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

Using a Caliper to Copy a Button

Here a caliper is used to measure a **Flying Screaming Monkey** head diameter.

No Flying Screaming Monkeys were harmed making this lesson.

Original button

Copied button

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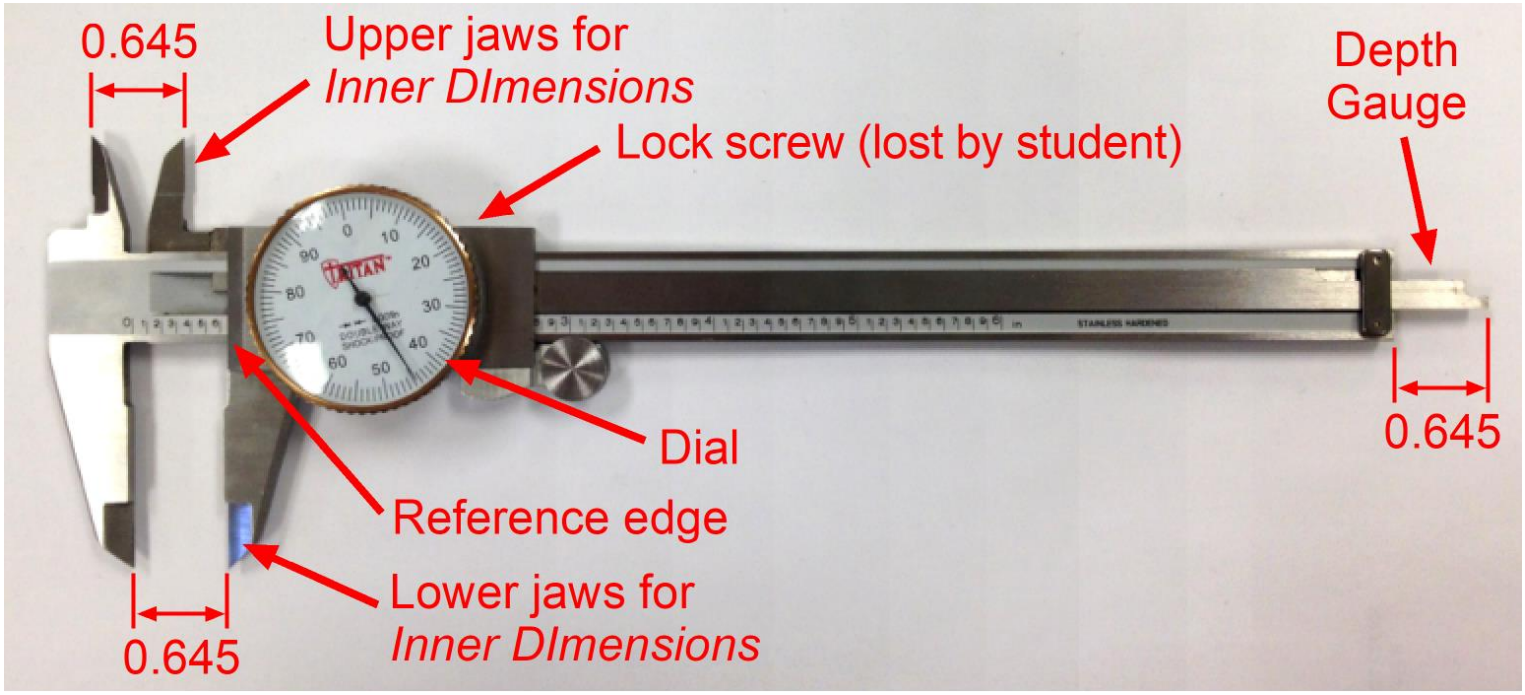
Contents

Introduction to the Caliper

The first caliper was invented by Giuseppe Calliperri for sorting olives by size for export out of Sicily. The accuracy was later improved with a design by Carlo Verniero, which came to be know as a Vernier scale. Some modern calipers still use this method and one can achieve resolution of 0.001" with a Vernier scale.

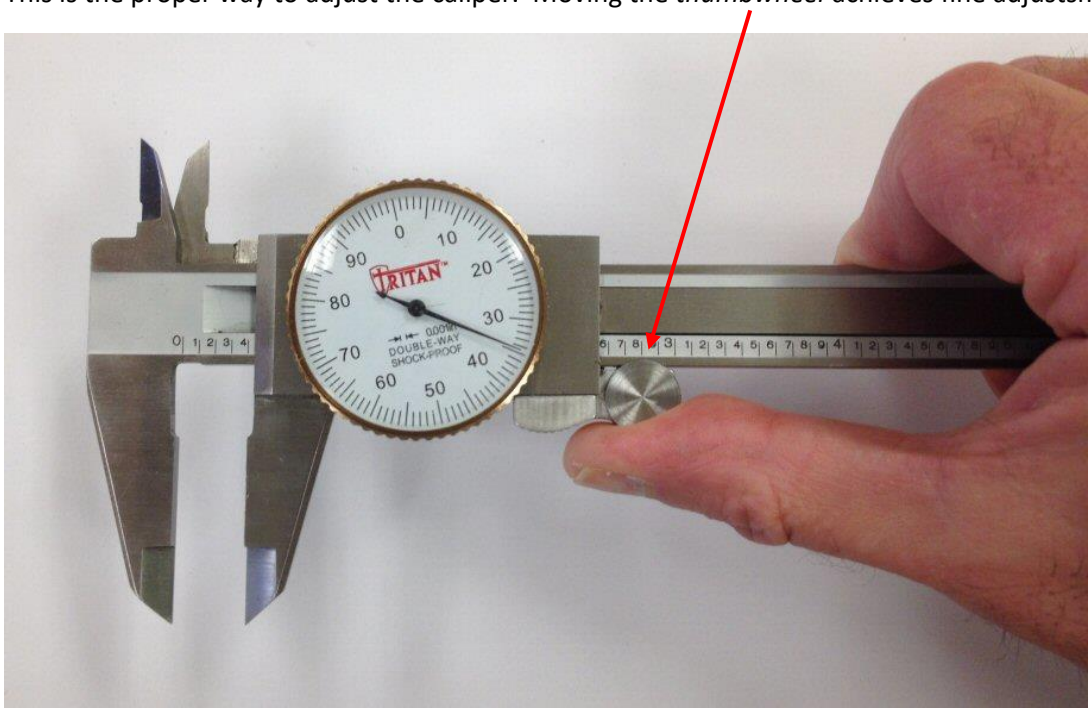
Perhaps the two most commonly used calipers today are *dial calipers* and *digital calipers*. Digital calipers are the easiest to use and can be switched between inches and millimeters at the touch of a button, but they rely on a battery.

One thing that is neat about a caliper is the fact that the **upper jaws**, **lower jaws**, and **depth gauge** are always snychonized.



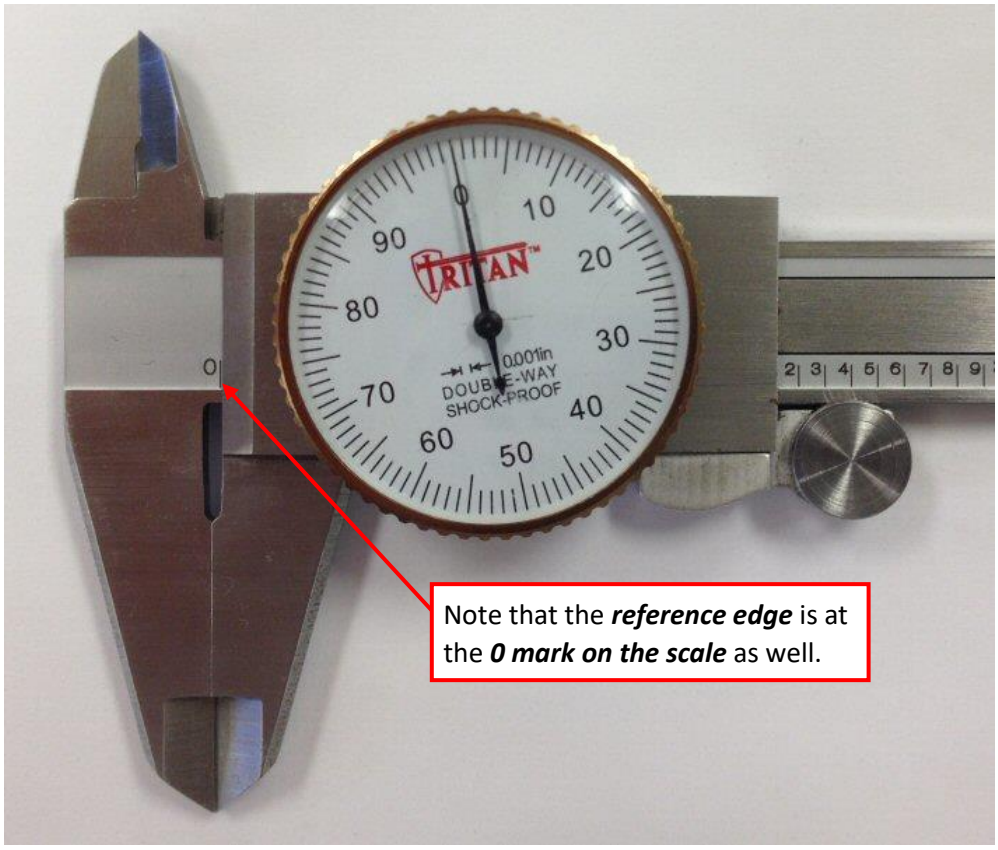
As with scissors **never run with your caliper**. The jaws are sharp.

