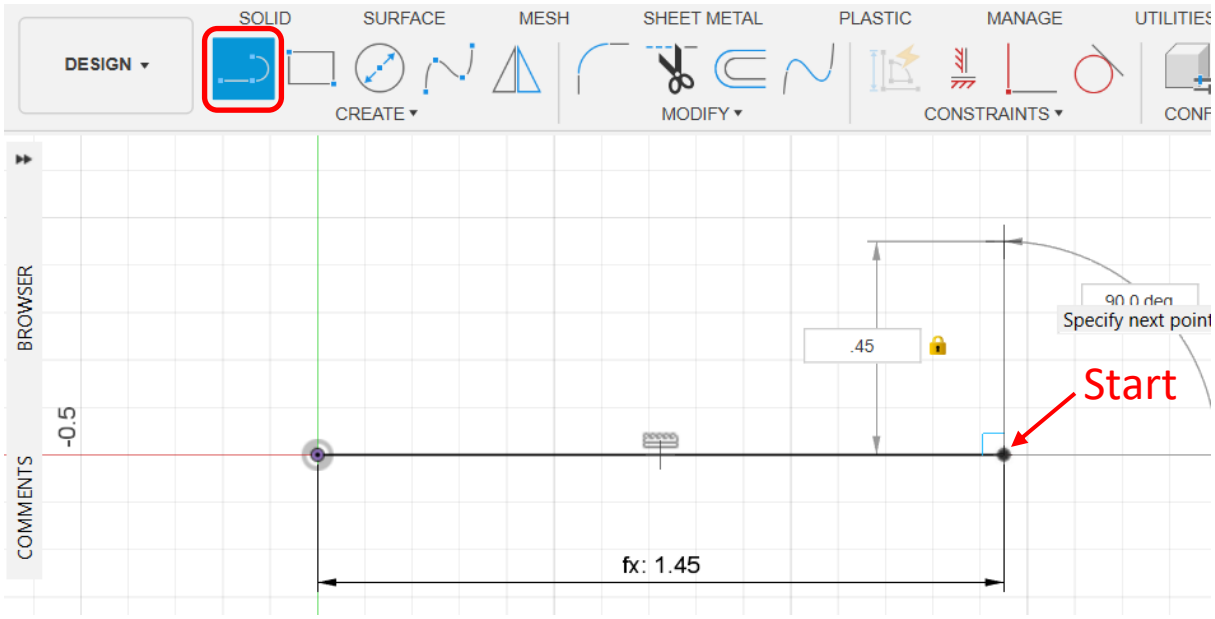
- type **0.5 * c** and press the **Enter key** to select **CapOuterDia**



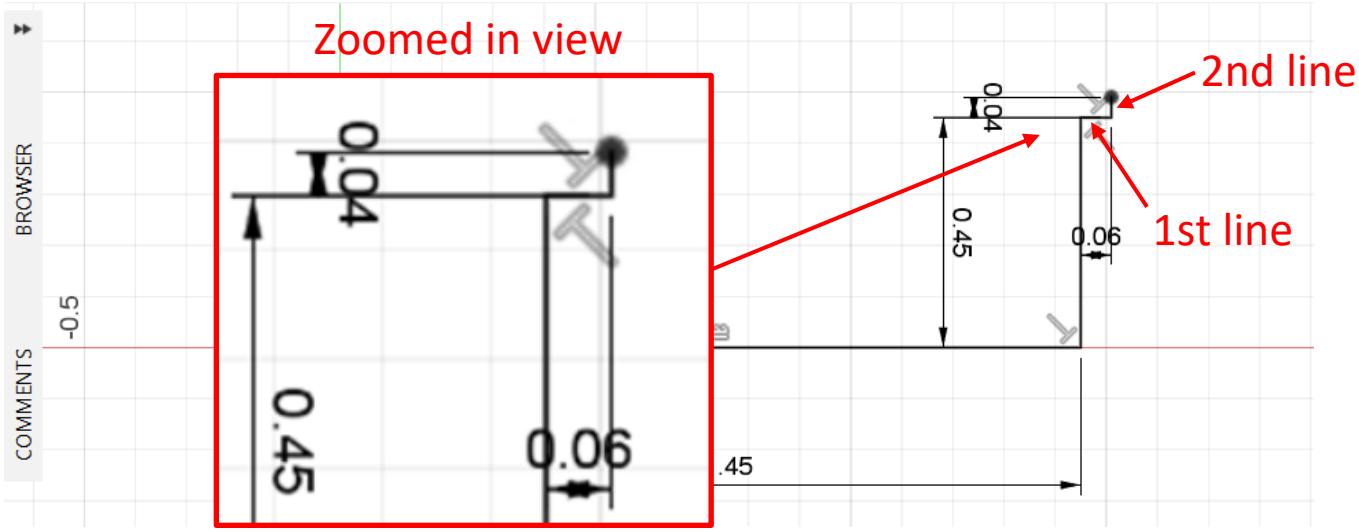
The result should look like that below. The thin lines are Dimension lines and it is OK if they look different.



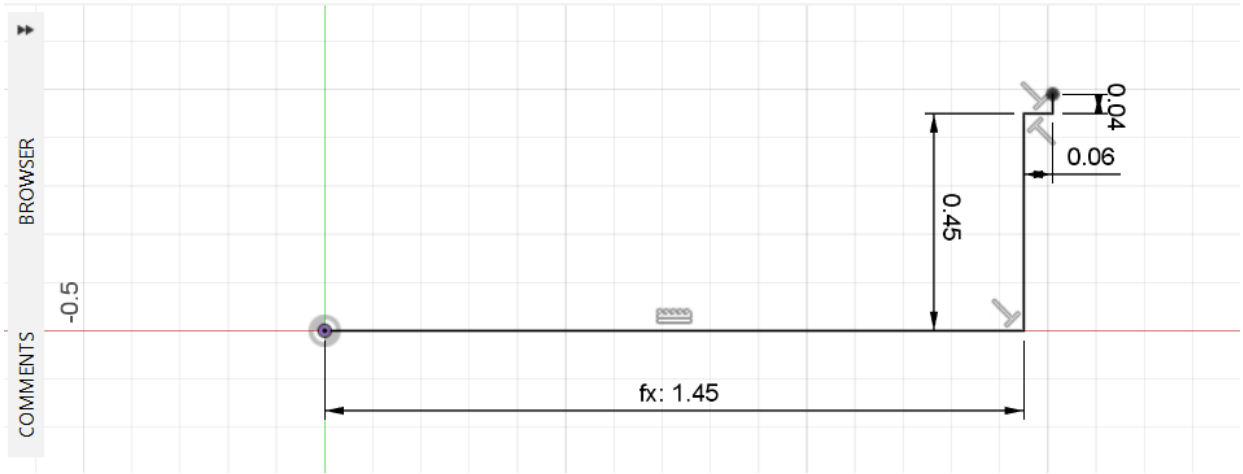
- select the **Line** tool again
- start a line from the **end of the line just created**, extend it upward, type **0.45**, and press the **Enter key**



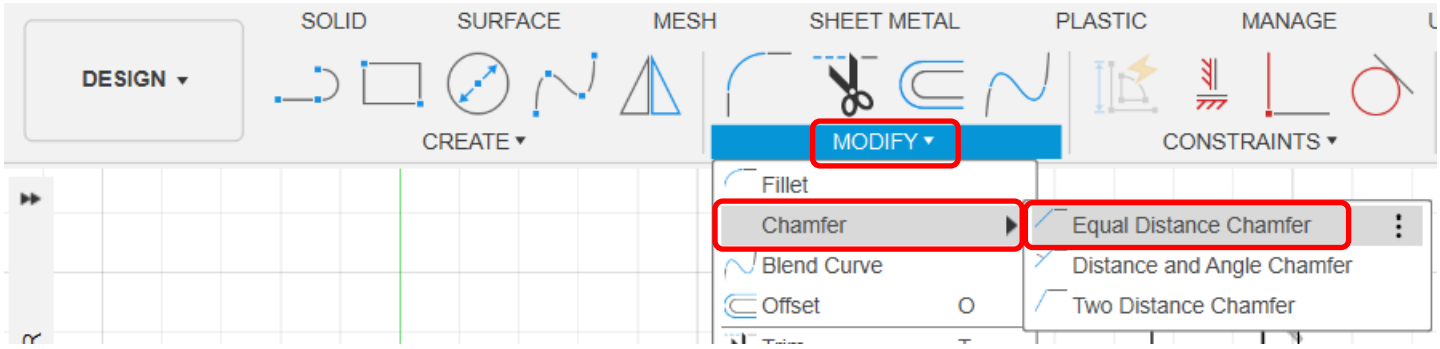
- using the **Line** tool create a line from the **top of the last line** and to the **right by 0.06** and then **upward by 0.04**



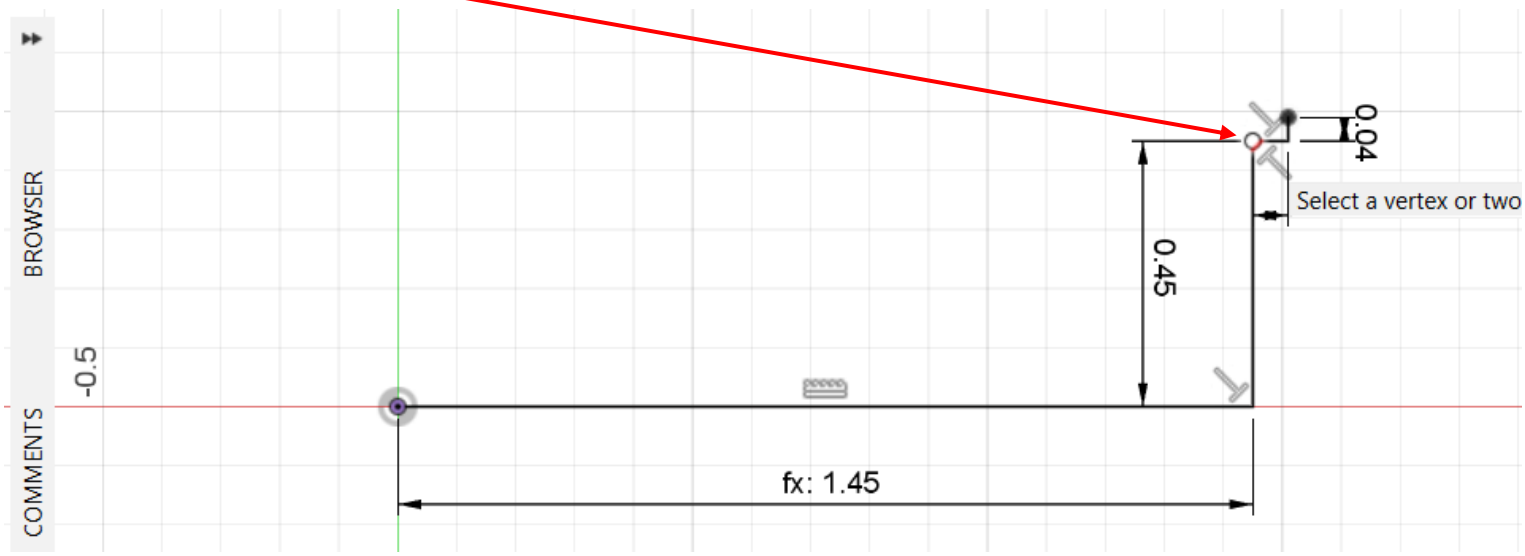
- drag the Dimension values to make them more visible, as shown below



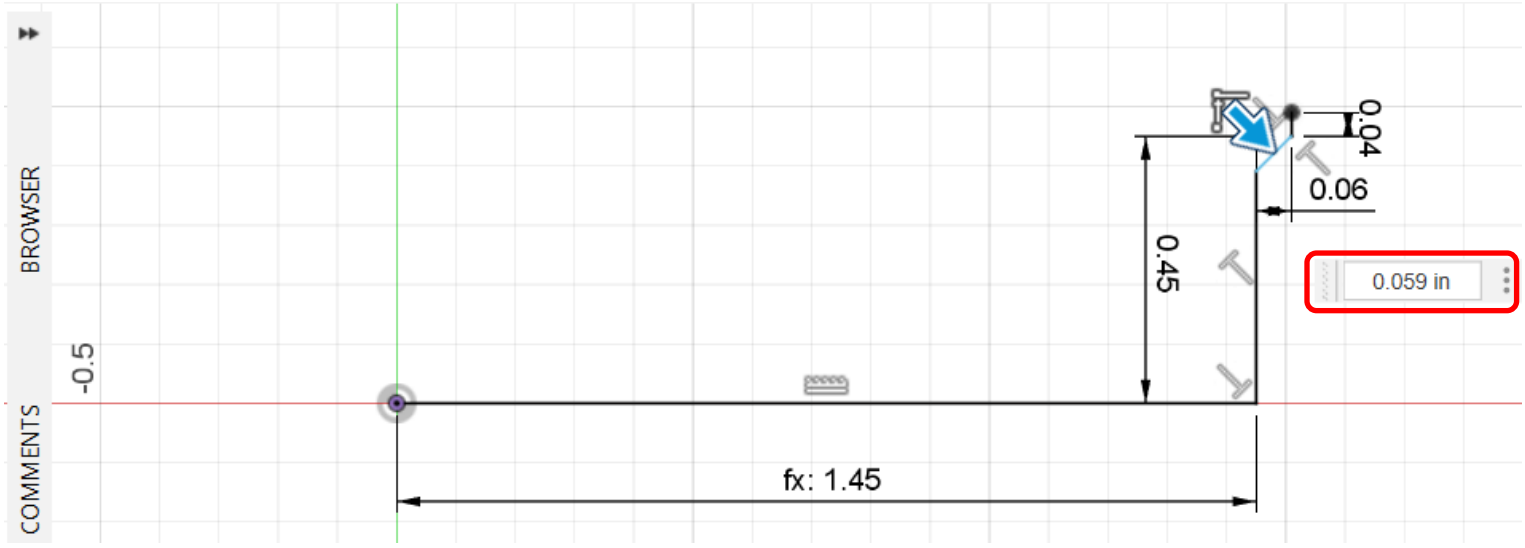
- from the **MODIFY** menu, select **Equal Distance Chamfer**



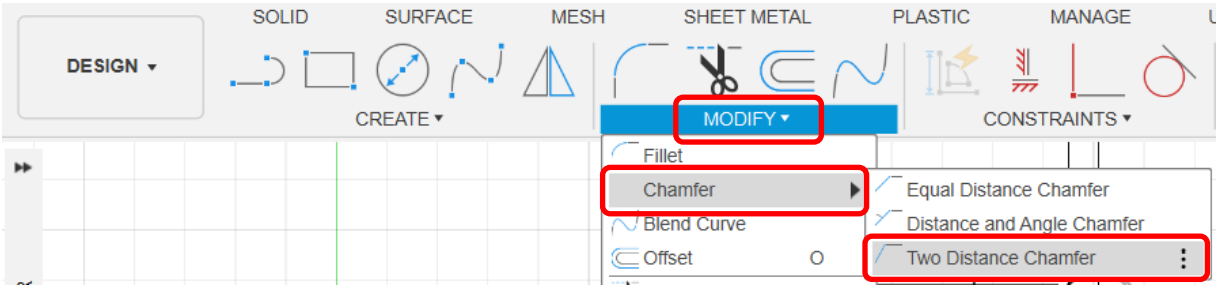
- click on the **corner** indicated



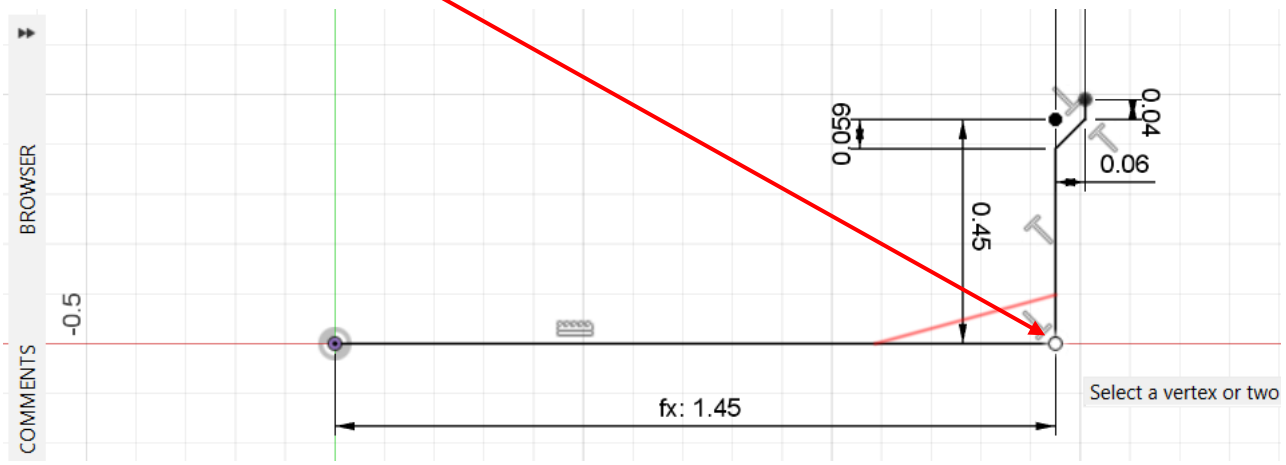
- enter a value of **0.059**. You can ignore any warning messages.