This is the proper way to adjust the caliper. Moving the *thumbwheel* achieves fine adjustsments.



Zeroing the Caliper

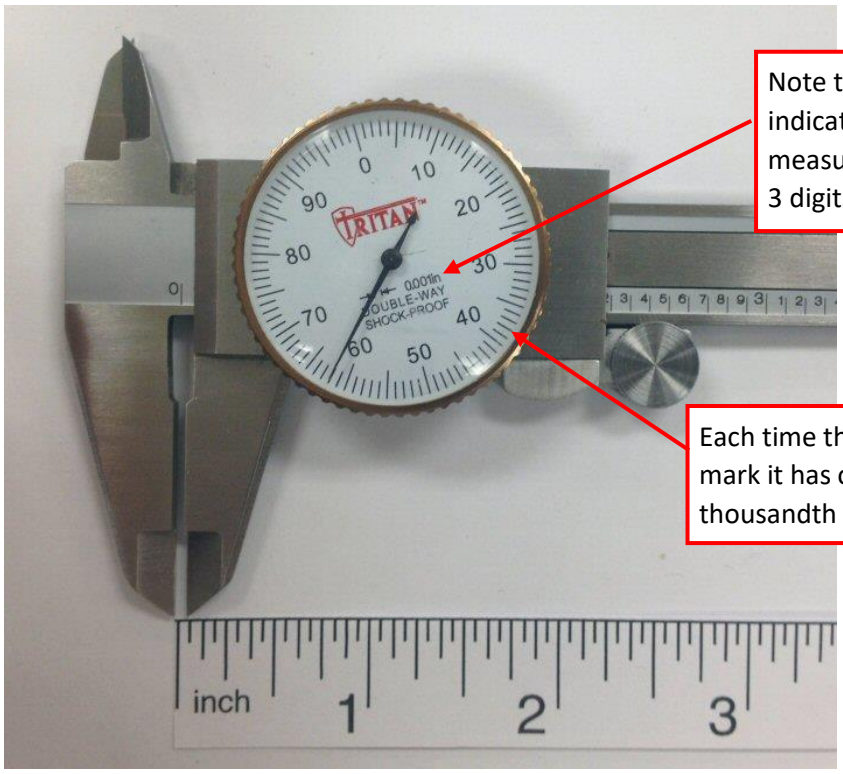
With any caliper, the first step is to ***zero the caliper***. This is done by closing the caliper completely and then turning the *dial bezel* so that the needle is on the ***0 mark***. Usually, the 0 mark will be at the top, but this may not be the case. If this were a digital caliper, one would press a Zero button.



If you have a digital caliper **and the battery is not dead**, one can press the **ZERO button** to ***zero the caliper***.

Reading the Caliper

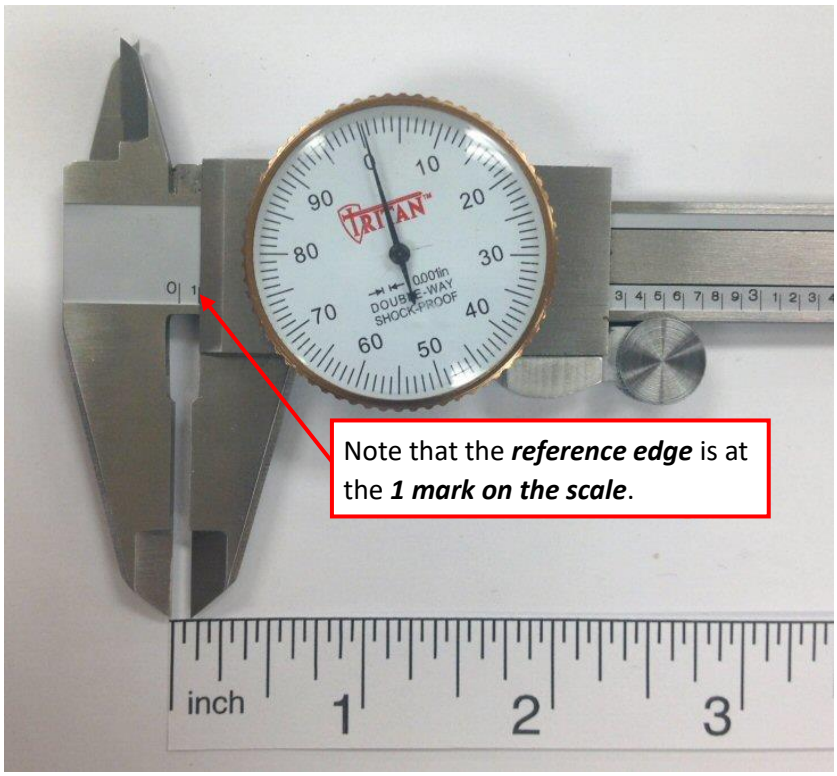
As the caliper is opened, the needle will turn clockwise. Each small mark on the dial represents a thousandth of an inch, 0.001" and every 10th mark is numbered. Here the needle is at **62** corresponding to **0.062"** (**1/16"**).



Note that the dial face states **0.001 in**. This indicates the ***resolution*** of the instrument and measurements made should be written with 3 digits after the decimal point, e.g. **0.250 in**

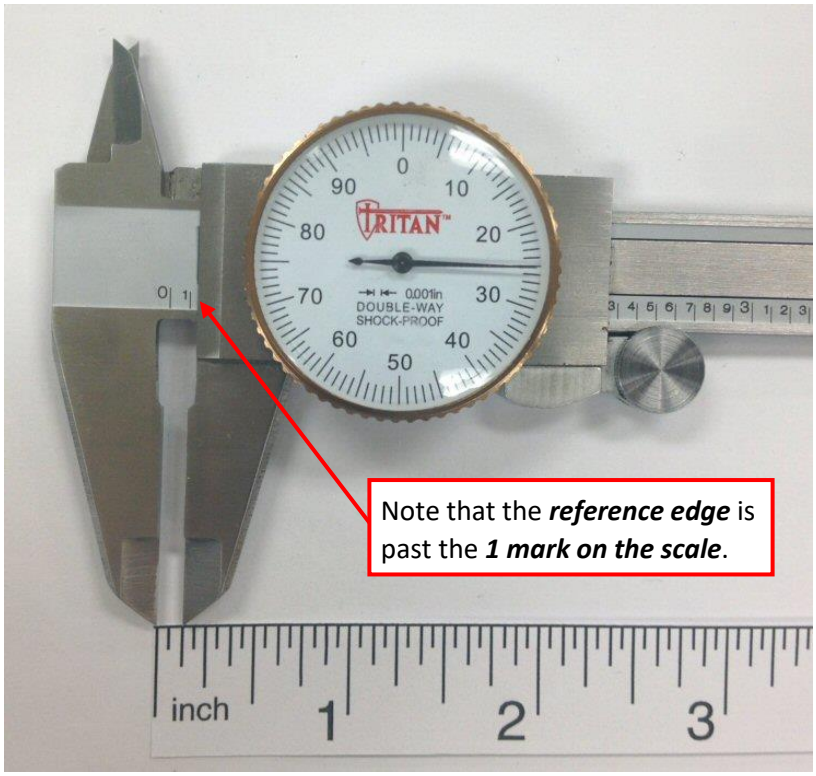
Each time the needle passes another mark it has opened another thousandth of an inch (0.001").