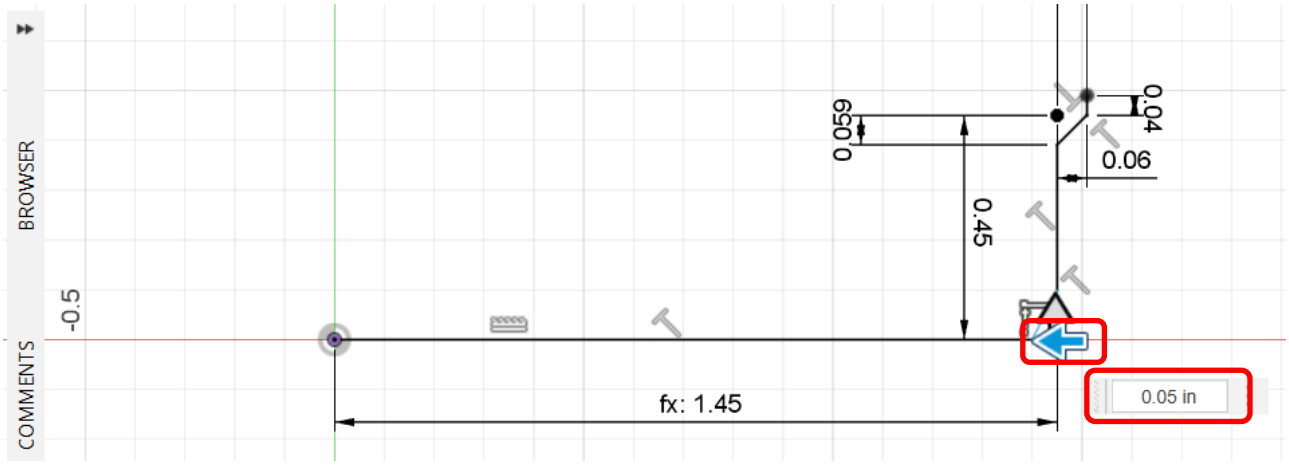
- from the **MODIFY** menu, select **Two Distance Chamfer**



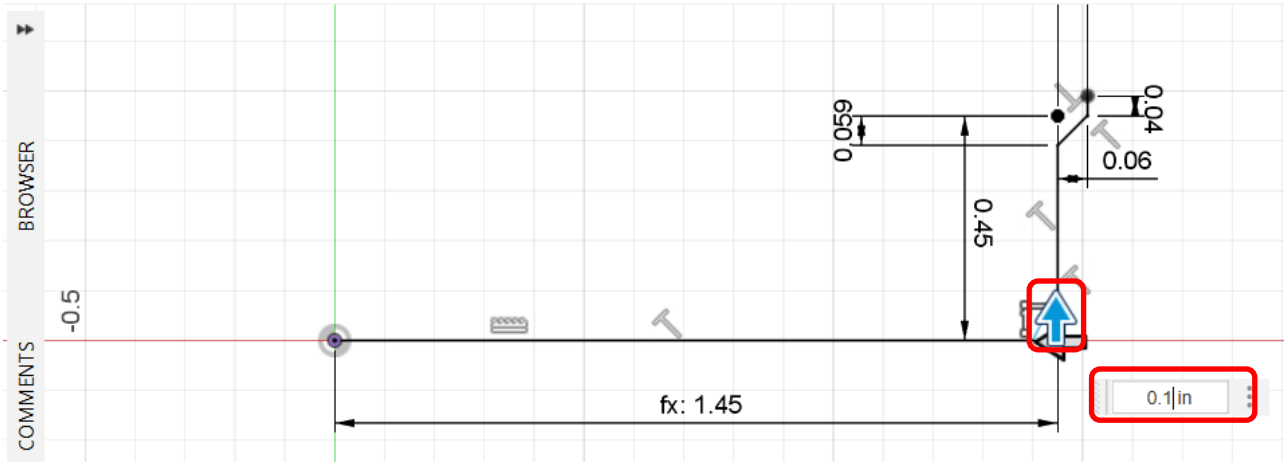
- click on the **bottom right corner**



- select the **horizontal blue arrow** and enter **0.05**

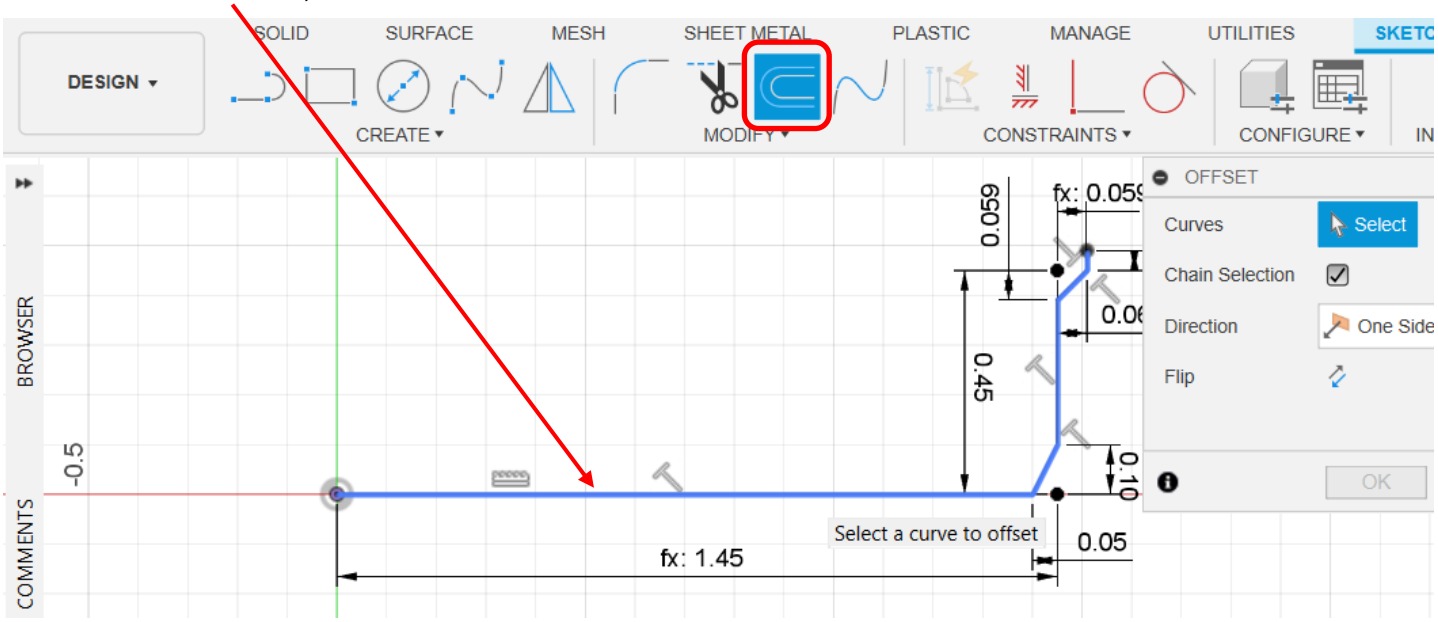


- select the **vertical blue arrow** and enter **0.1**

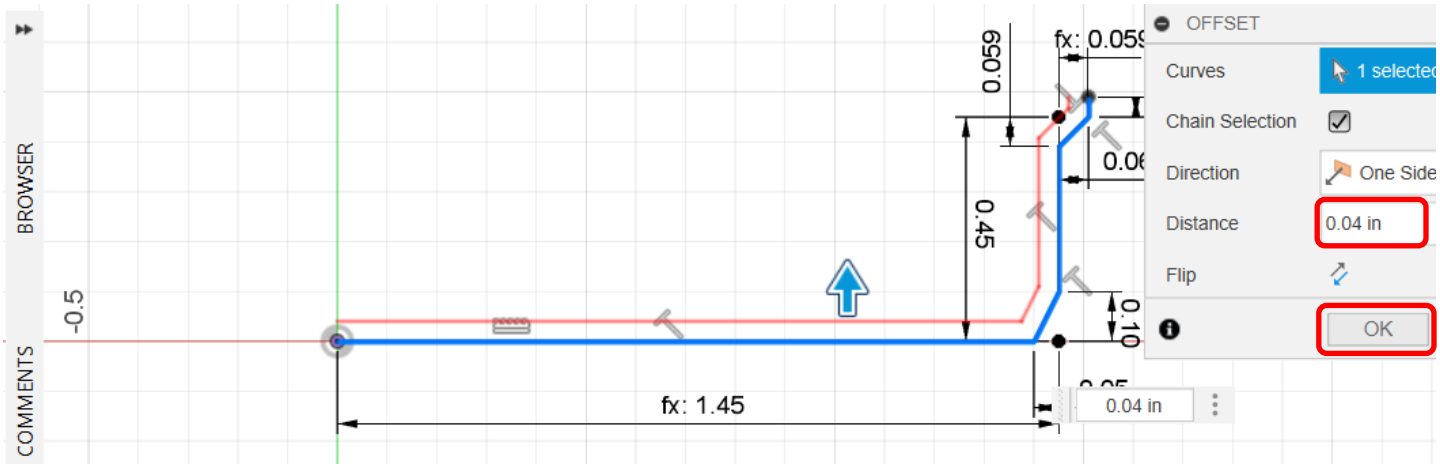


- result after some cleanup of Dimensions

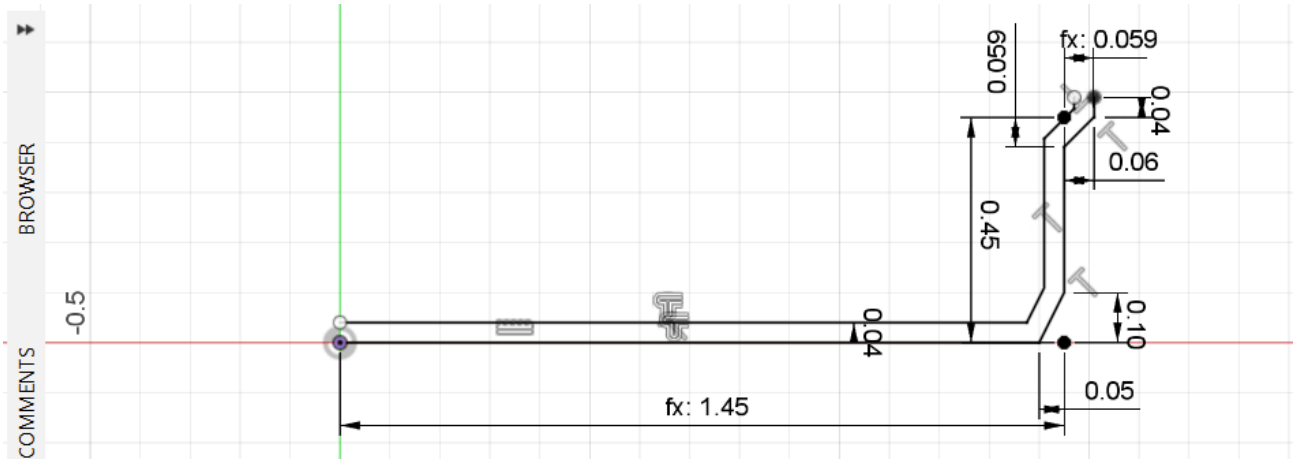
- click on the **Offset** tool. If it is not visible, find it in the **MODIFY** menu.
- click on the **bottom line**, which will turn it blue



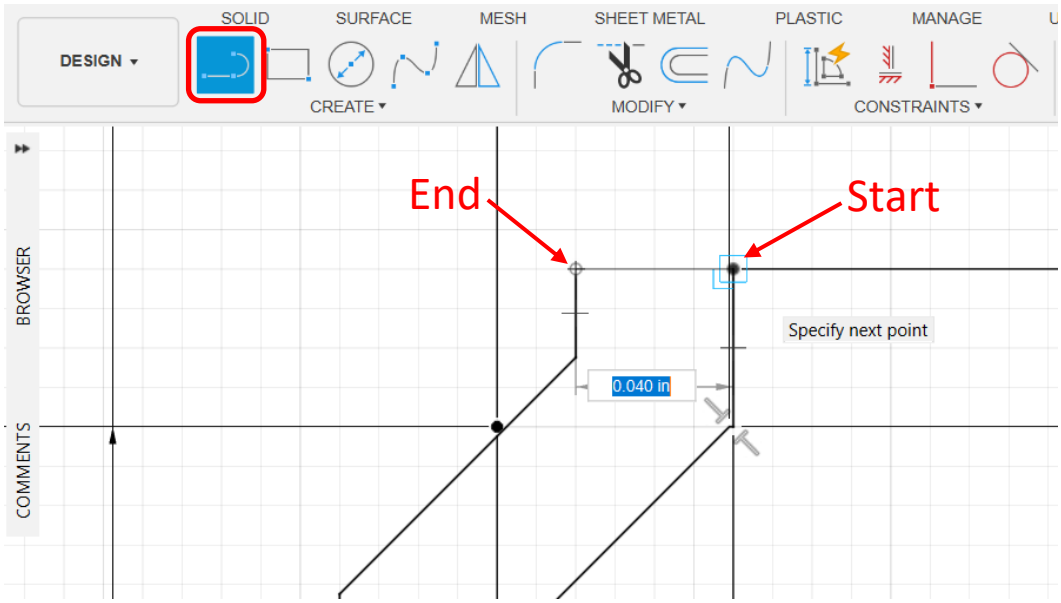
- for **Distance** enter **0.04**. Red lines should appear as shown below. If the red line is on the opposite side of the blue lines, click the Flip icon.
- click **OK**



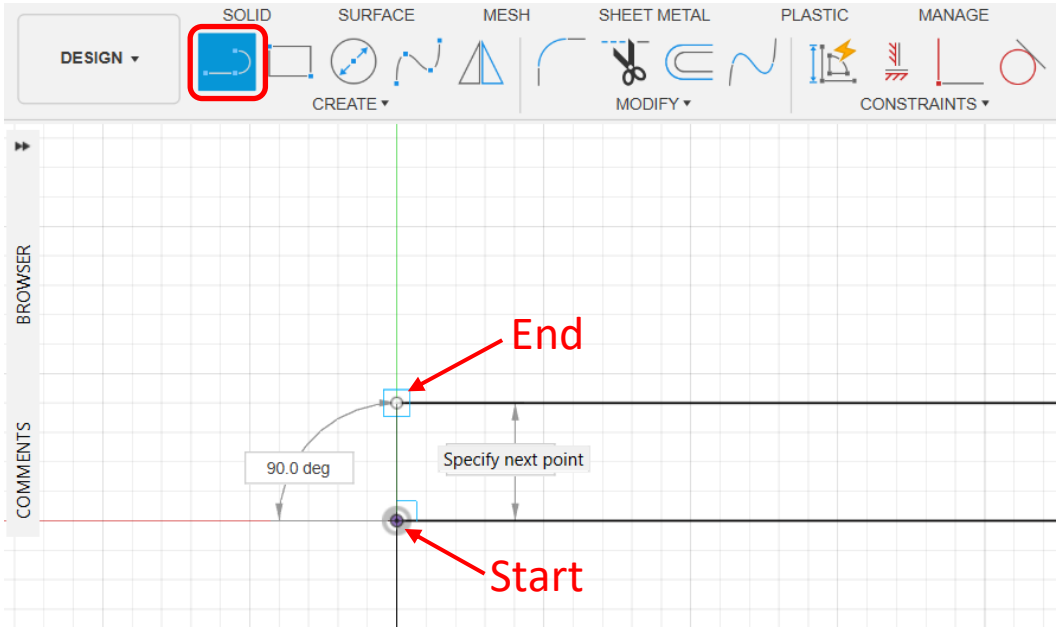
This should be the result.



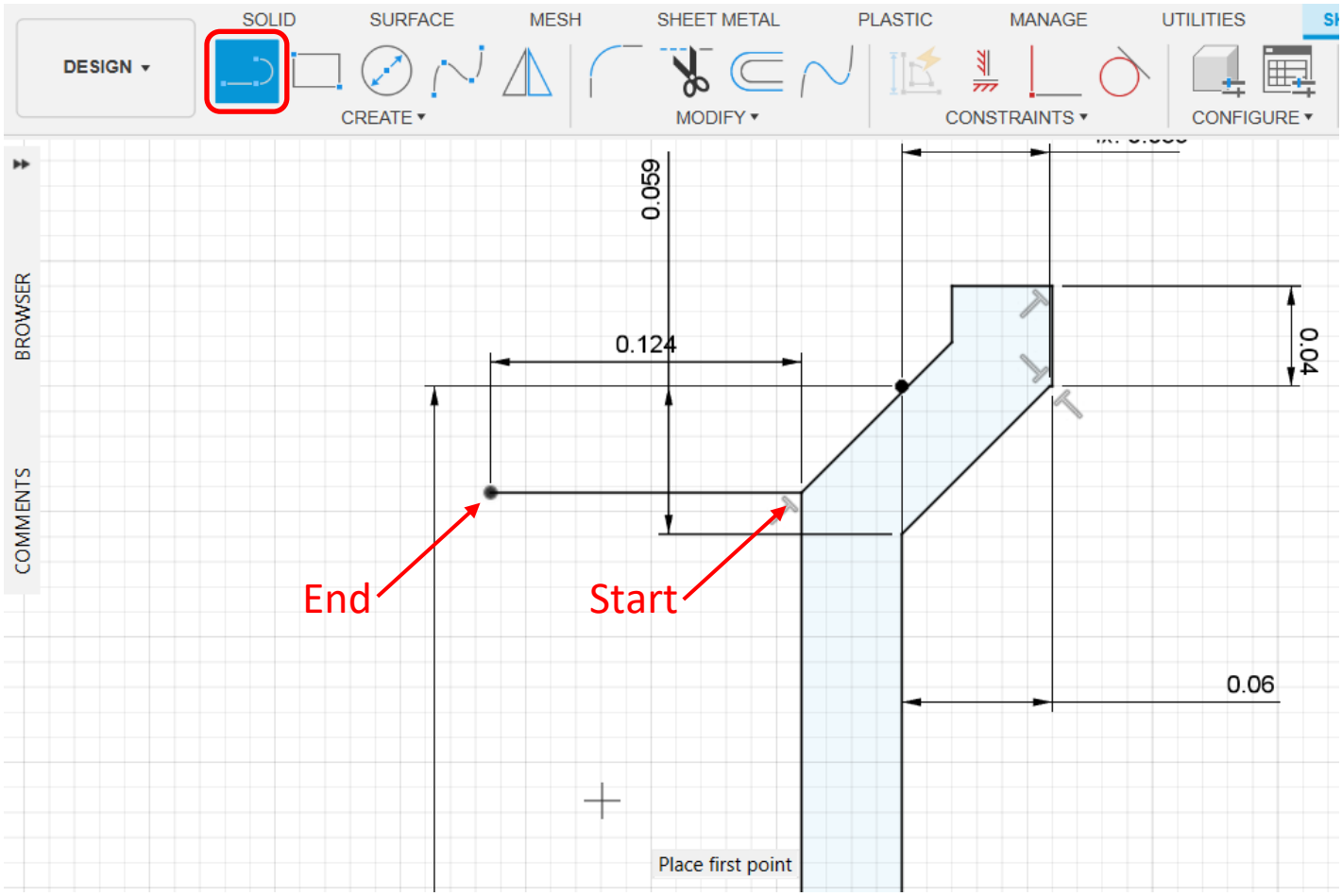
- zoom into the top right of the shape and create a line from the **point indicated** to the **top of the adjacent line**. No value needs to be entered.



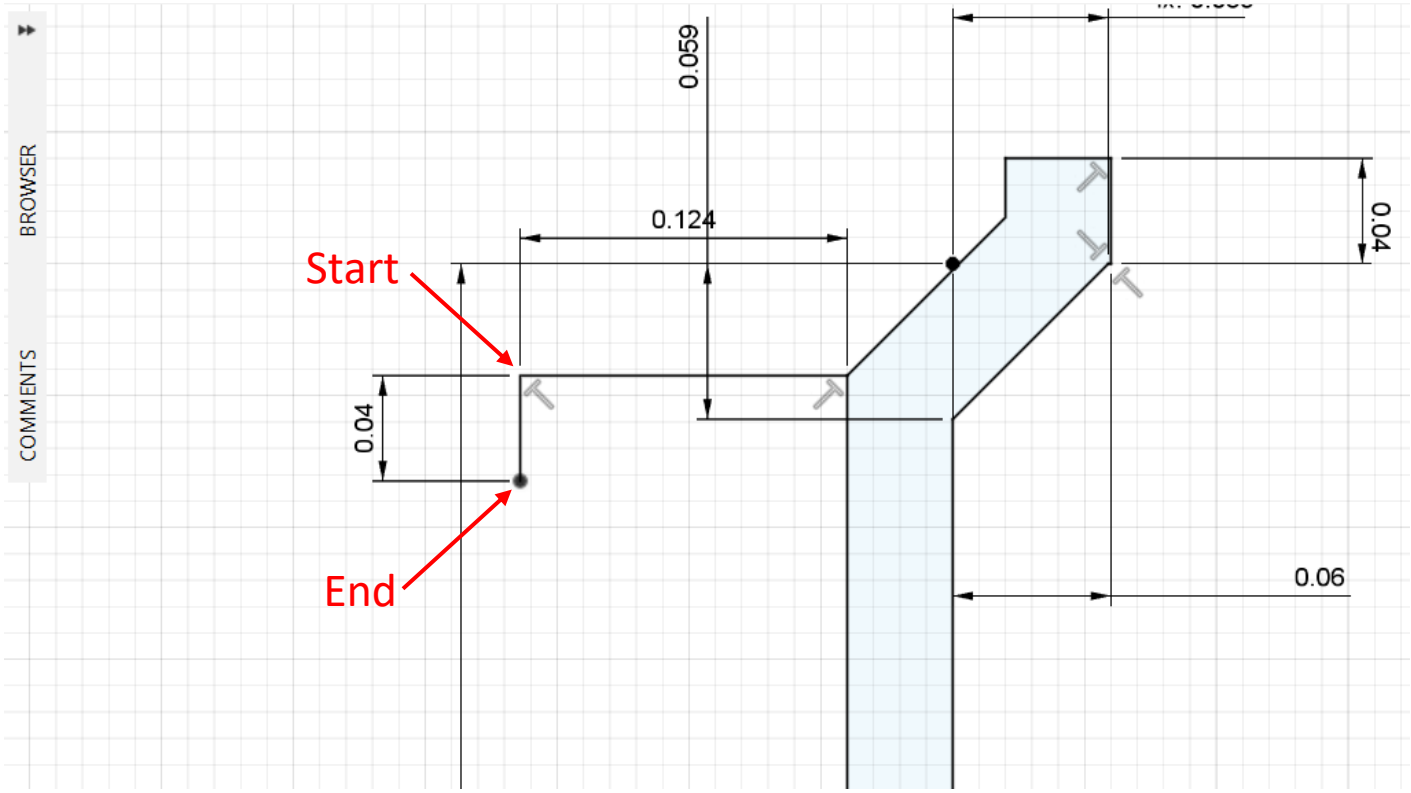
- zoom into the bottom left of the shape and create **a line between the 2 endpoints**. No value needs to be entered.



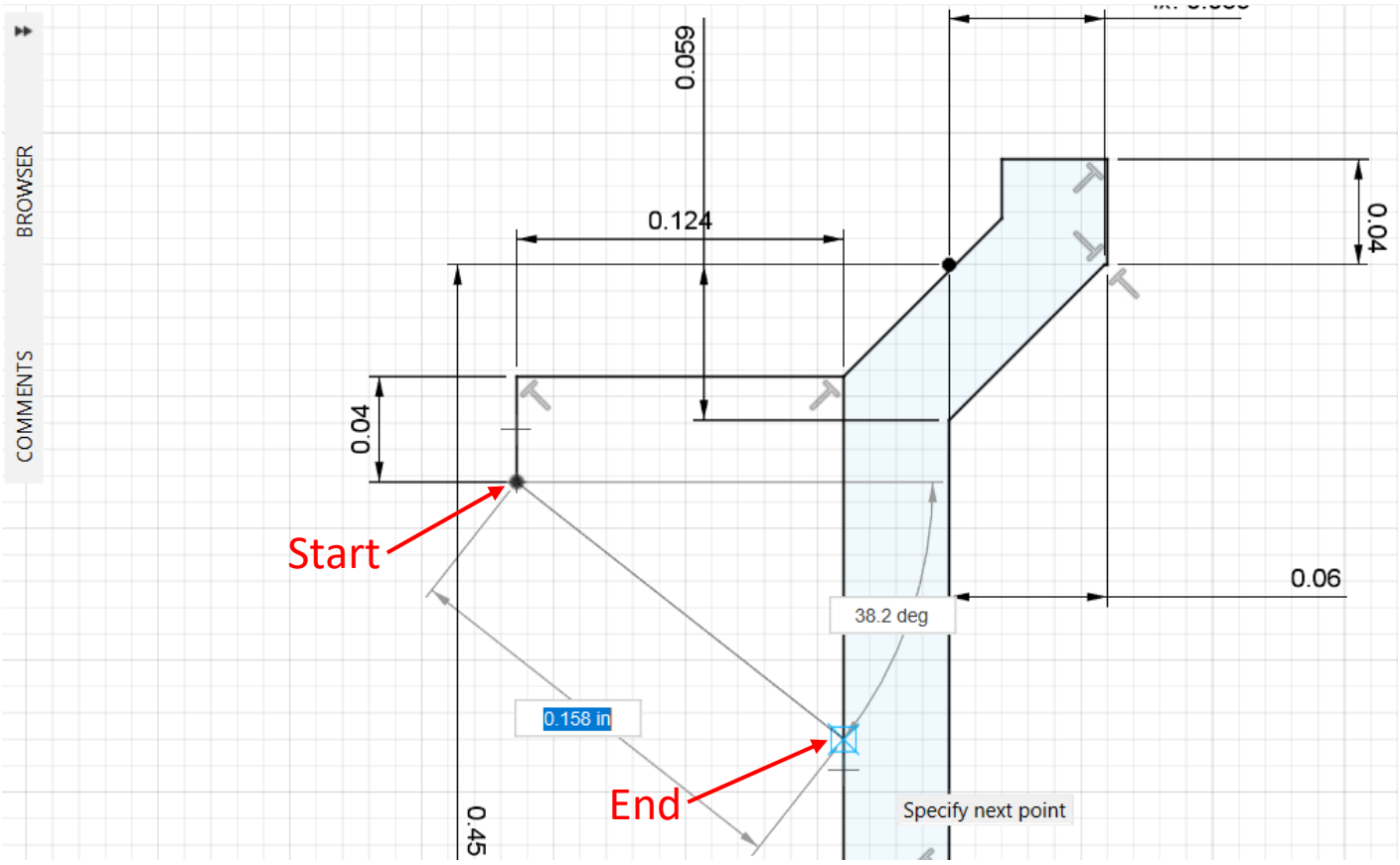
- zoom into the top right of the shape again
- create a line from the **corner indicated** and to the left by **0.124**



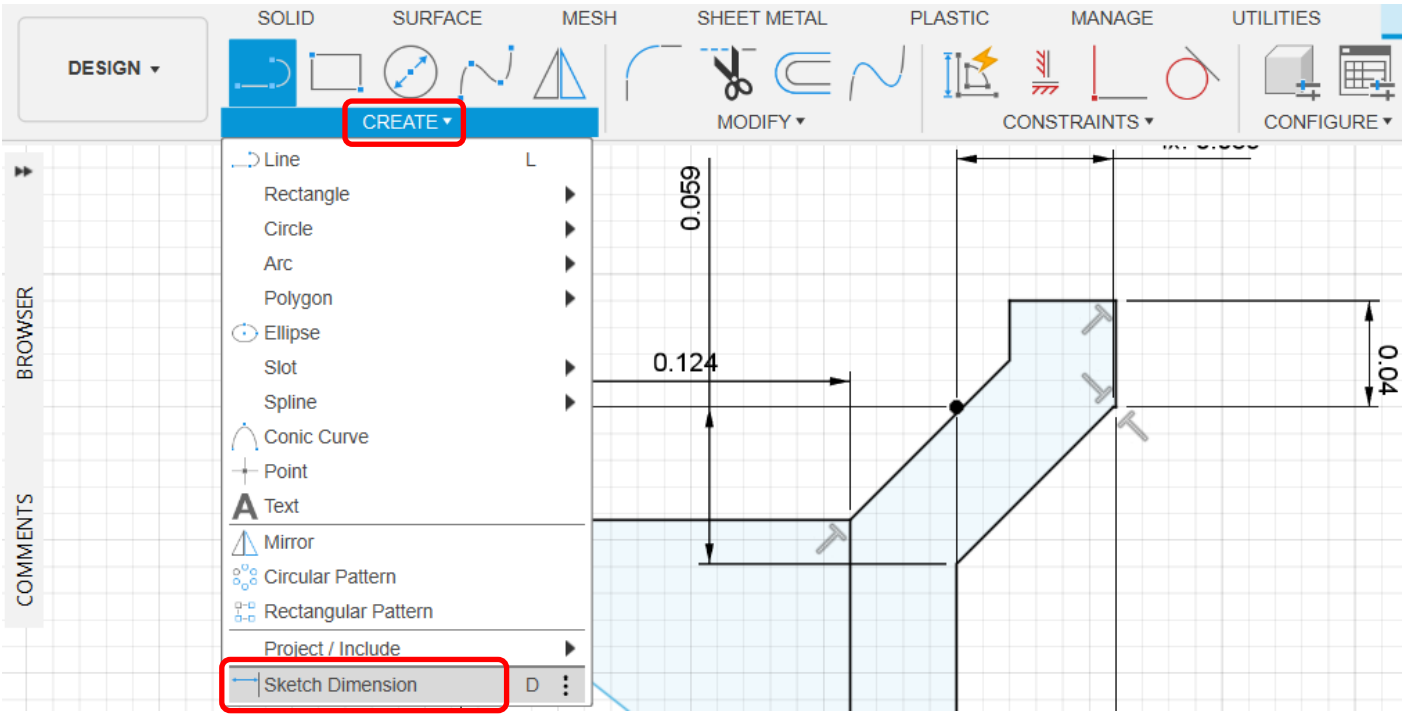
- create another line downward by **0.04**



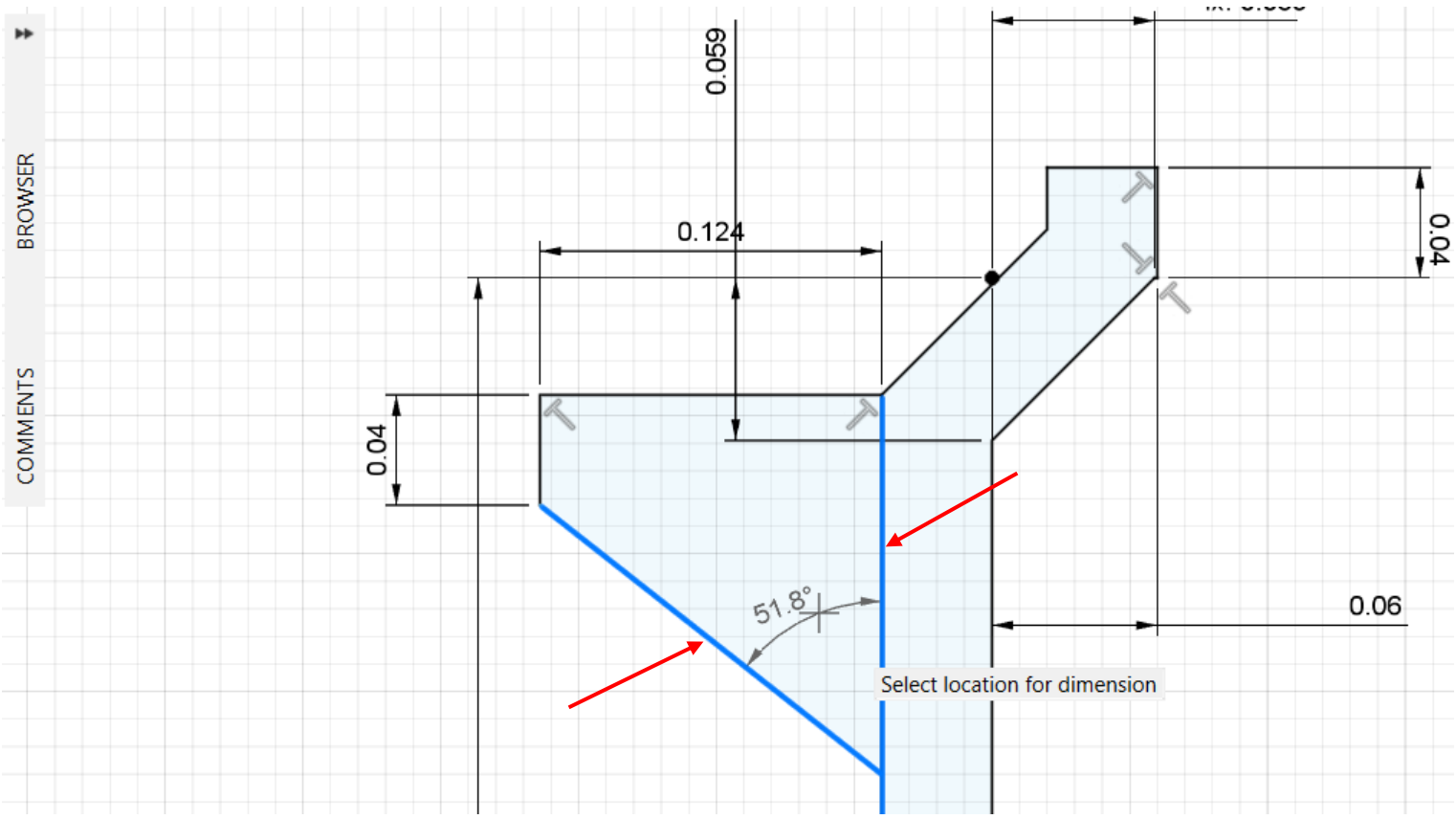
- draw a line **diagonally down** and click when it **snaps to the vertical line**. The length and angle can be different for that shown.