As the caliper is opened further, at point is reached when the needle has made a full revolution and is back at **0**. The **reference edge** as shown is now at the **1 mark on the scale**. The measurement is now at **100 thousandths of an inch or 0.1"**. One should write this measurement as **0.100"**. This is mathematically equivalent, but maintaining **3 digits after the decimal point** indicates the **resolution** of the measurement. Remember significant digits?

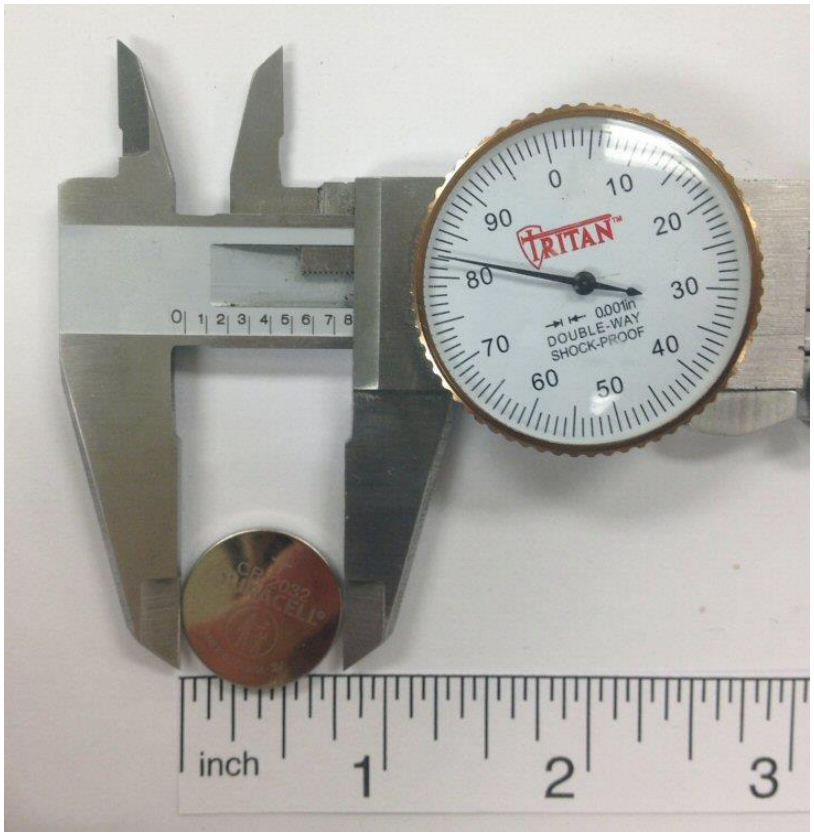


Note that the ***reference edge*** is at the ***1 mark on the scale***.

Here the caliper is shown open to **0.125"** (**1/8"**). The reference edge is past the **1 mark** (**0.100"**) and the needle is at **25** (**0.025"**). Adding **0.100"** and **0.025"** results in **0.125"**. For every measurement one adds the equivalent of the mark on the scale to that of the needle.



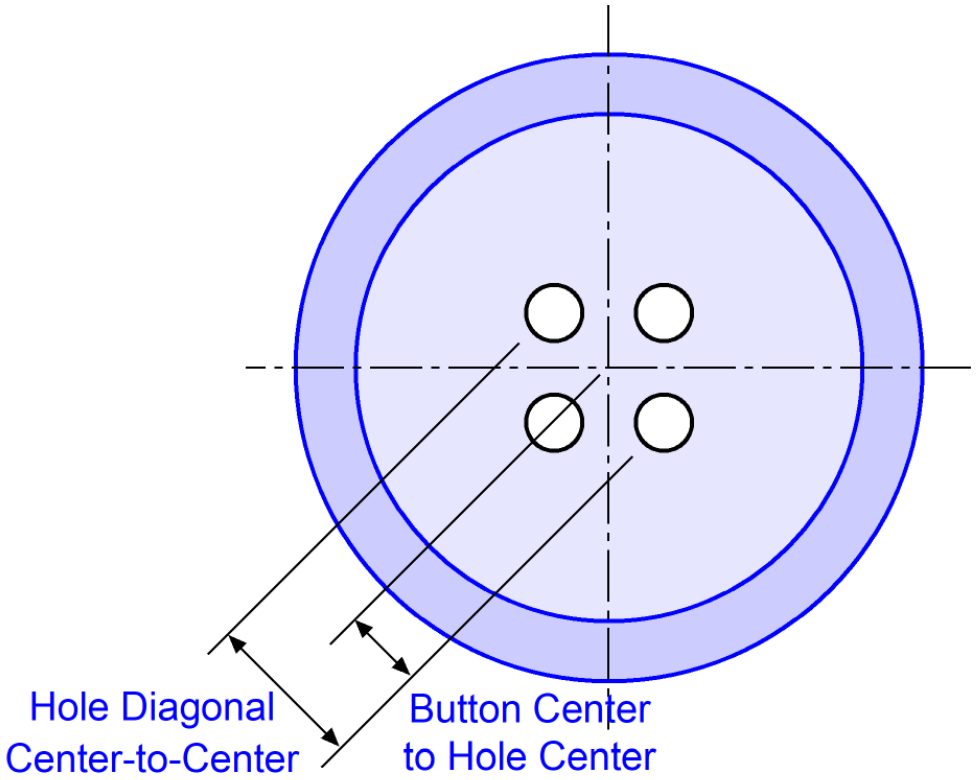
Here a battery (coin cell) is measured. The reference edge is past the **7 mark** (**0.700"**) and the needle is at **78** (**0.078"**) and thus revealing a measurement of **0.778"**. Note that one can see the 8 on the scale, but the reference edge has not reached its mark. If it had, the needle would be at 0 or slightly past 0. Converting 0.778" to mm by multiplying by 25.4, yeilds 19.76 mm. This must be a 20 mm coin battery.

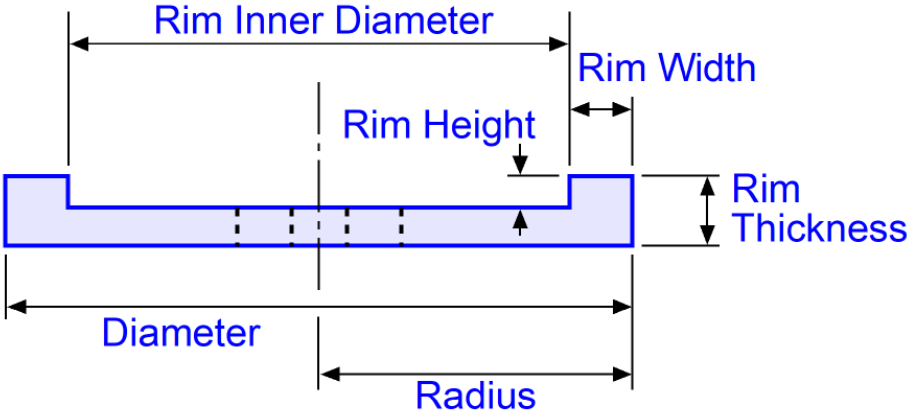


Measuring a Button

The button drawing shown here is just an example. Your button will be different!

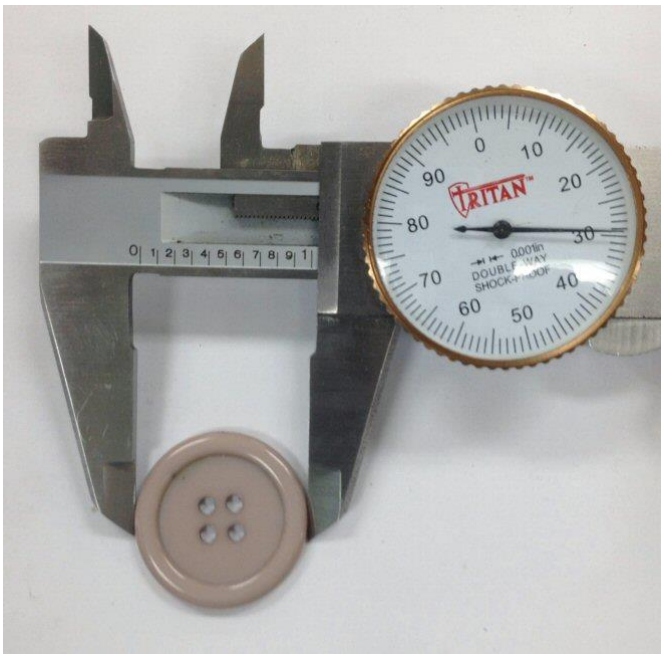
Here is a drawing (top view and cross section) of a simplified button and dimensions that can be measured to create a model of the button in a CAD program, such as Fusion 360.