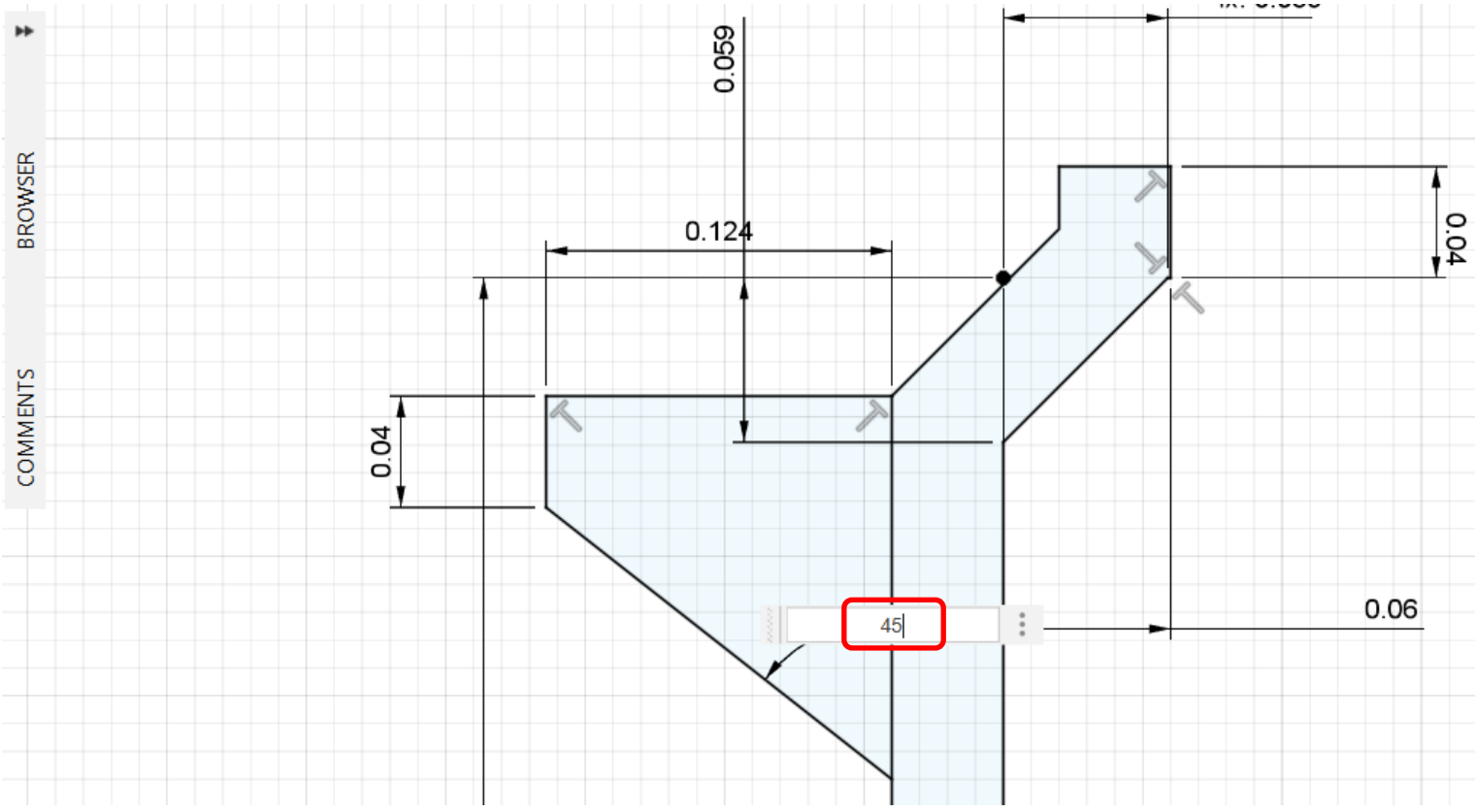
- from the **CREATE** menu select **Sketch Dimension** at the bottom of the menu



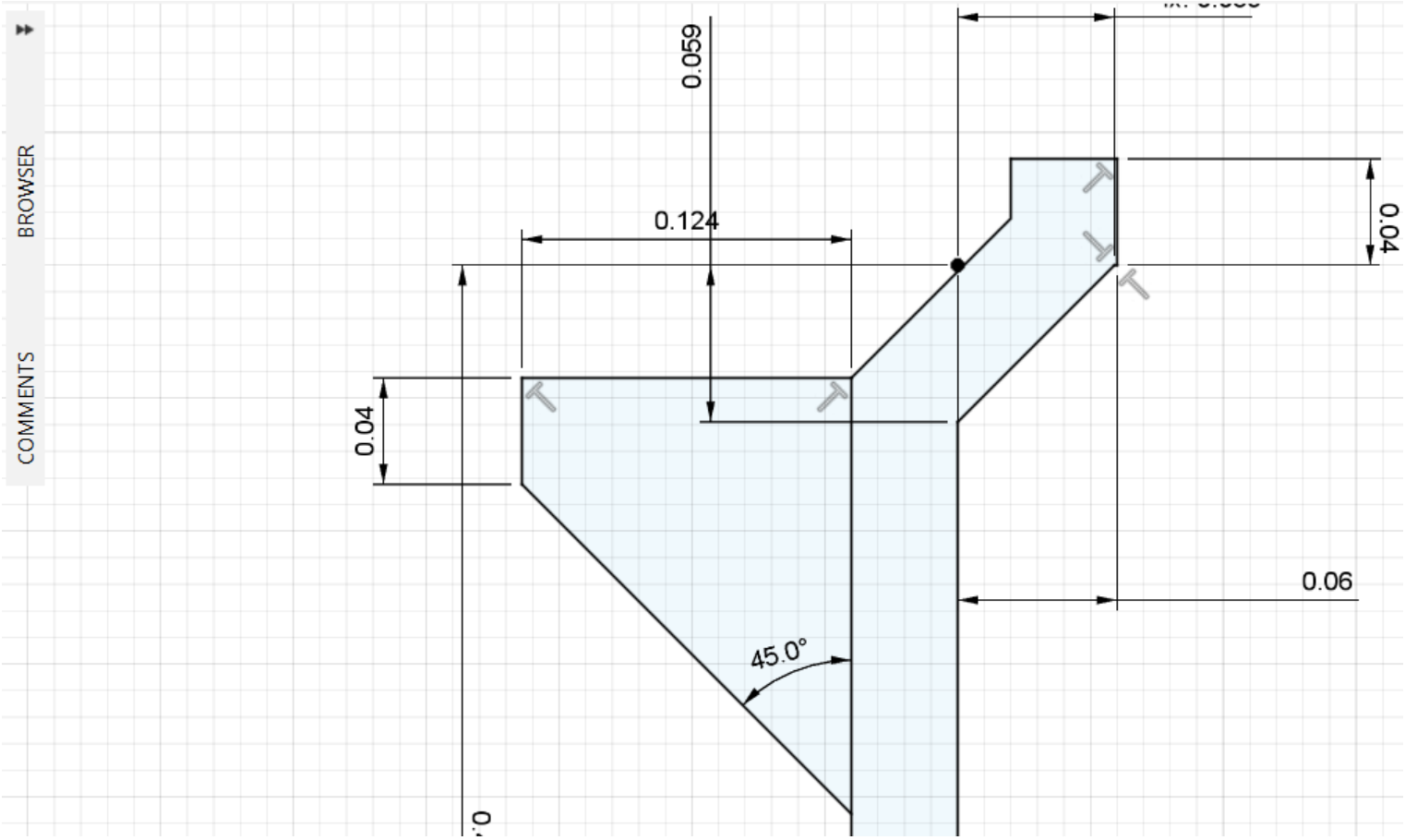
- click on the **vertical line** and then click on the **diagonal line**, which will turn them blue



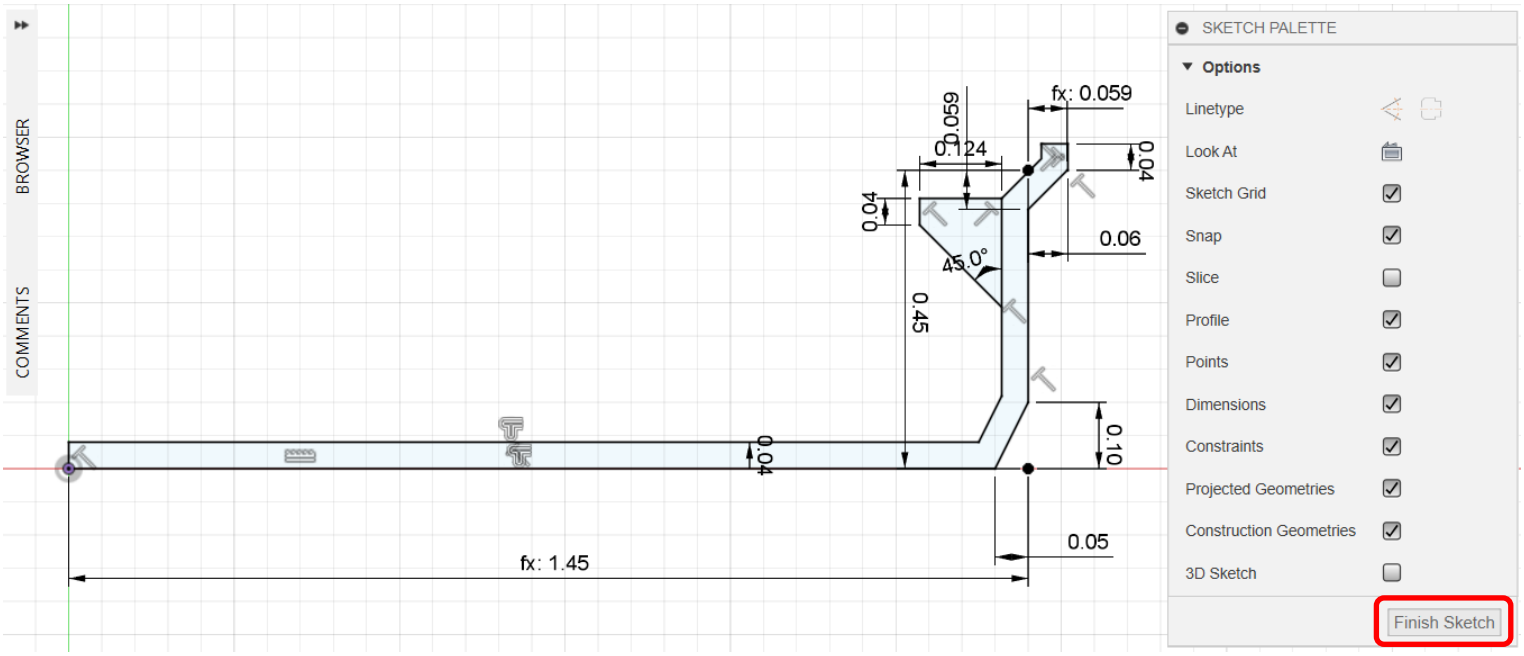
- enter **45**



The result should look like that below. It is OK if the Dimension lines look different.



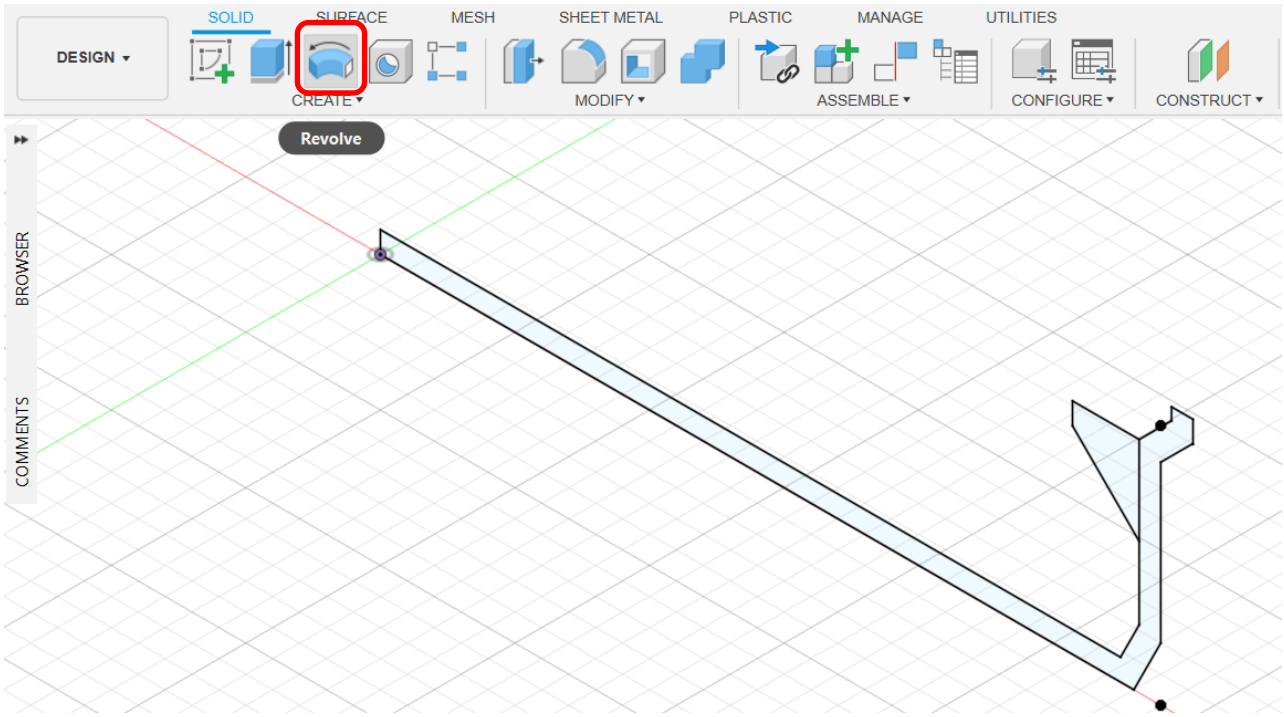
- zoom out to admire your Sketch
- click **Finish Sketch**



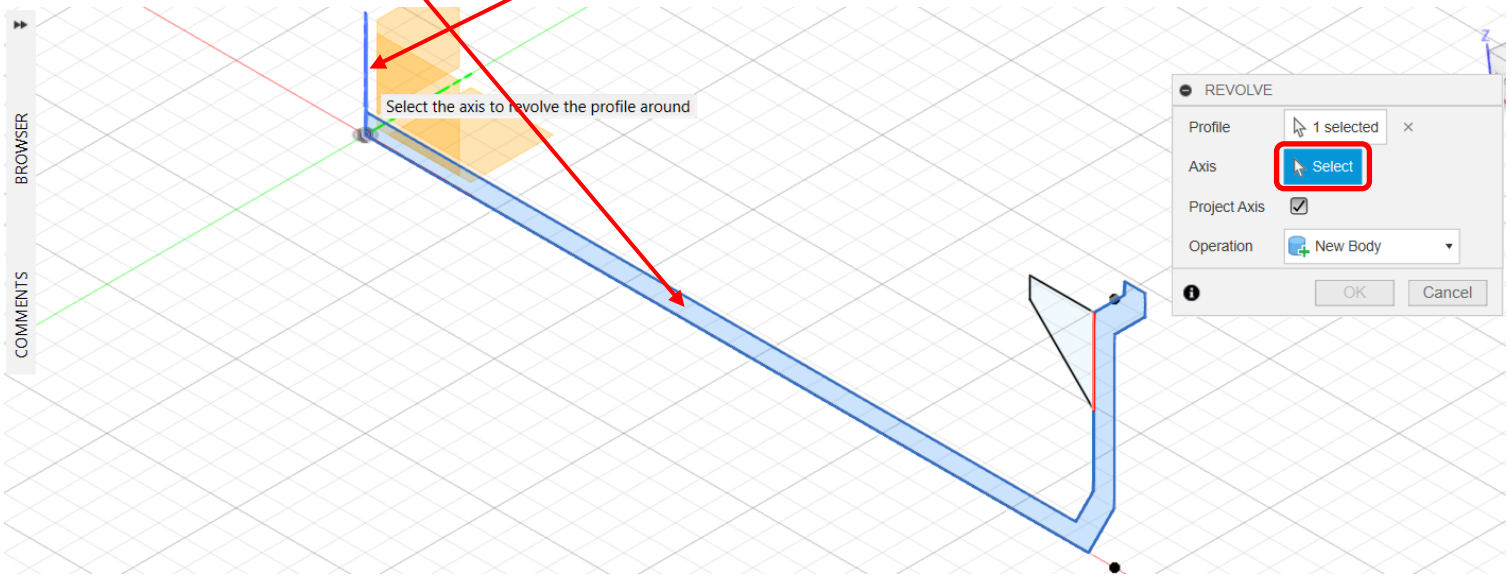
Performing a Revolve Operation

The Revolve operation creates a solid body from a profile by revolving it around an axis. This is why we used the radius for the size of the profile instead of the diameter.