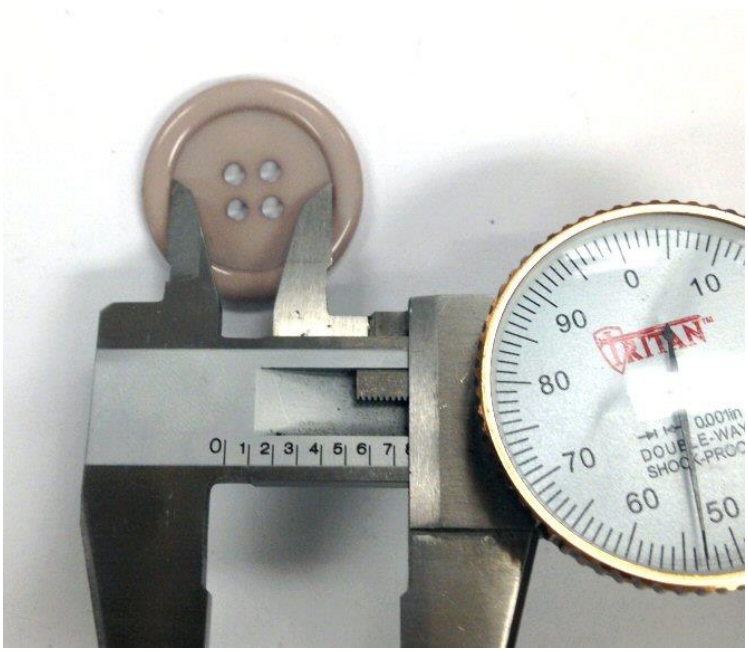


The measurements shown here are just an example. Your button will be different!

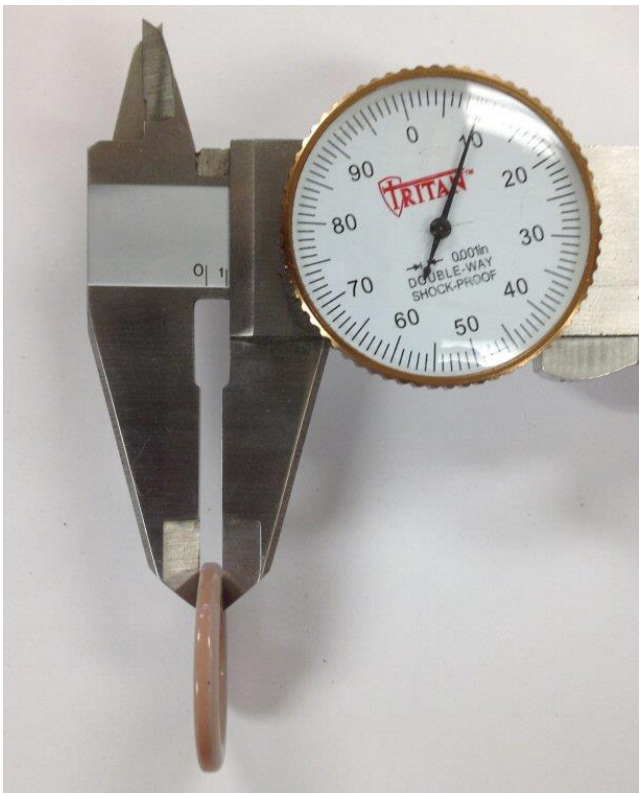
Measuring the **Diameter**



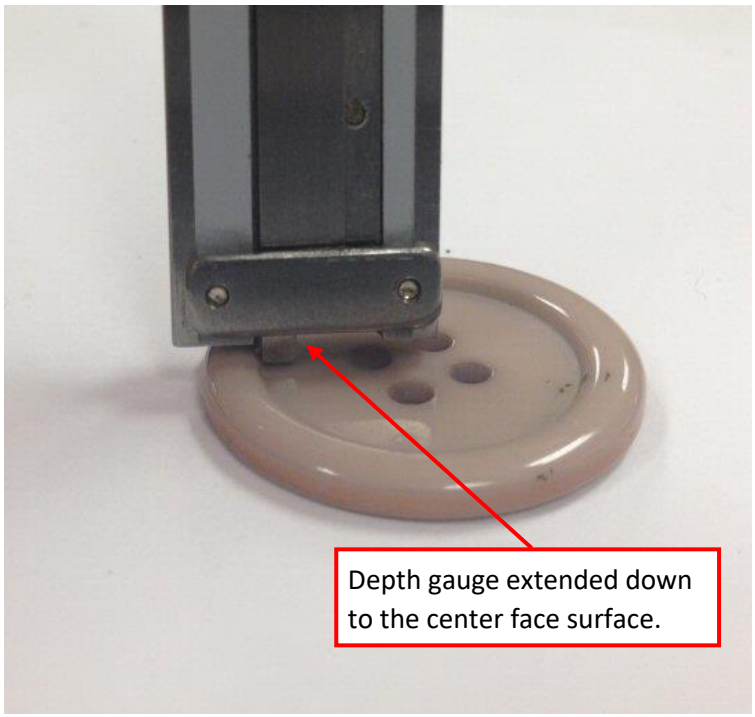
Measuring the **Rim Inner Diameter**



Measuring the **Rim Thickness**

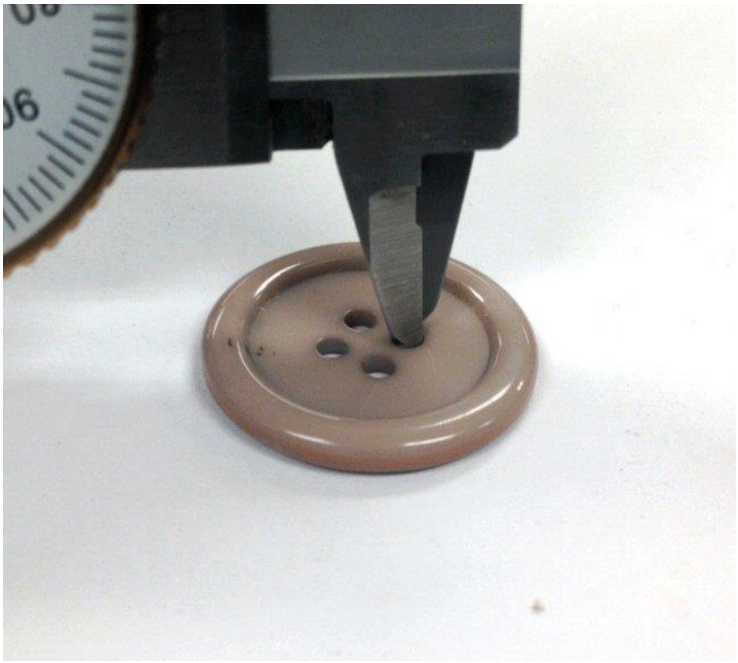


Measuring the **Rim Height**. The **depth gauge** is extended down to the surface below the edge.



The measurements shown here are just an example. Your button will be different!

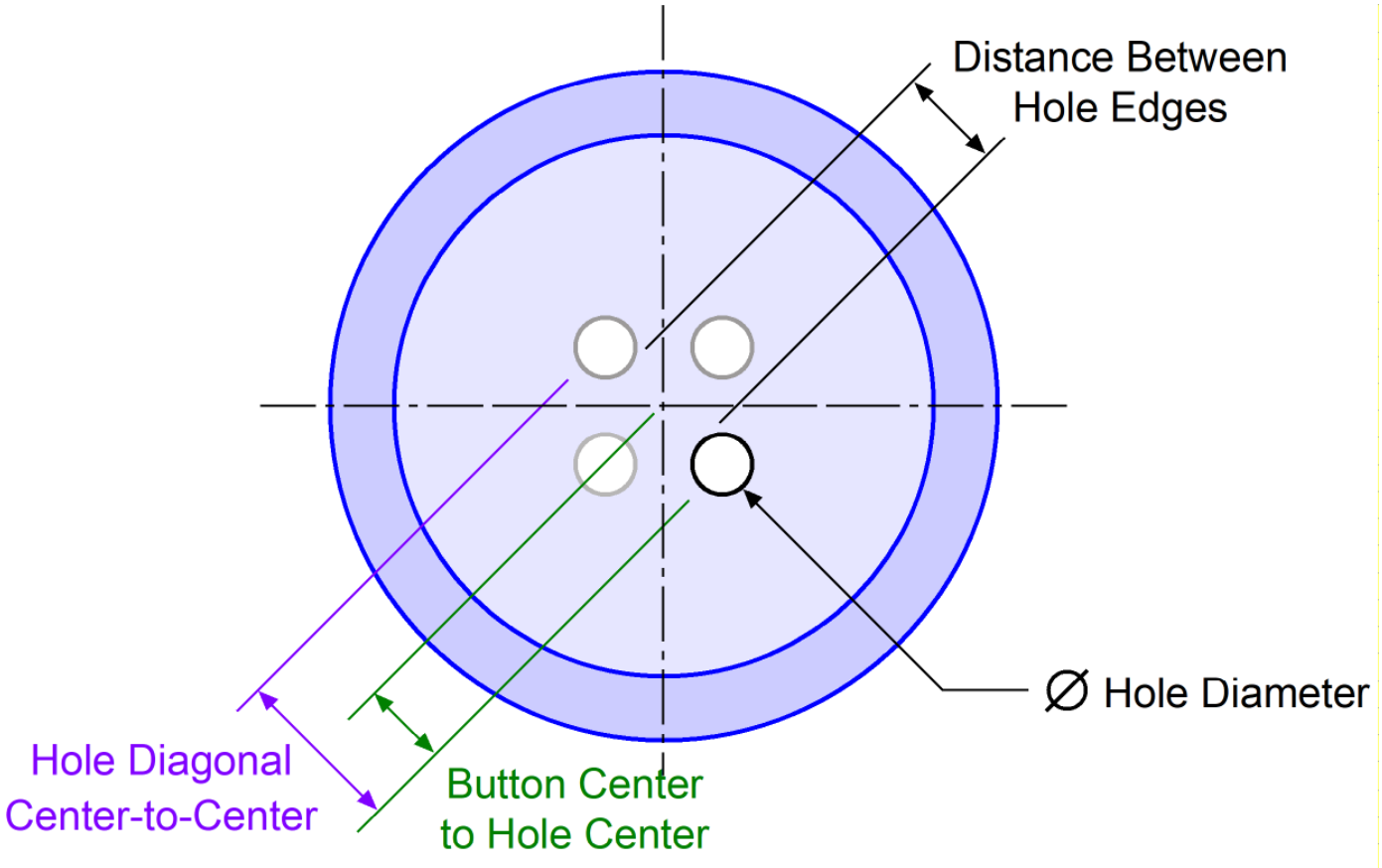
Measuring the **Hole Diameter.**



Measuring the **Distance Between Hole Edges.**

The button drawing shown here is just an example. Your button will be different!

One can see in the drawing below that adding the measured **Hole Diameter** to the measured **Distance Between Hole Edges** will result in the **Hole Diagonal Center-to-Center** dimension. Dividing this by 2 results in the **Button Center to Hole Center** dimension. In Fusion 360 a line will be created using this dimension to position a hole location from the center. The Circular Pattern tool when then be used to create the other 3 holes.



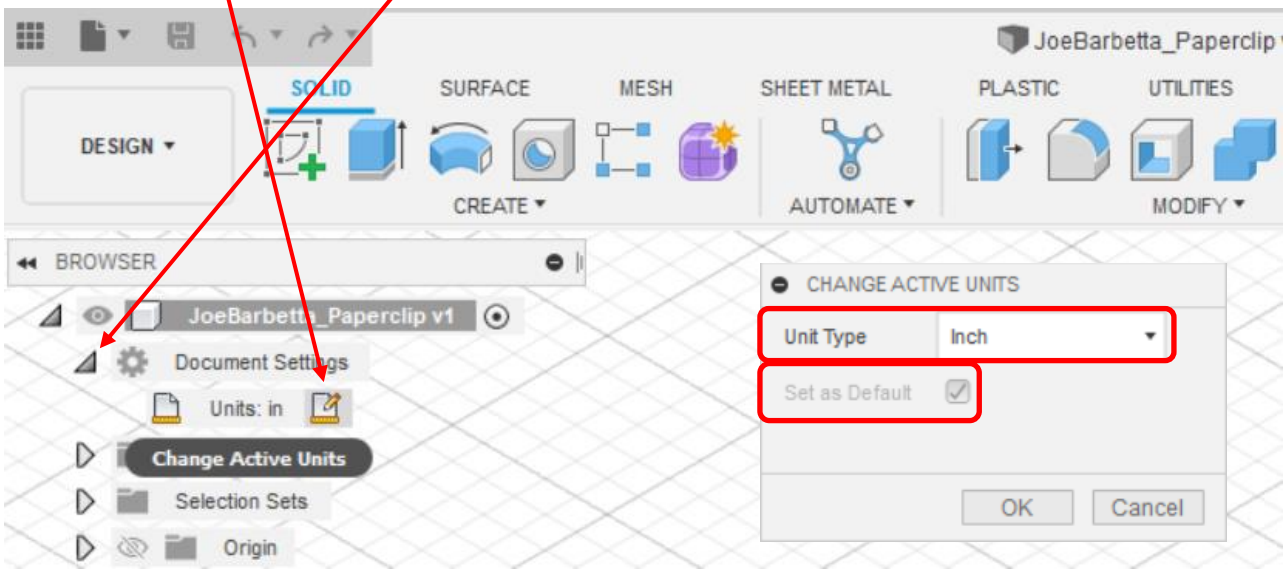
Starting a Design in Fusion 360

- open **Fusion 360**. If there is no icon on the Desktop, use the Windows search (magnifying glass icon) and type fu
- from top **File** icon select **Save** and name the file.
  Use your name followed by **_Button** e.g. **JoeBarbetta_Button** (note the use of the underscore)

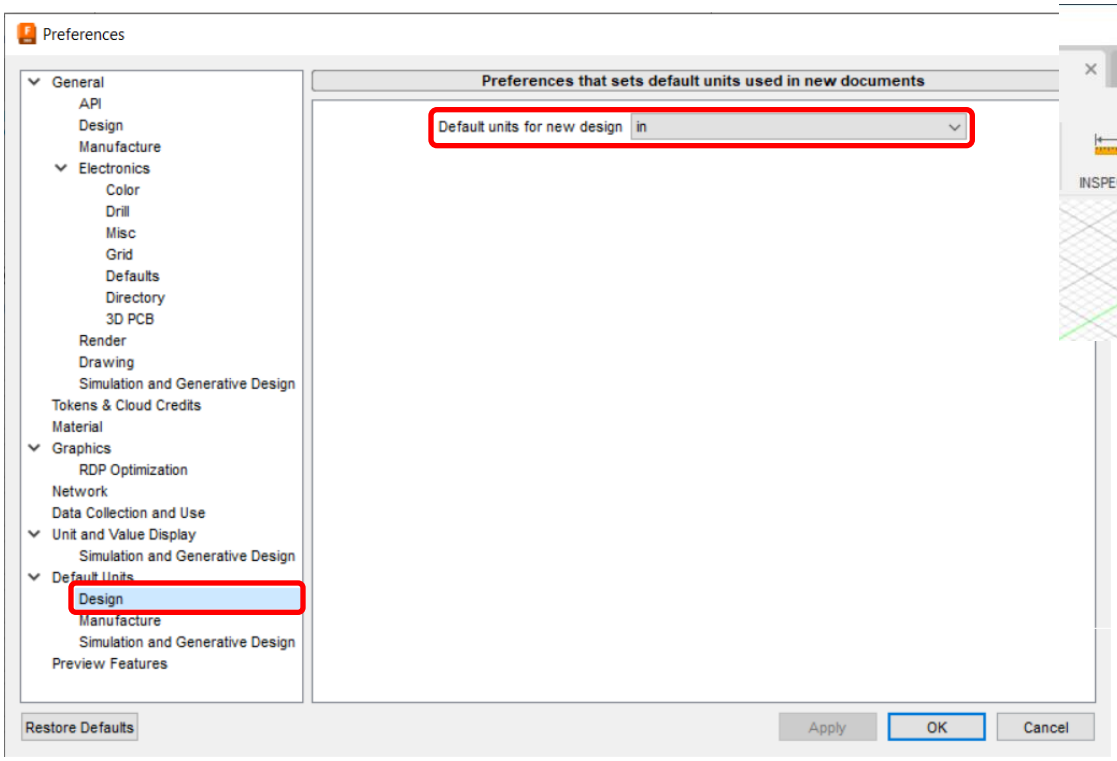
Note that by default Fusion 360 saves your project to "the cloud", which are the servers managed by AutoDesk. When you log into Fusion 360 on a different computer, your projects will be available.