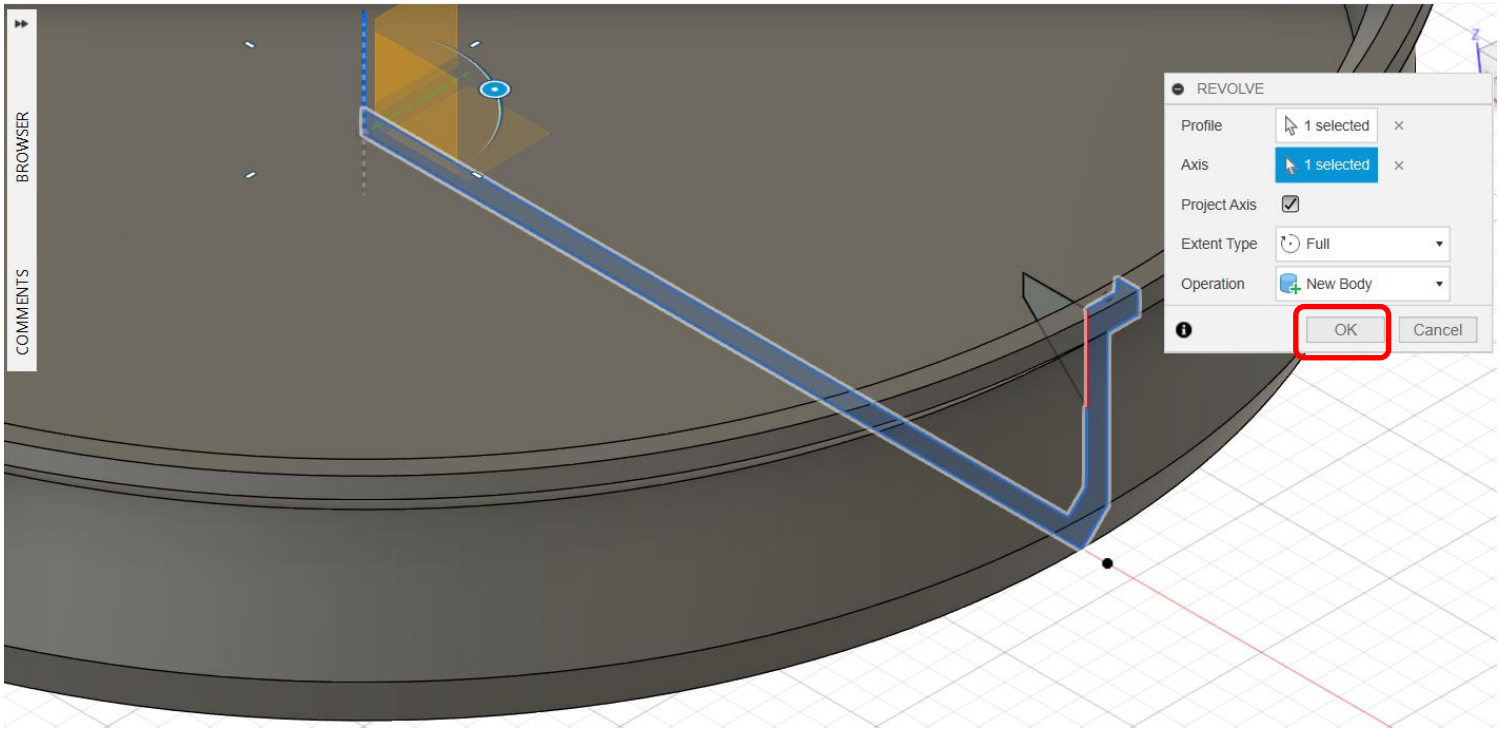
- click the **Home** icon at the **View Cube** and zoom into the profile as shown below
- select the **Revolve** tool. If it is not visible, find it in the **CREATE** menu.



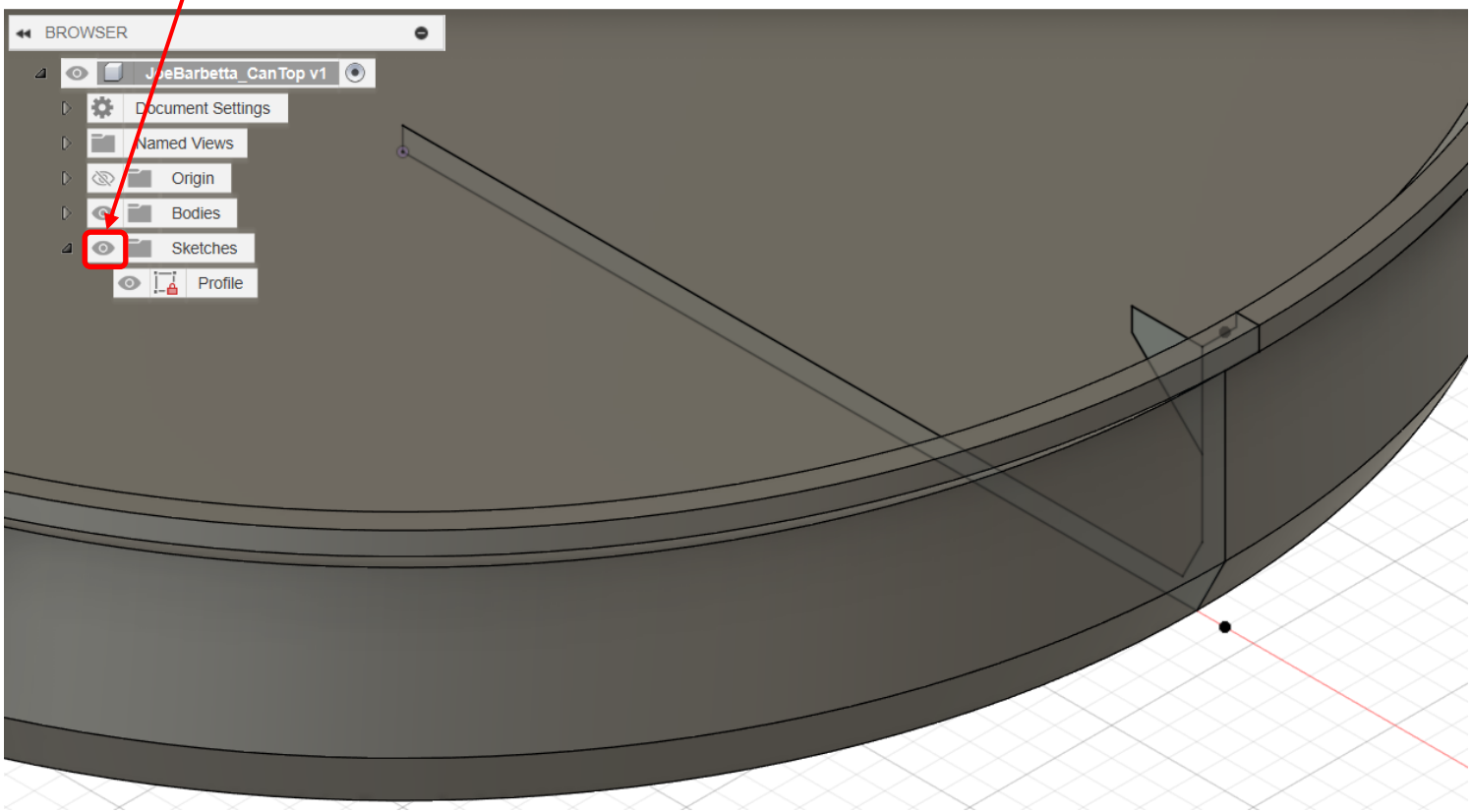
- click on the **bottom of the profile**, which should turn it darker blue. Note that the smaller area is not being selected.
- click on **Select** for **Axis** and click on the **vertical axis**



- yell “**That was neat!”** and click **OK**



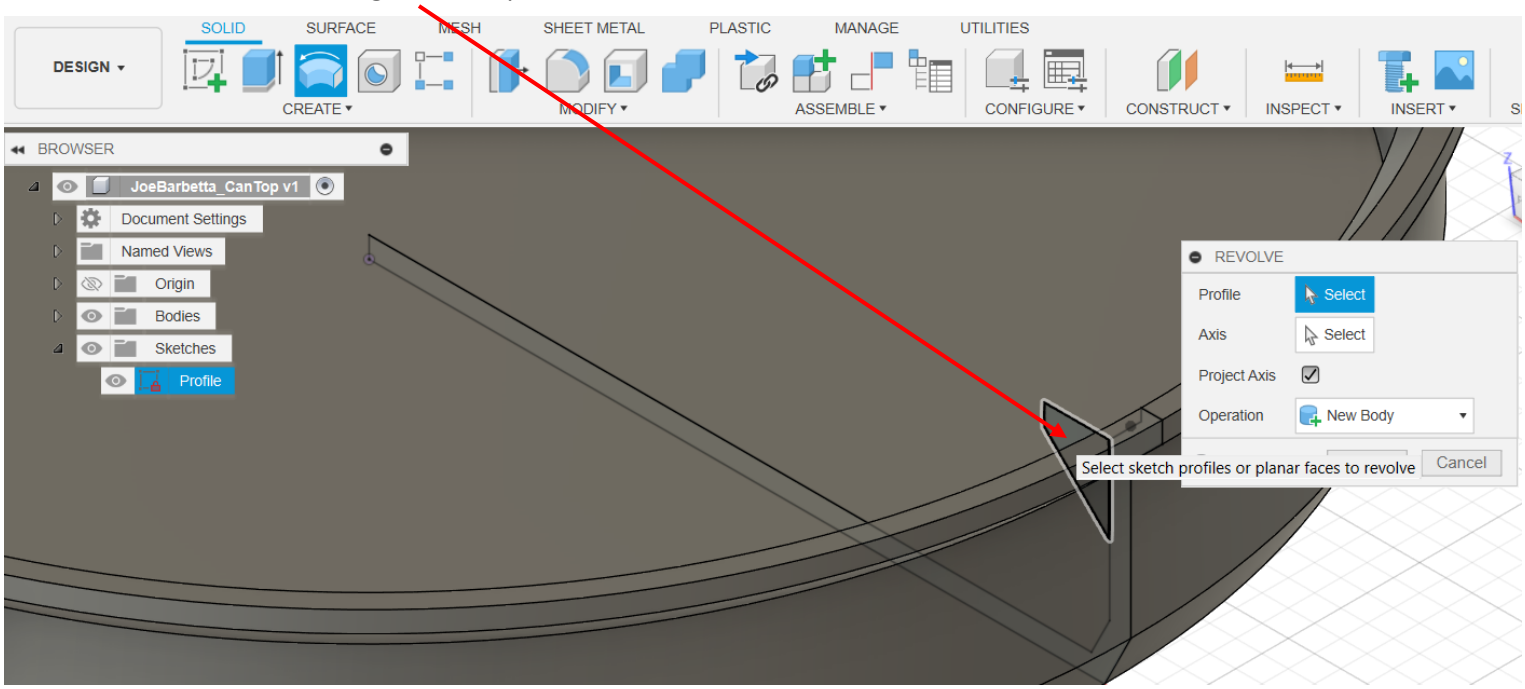
- click on the **double-arrow** to open the BROWSER
- click on the **eye** icon for the **Profile** Sketch to make it visible again



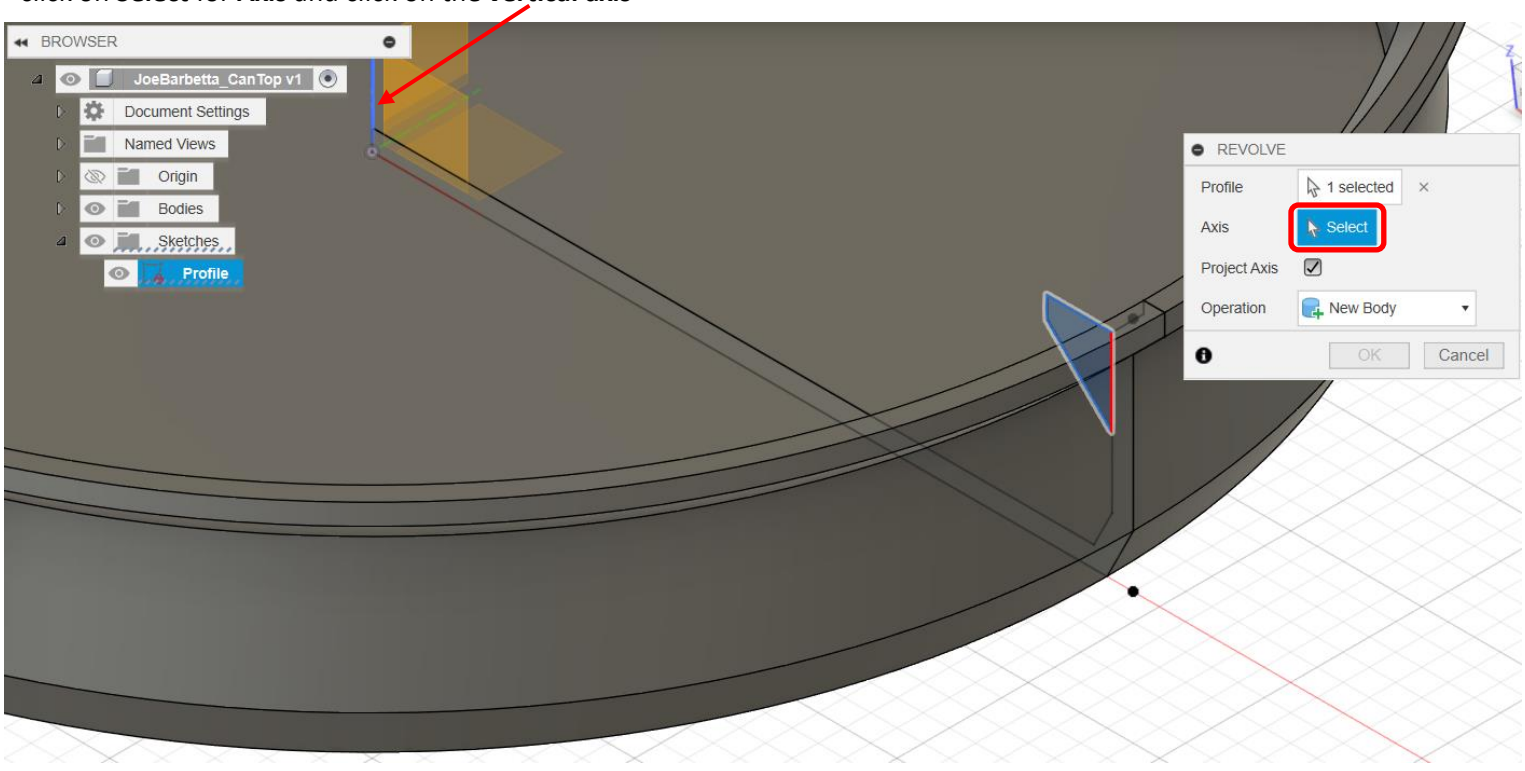
Performing a Partial Revolve Operation

By default a Revolve is performed over a complete 360 degree circle. (2 pi for math people) Here we will perform a partial revolve where we can define an angle smaller than 360.