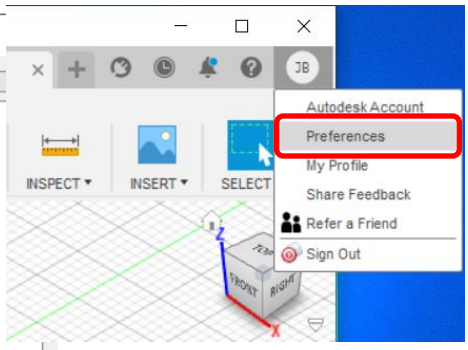
As you work, occasionally save your work in case Fusion 360 crashes or a squirrel chews the school's fiber optic.

- in the left **"BROWSER"** click on the **arrow next to Document Settings**
- click on the **edit icon** that appears to the left when you hover over **Units**
- change **Unit Type** to **Inch** and click **OK**. You can also enable **Set as Default** if it is not grayed out.



One can also change the Design Units to Inches in Preferences so any new project will use Inches automatically.

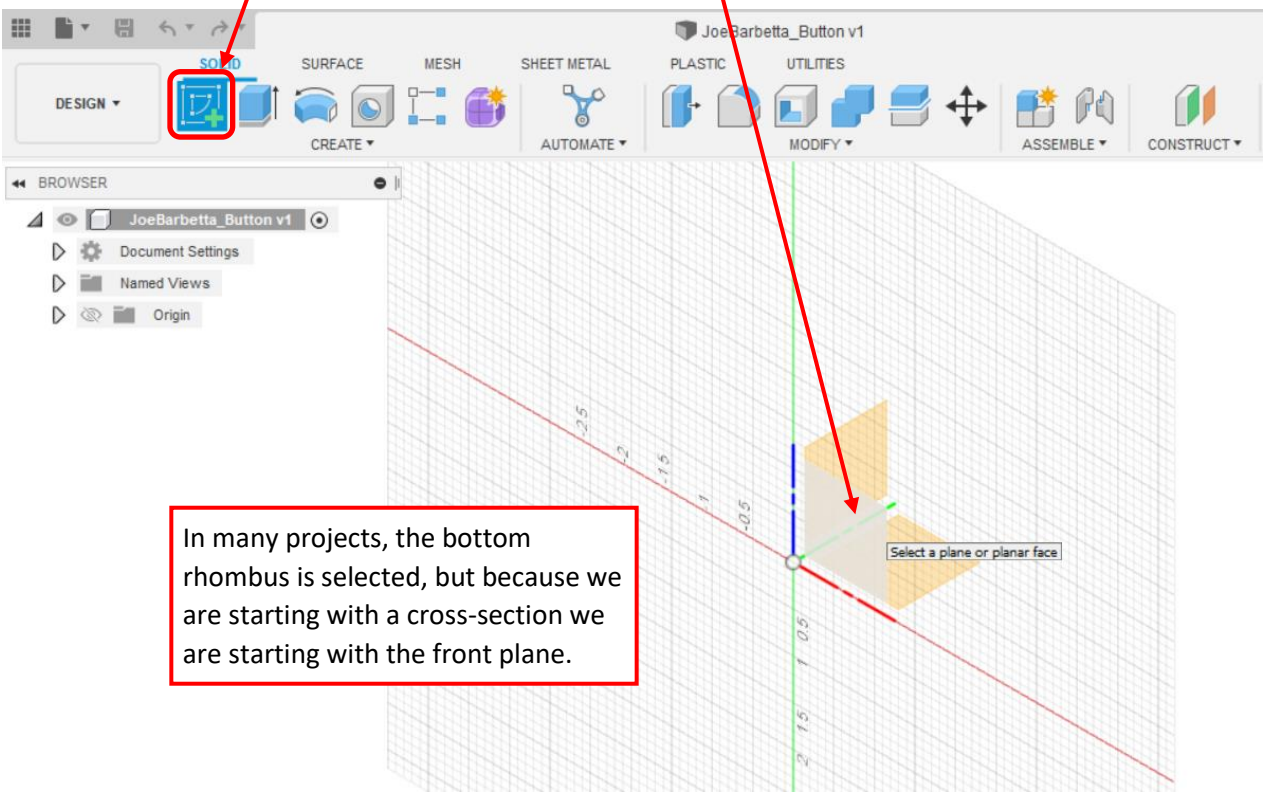




Creating the First Sketch

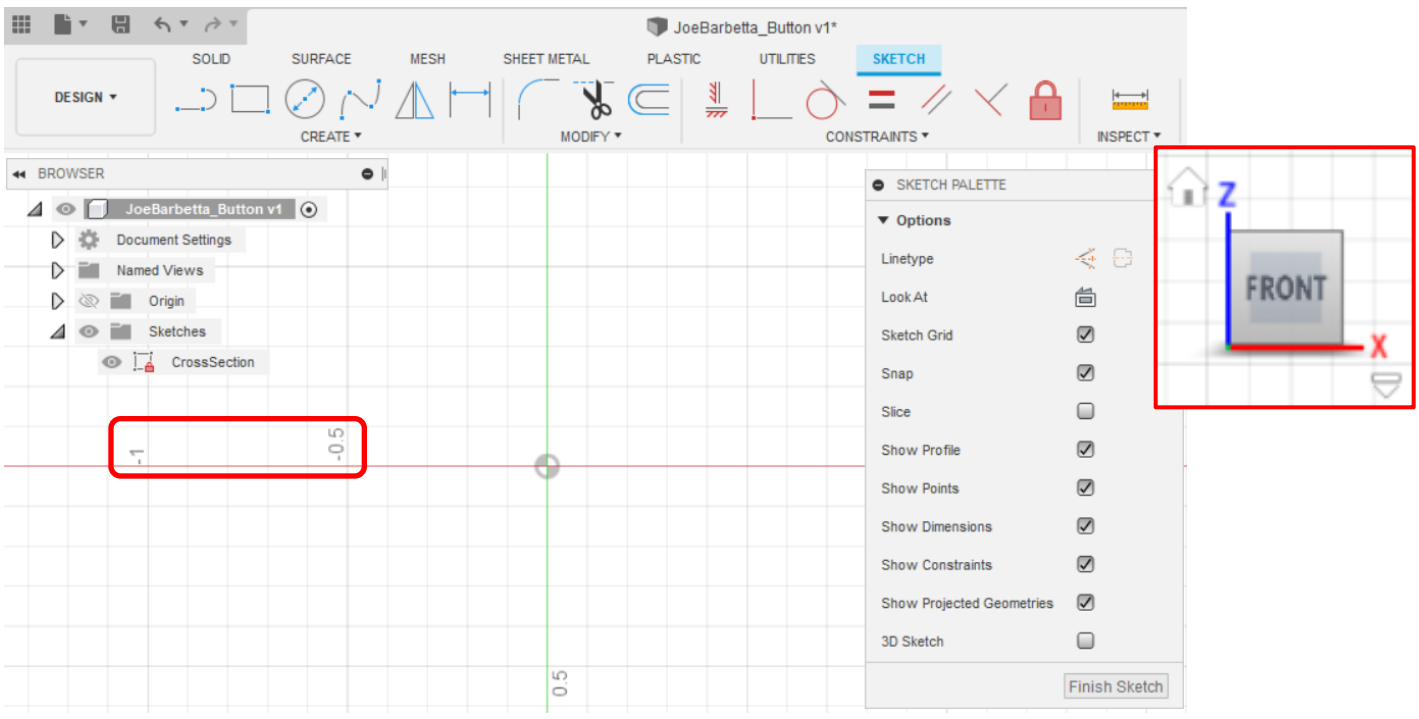
Note that a Fusion 360 expert may tell you to create a Component first, but you can say **"Dude. I'm just making a button."**

- 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.



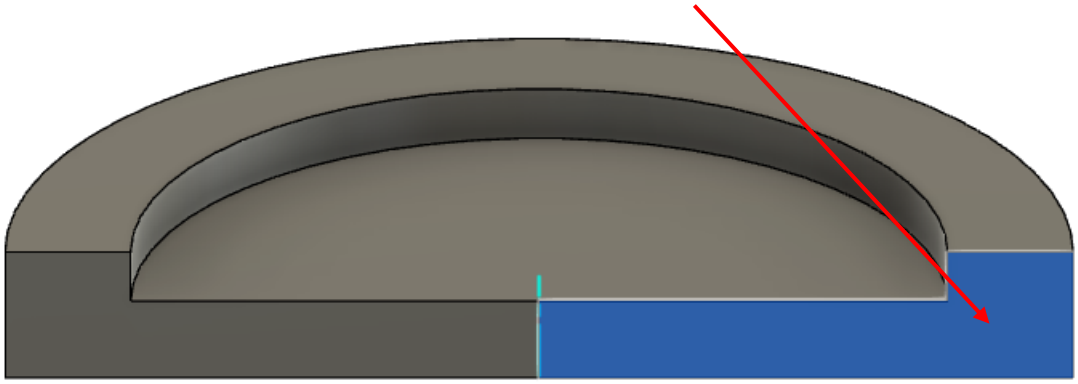
In many projects, the bottom rhombus is selected, but because we are starting with a cross-section we are starting with the front plane.

- zoom in as shown below. The scale labels can give an idea of how far one is zoomed in. A button will likely be around 1" or less in diameter. The **View Cube** should indicate you are sketching on the **X-Z Plane**.



The button drawing shown here is just an example. Your button will be different!

Much of the work to create the button is creating the **Sketch of half of the button's cross section**.



Note that the cross section, which will be sketched is simplified. It doesn't have filleted edges, which will be done later in the process.

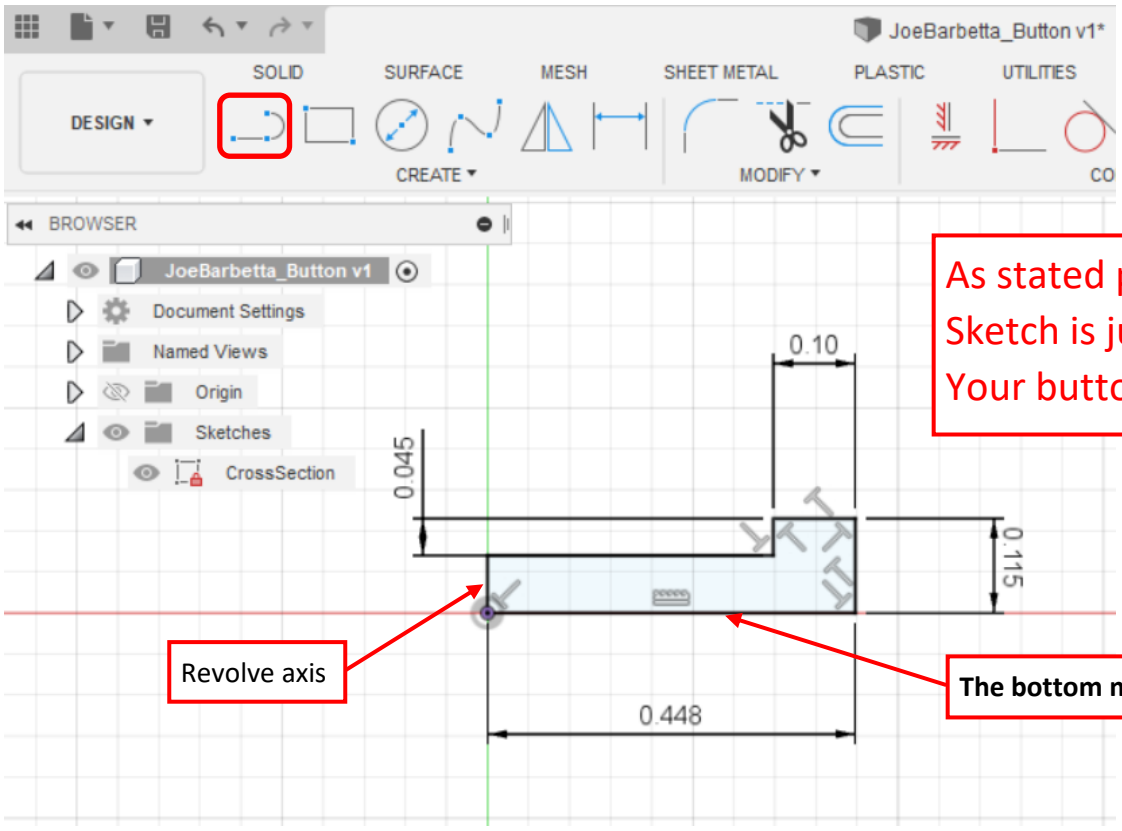
Below is an example of sketch created for the cross-section of a button based on made up measurements. It is actually just one half of the cross-section, which is all that is needed because a later step will **Revolve this Profile** around the line segment indicated as the **Revolve axis**.

An example such as this would be created be selecting the **Line tool** and clicking on the Origin and typing a dimension before additional clicks to complete a closed figure. When closed the interior will turn light blue indicating a Profile was created. Once the profile is completed, click **Finish Sketch**.

It is important that the bottom edge of the profile is straight so that the bottom surface of the button is flat.

- create a Sketch for your button using the **cross section shape for your button and your measurements**.

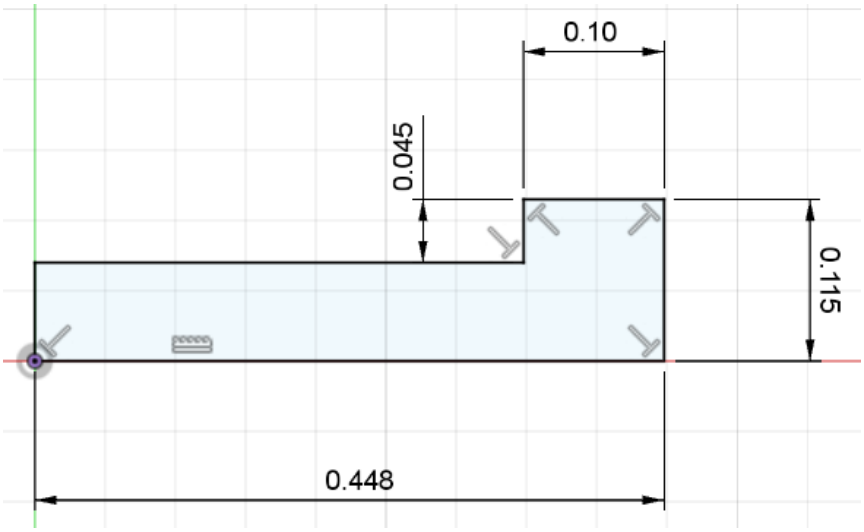
Before doing so, see the next page for possible cross section shapes.