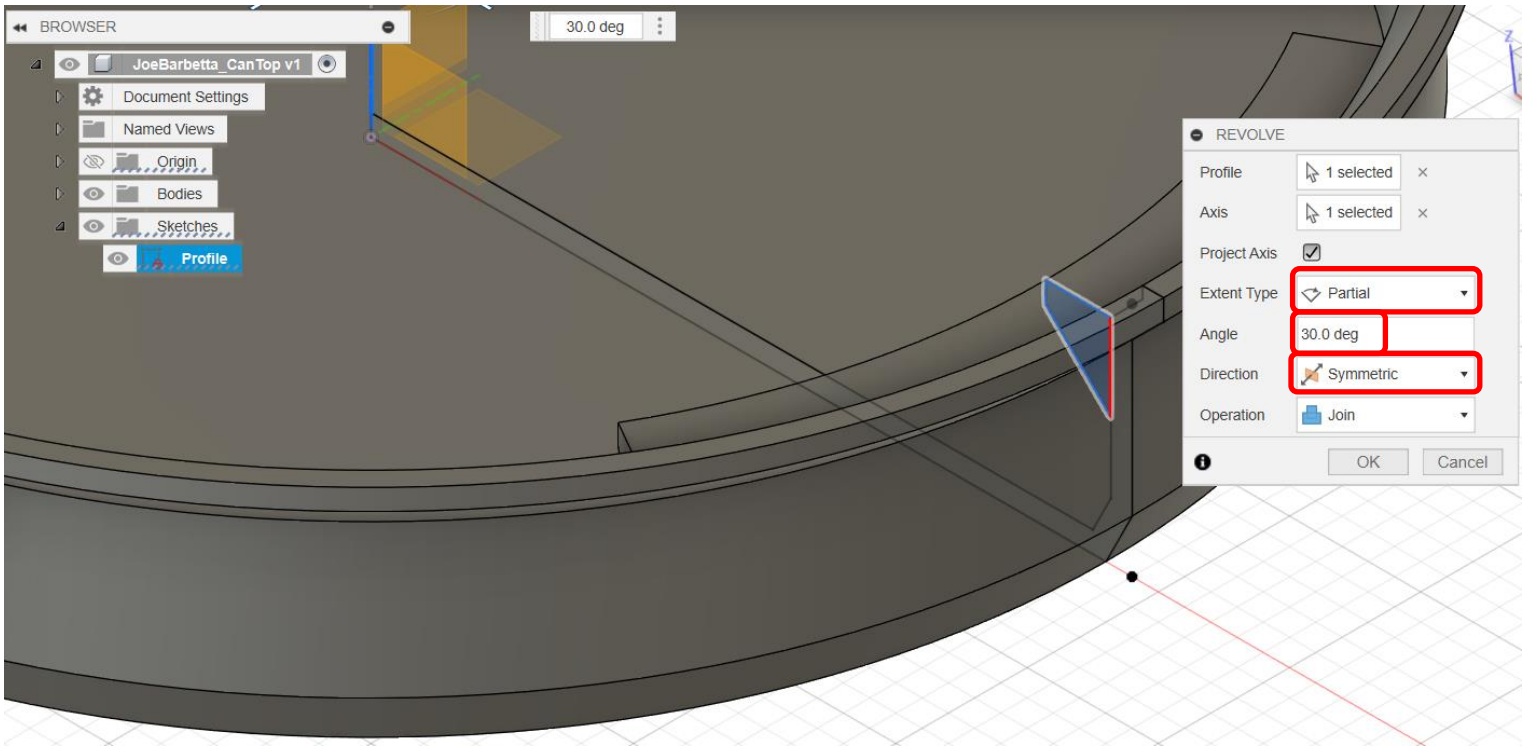
- select the **Revolve** tool again
- this time click on the **small region** of the profile



- click on **Select** for **Axis** and click on the **vertical axis**

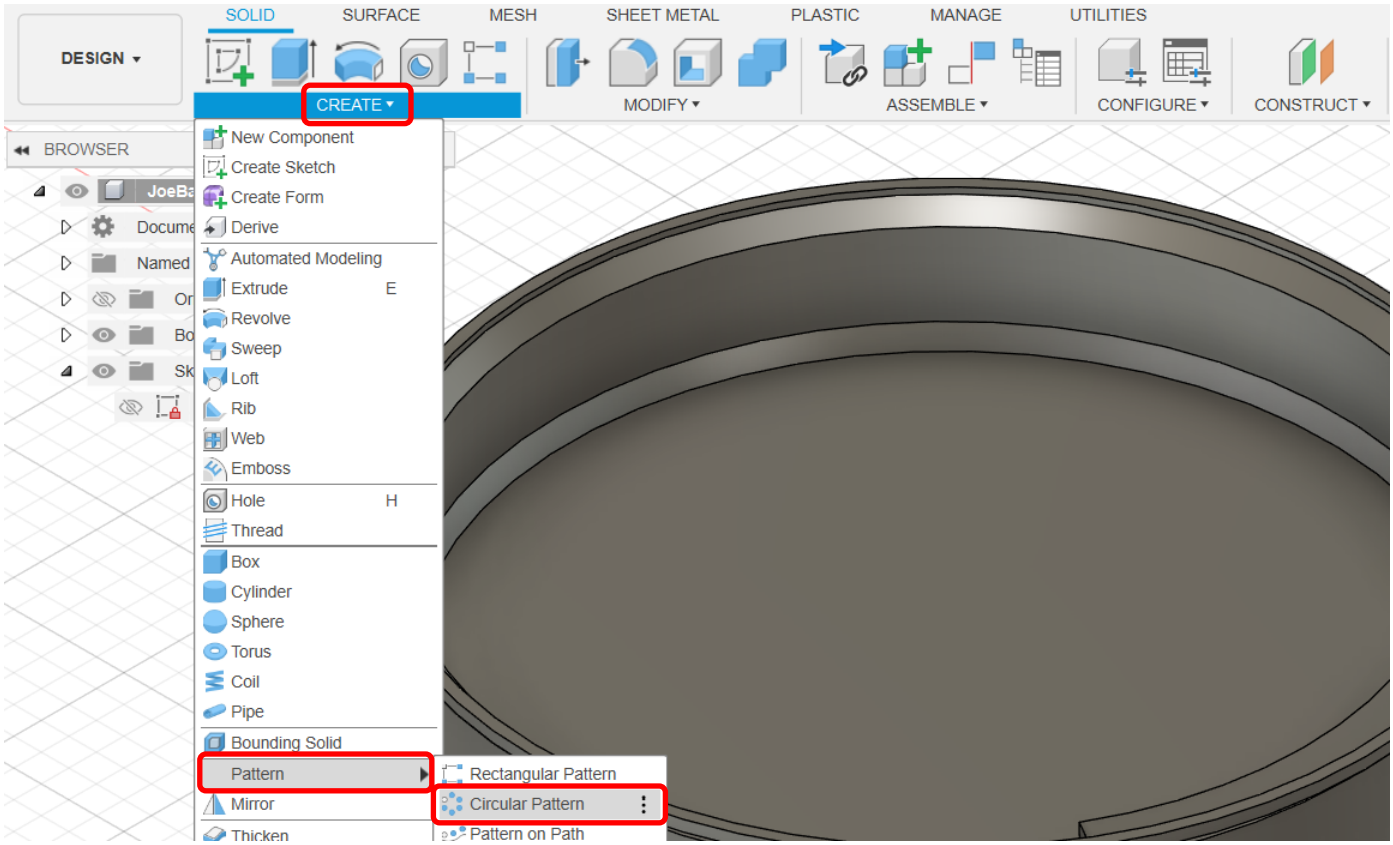


- for **Extent Type** select **Partial**
- for **Direction** select **Symmetric** and for **Angle** set the value to **30.0**
- yell **“That was almost as neat as the full revolve!”**

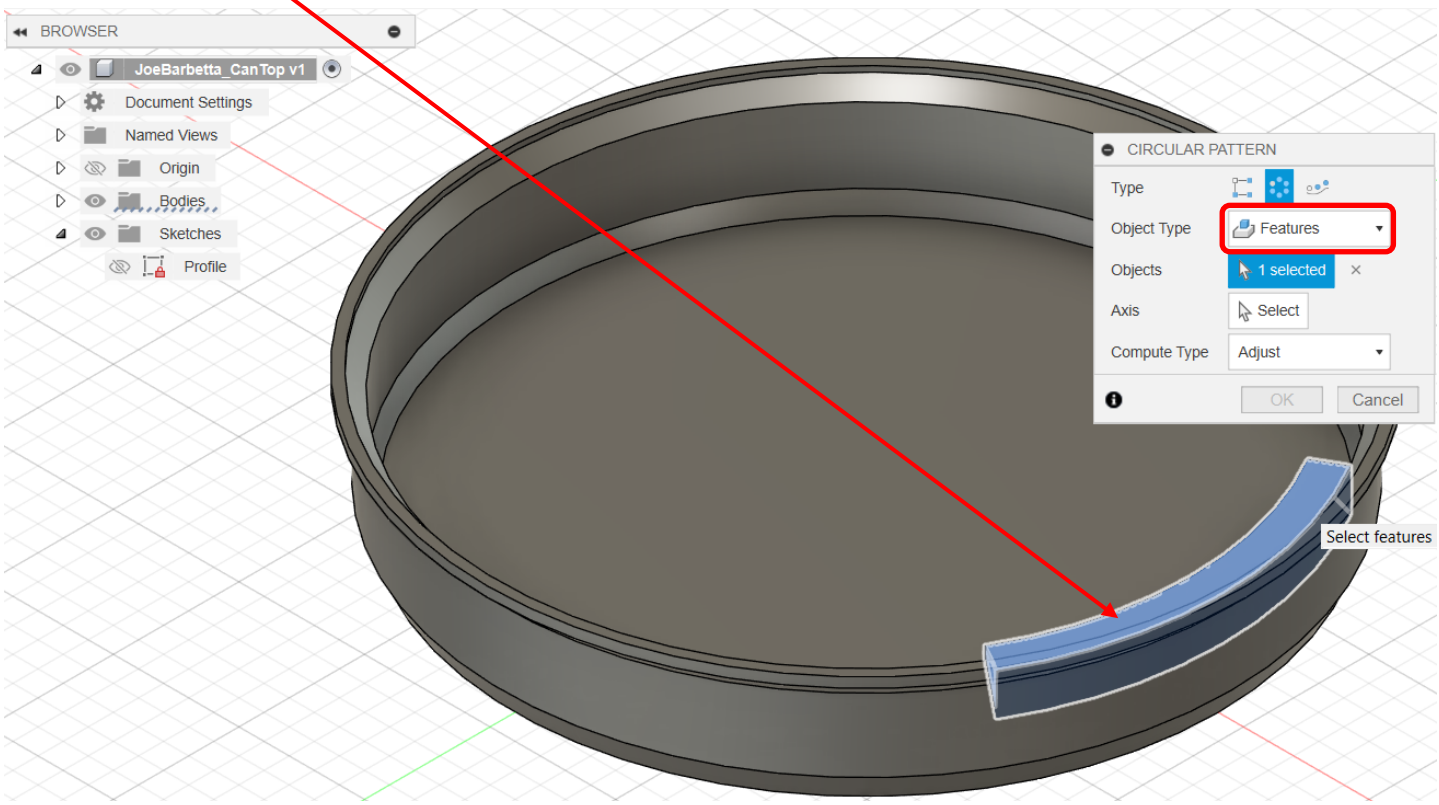


Creating a Circular Pattern for a Feature

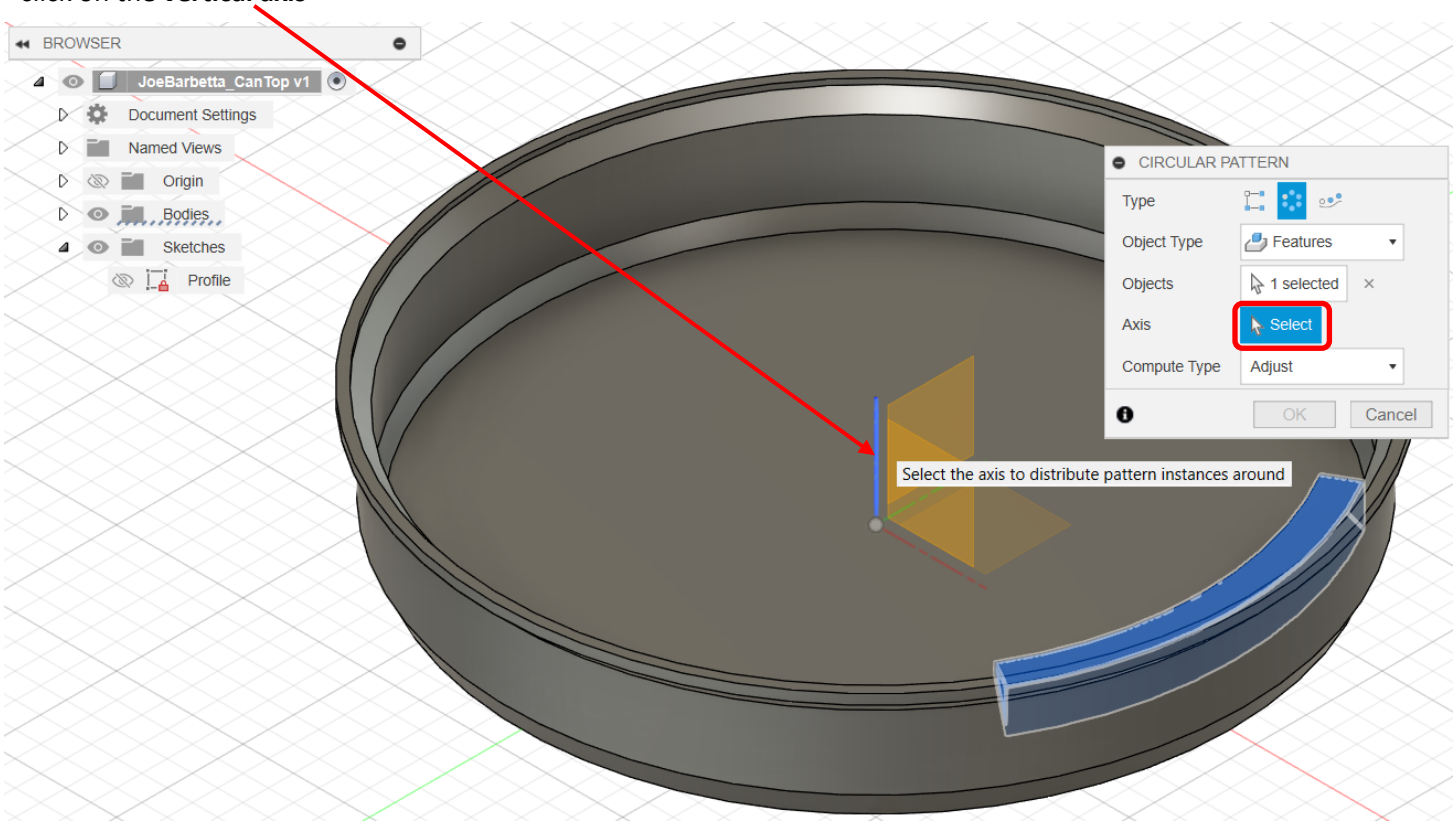
- from the **CREATE** menu select **Pattern** and **Circular Pattern**



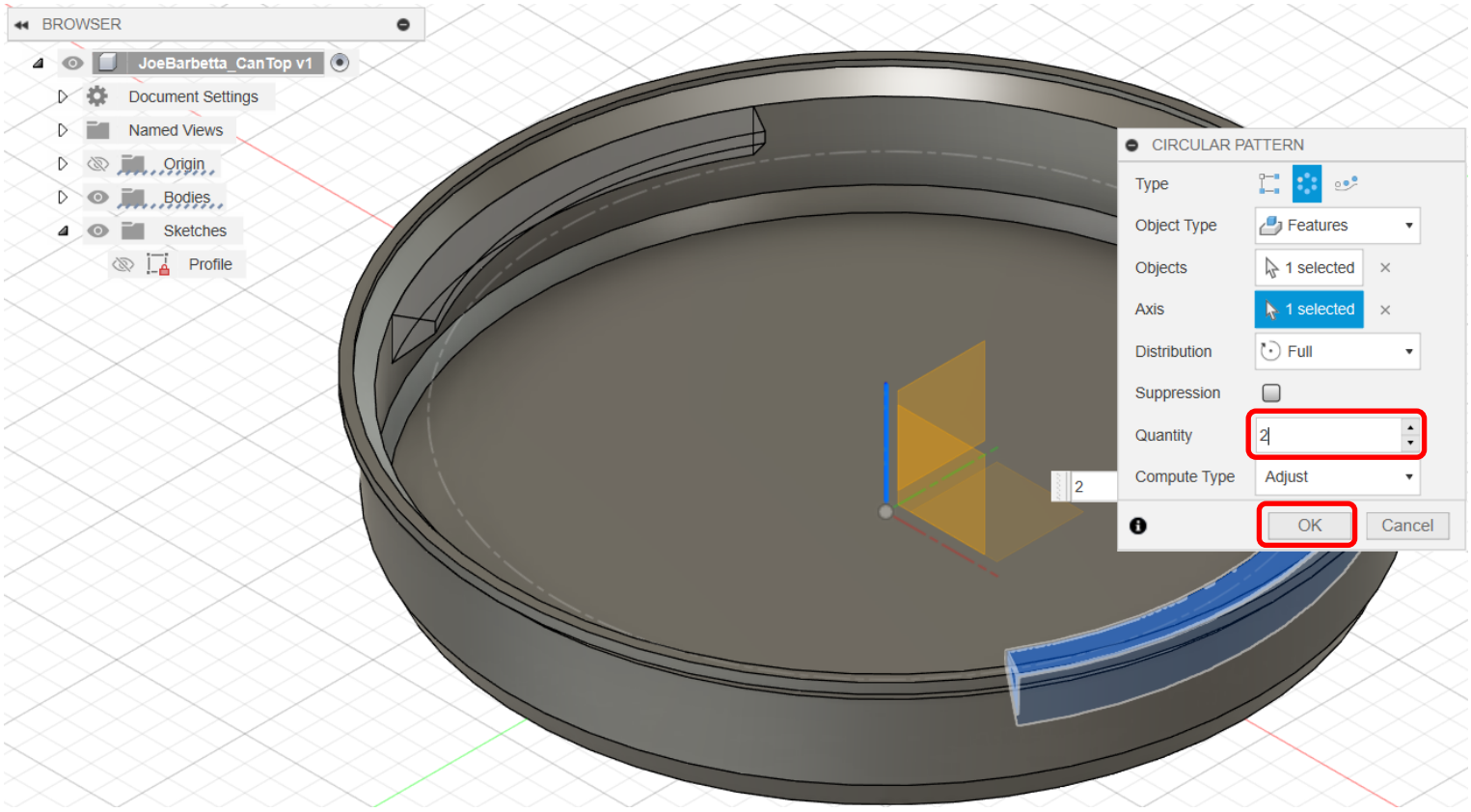
- for **Object Type** select **Features**
- click on the **resulting section** of the partial revolve, which should turn it blue



- click on **Select** for **Axis**
- click on the **vertical axis**

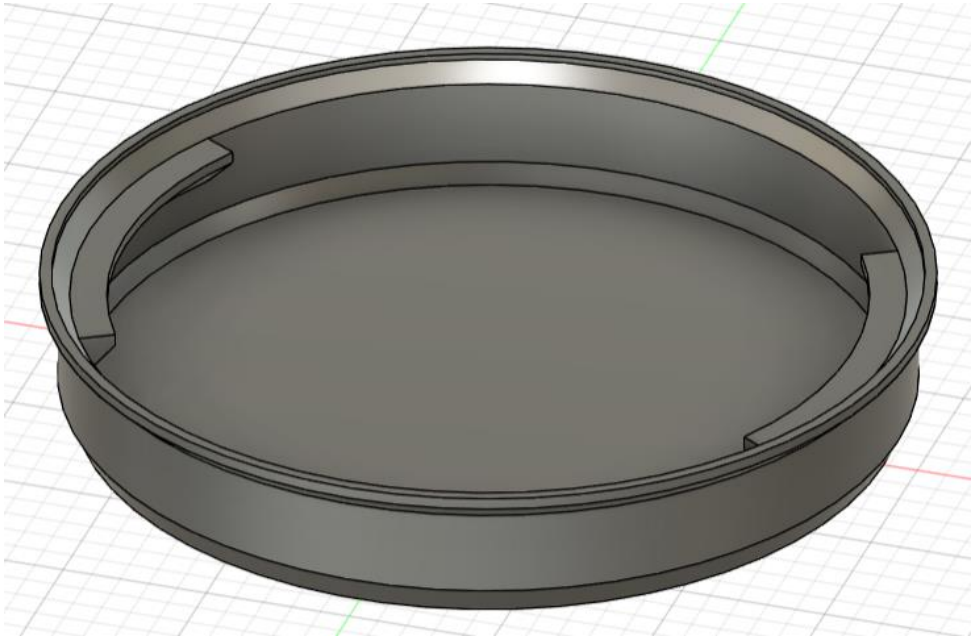


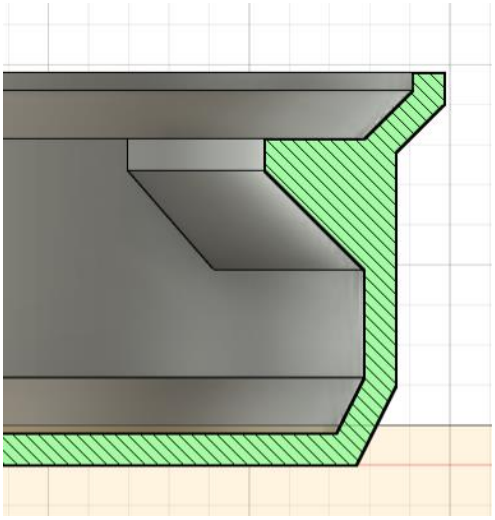
- change the **Quantity** to **2** and click **OK**



The purpose of the two inner sections is to have locations where a finger can grab the cap to remove it from a can.

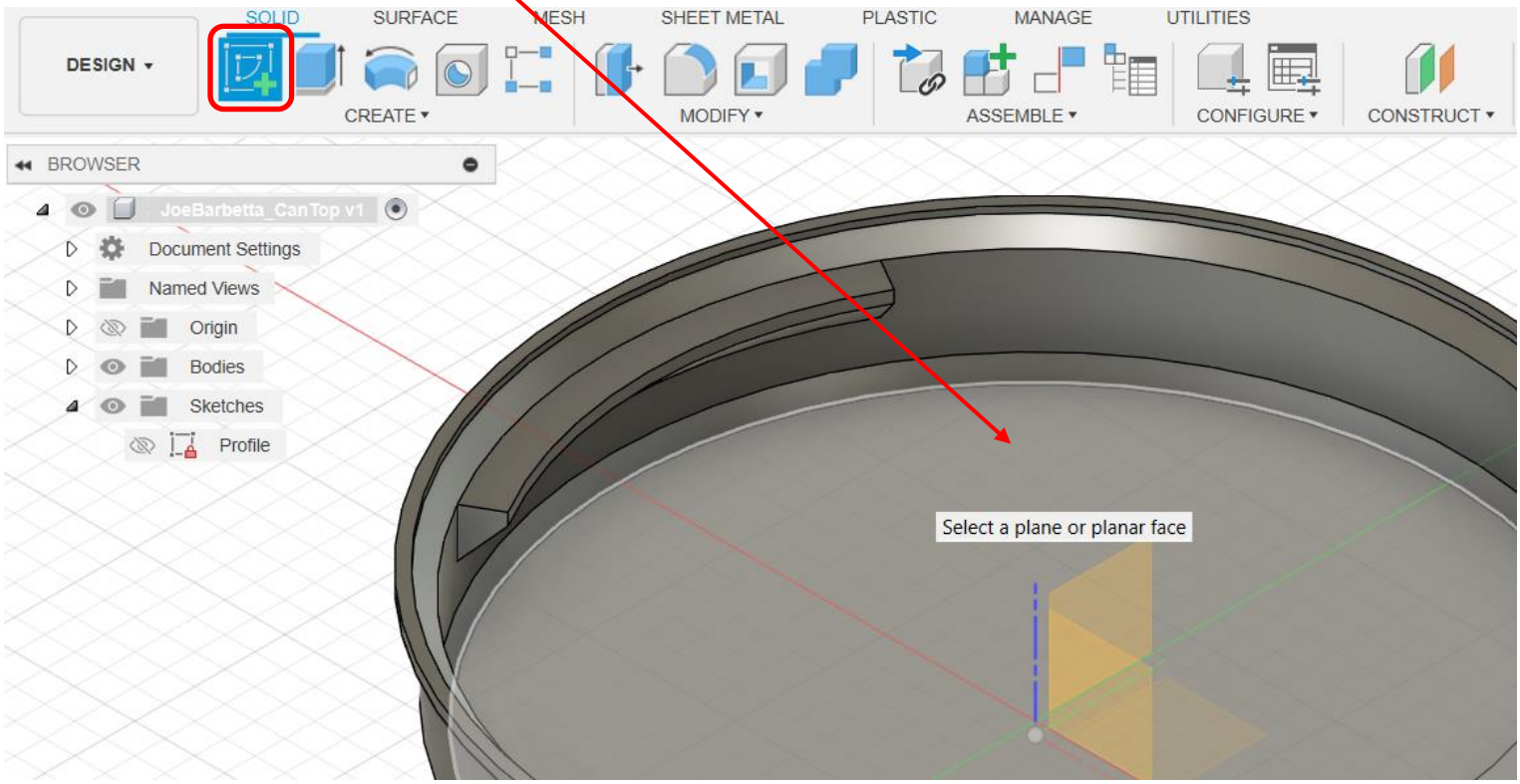
The cross-section view on the right shows that the 45 degree angle used allows the cap to be printed without supports.



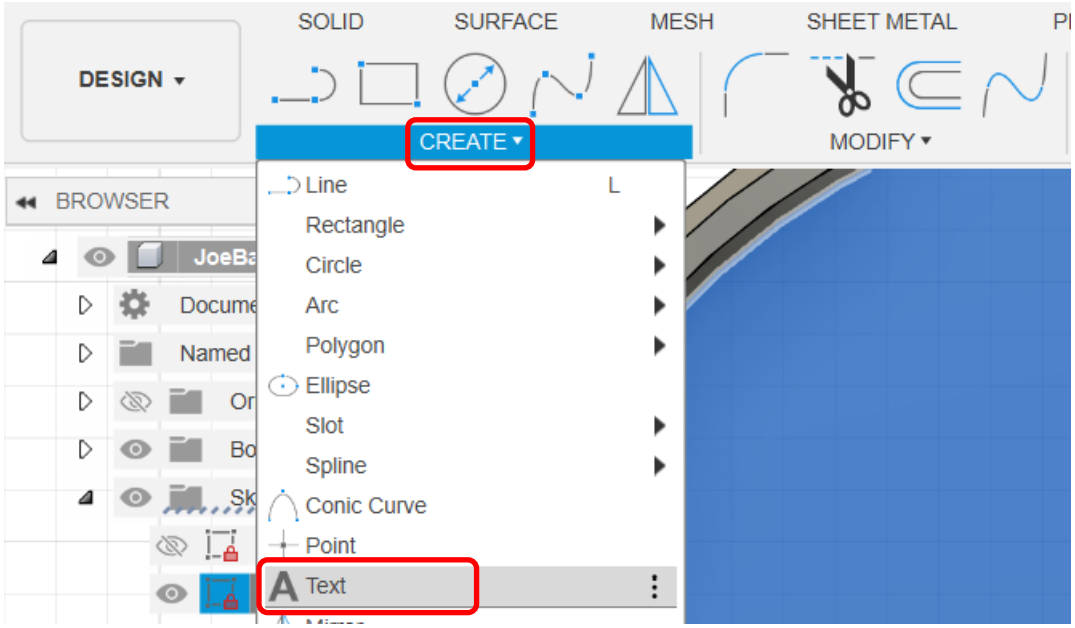


Adding Text

- select **Create Sketch** and click on the **top of the cap**

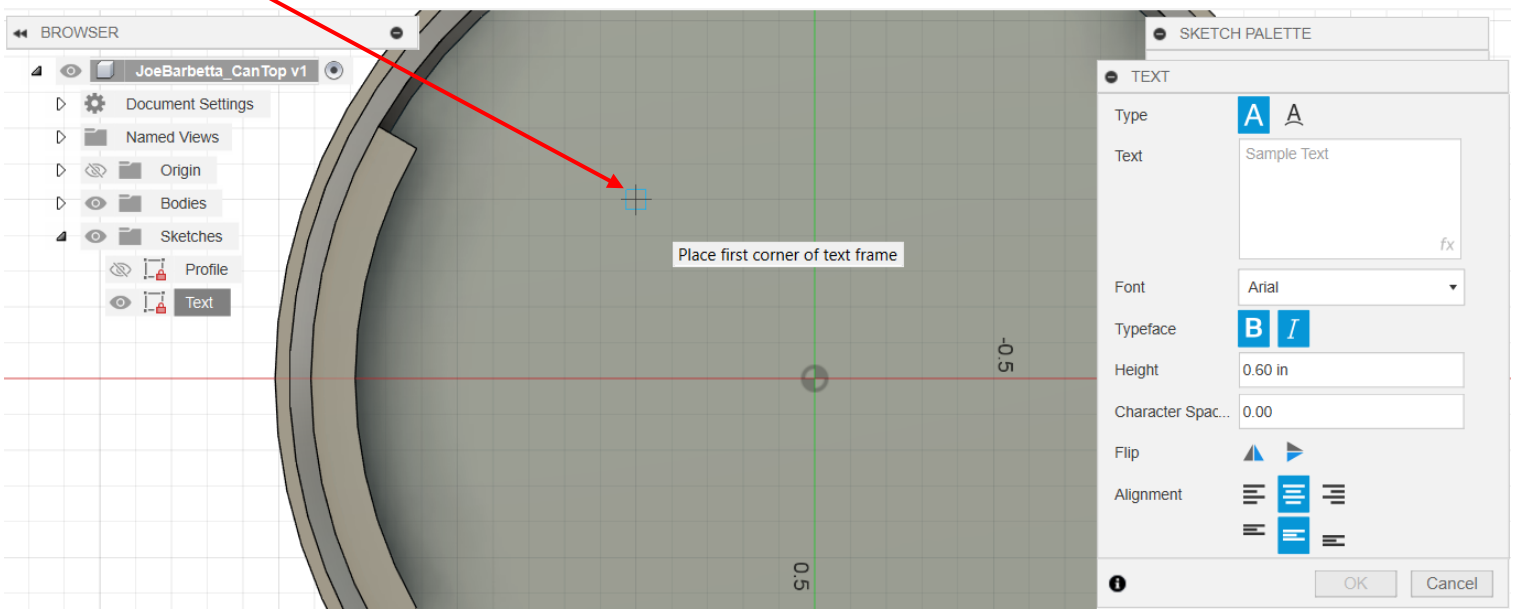


- from the **CREATE** menu, select **Text**
- if a **Parametric Text** message window appears, click its **OK** button

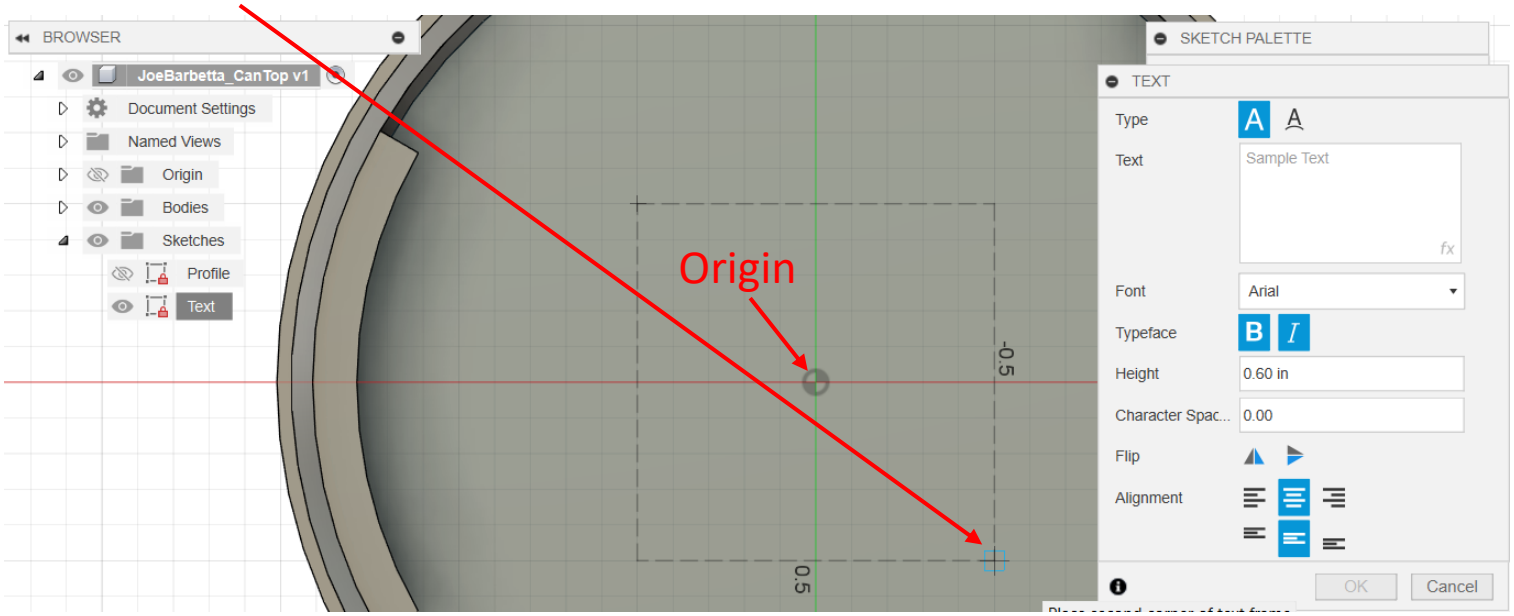


The following 2 steps will create a rectangle to define the location of the text. The rectangle should be centered around the Origin and thus centered on the cap, but the position is not critical.

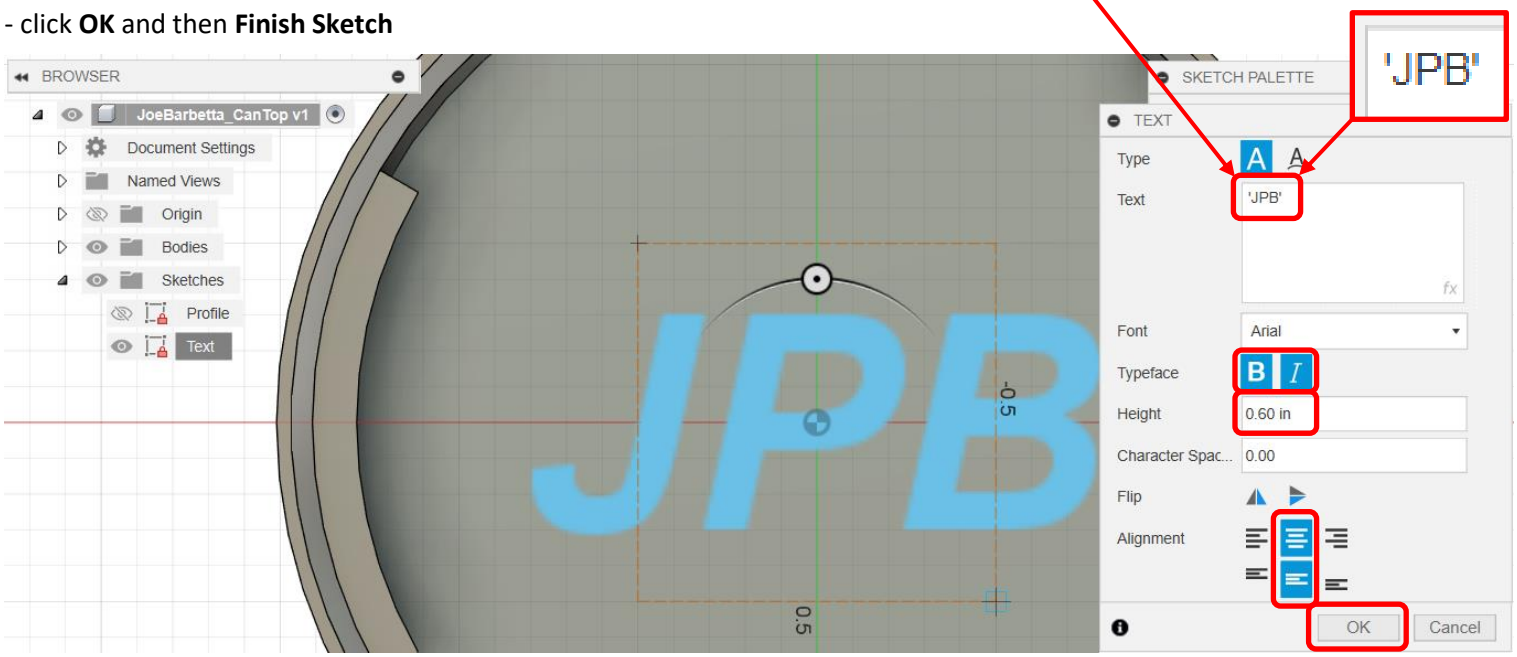
- click on a **point** in the indicated area. The location is not critical.
- if a **Parametric Text** message window appears, click its **OK** button



- extend the rectangle down and to the right and click again so that the rectangle is centered around the Origin
- click on the **2nd point**

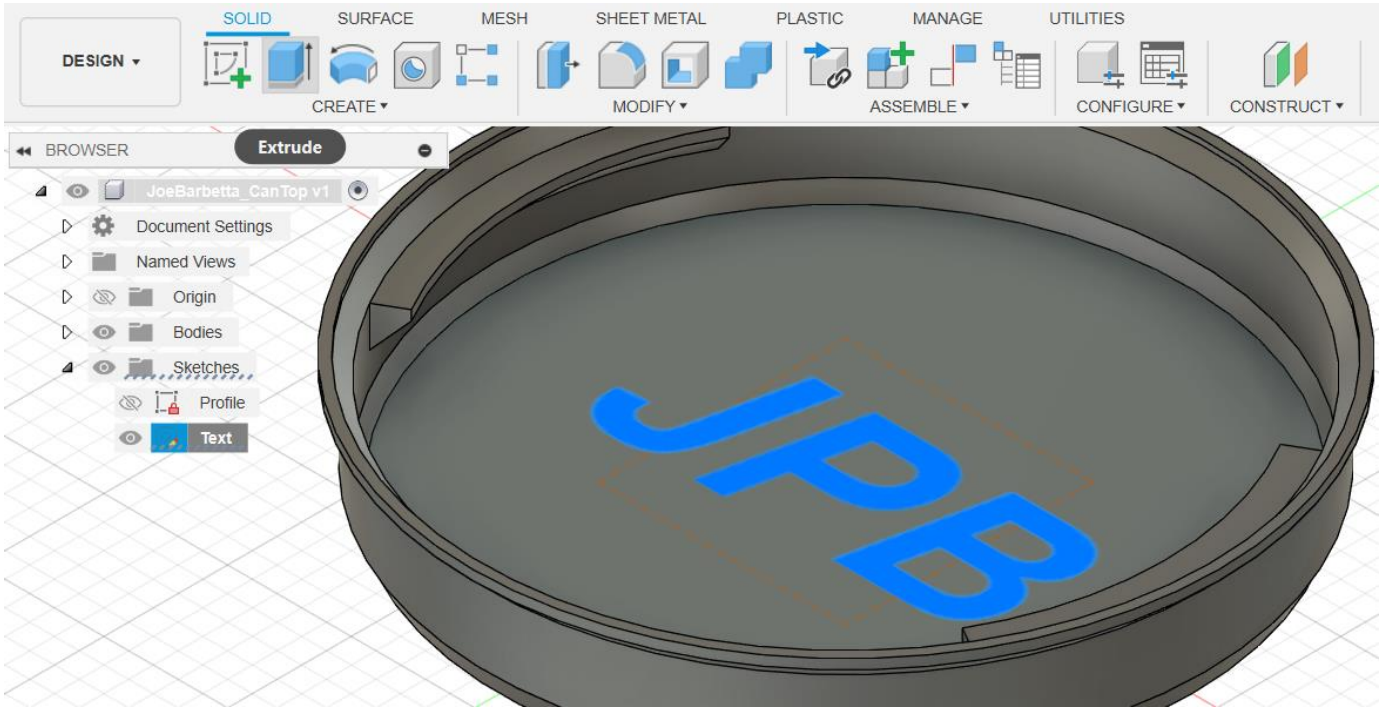


- in the Text box enter your **3 initials preceded by and followed by a single quote**
- click on the **Bold** and **Italic** options to highlight them
- set the **Height** to **0.60**
- click on the two center **Alignment** options
- click **OK** and then **Finish Sketch**

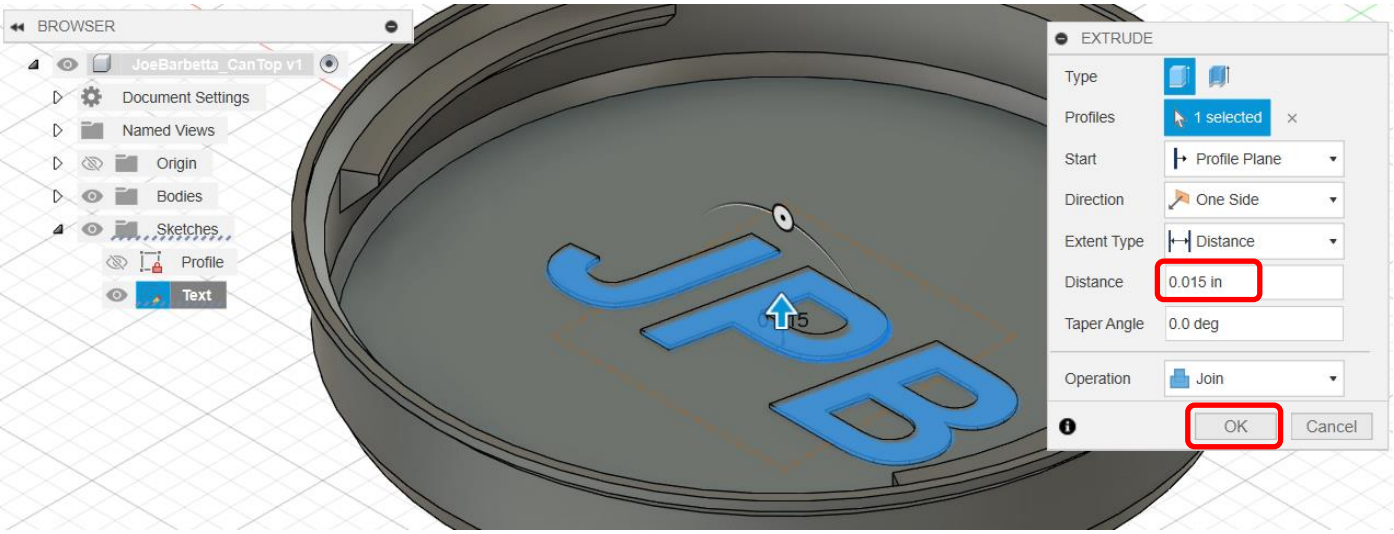


Extruding Text

- click on the **text** to select it
- select the **Extrude** tool. If it is not visible, find it in the CREATE menu.



- for **Distance** enter **0.015** and click **OK**

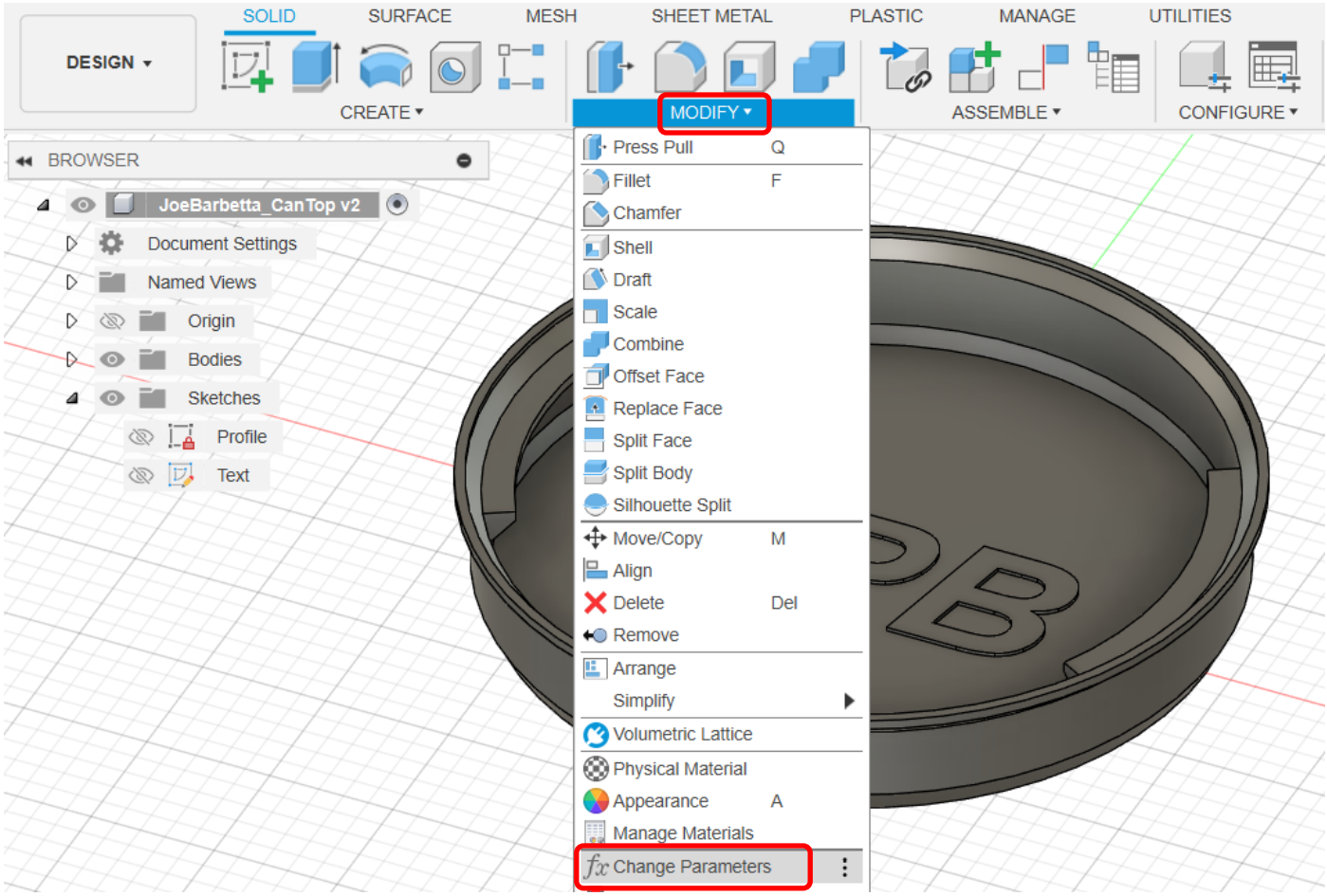


- admire your cap

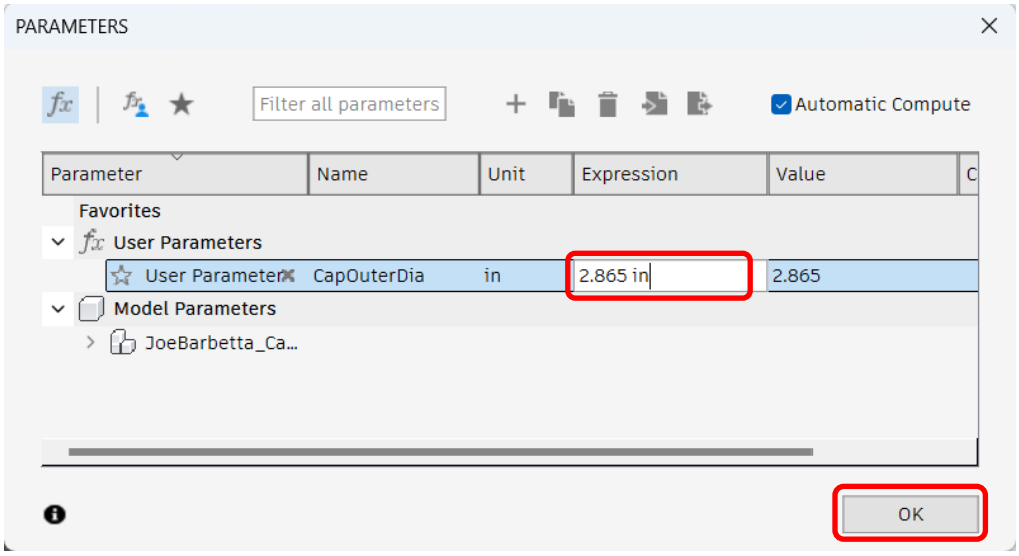
Changing User Parameters

After performing an initial print, one may find that the top is too tight or too lose. In this case it was found that using 2.865 instead of 2.9 provided a good fit. Because we started with a User Parameter for the diameter, we can change it and the entire model will change as needed.

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

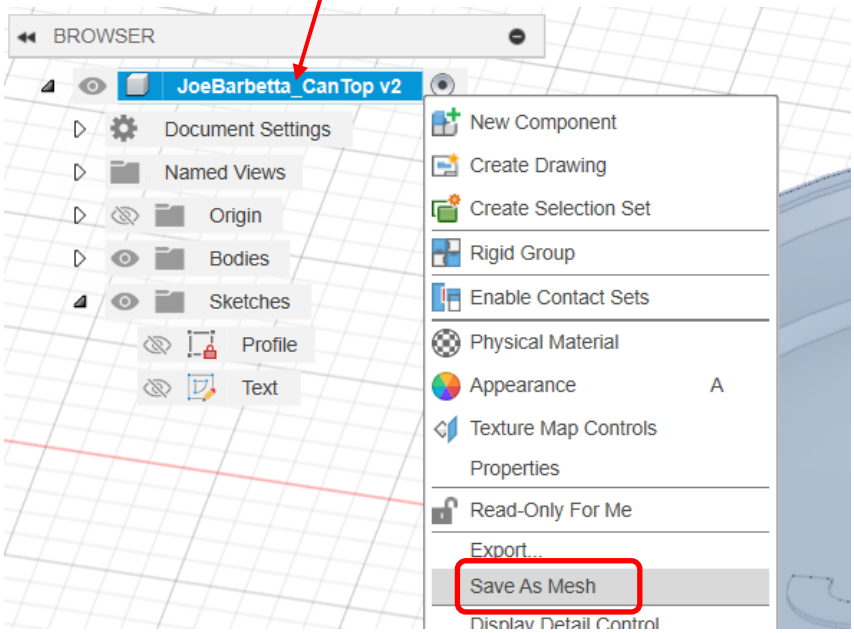


- click on the Expression field and change the value to 2.865 and click OK. You should notice the cap will become slightly smaller.

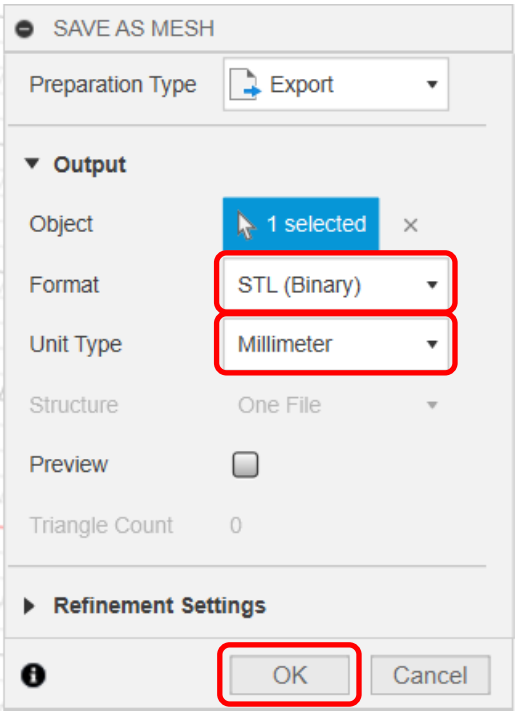


Creating a STL File

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



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



- ensure that **Save to my computer** is checked
- make note of the save location. By default it should be the Downloads folder.
- click **Save**