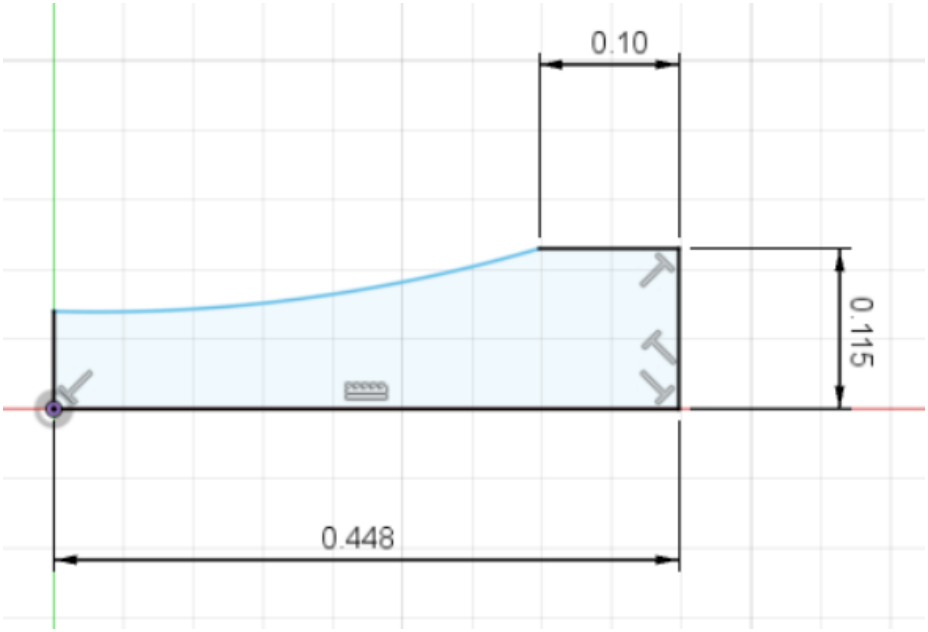


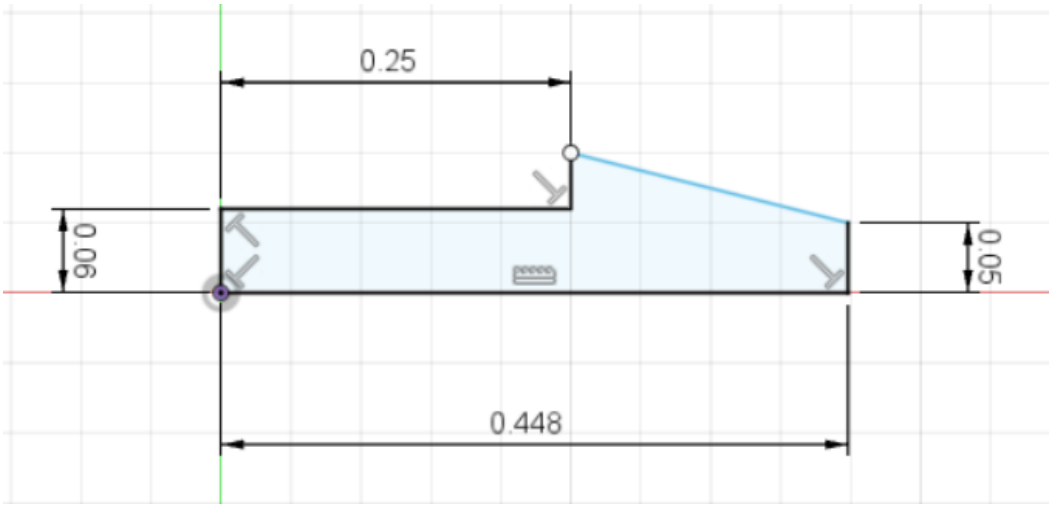
As stated previously, this Sketch is just an example. Your button will be different.

Here are examples of other sketches that could be appropriate for other button cross-sectional shapes.

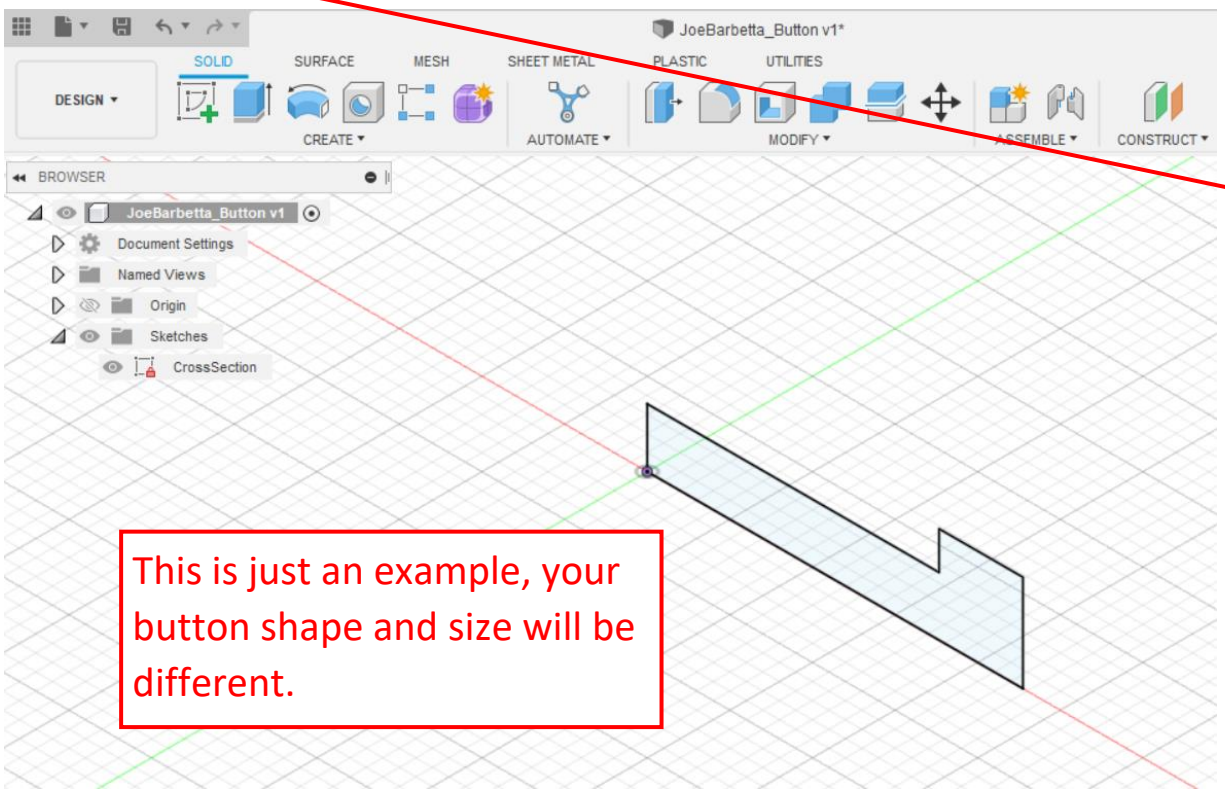
The button Sketches shown here are just examples. Your button may have a similar or different shape. Your dimensions will be different!





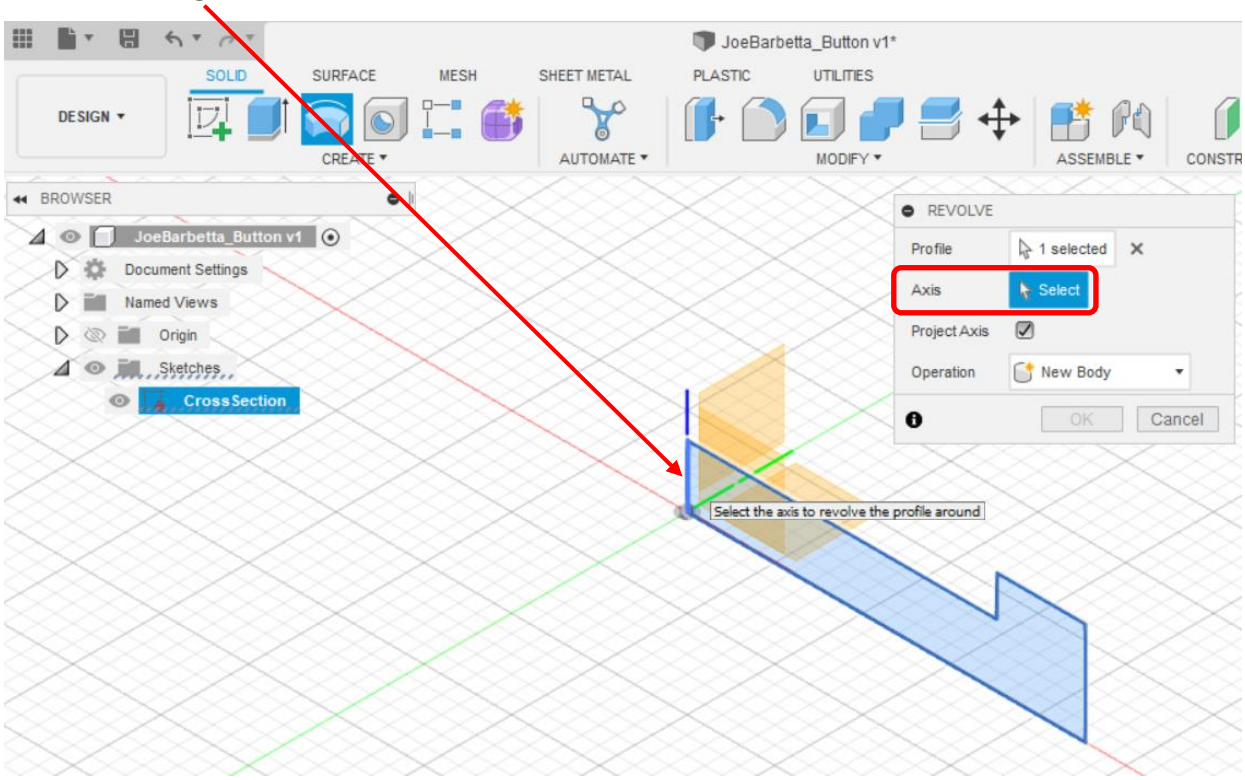


- once your Sketch is complete, click on the **Home icon** at the **View Cube** to reset to the Home View

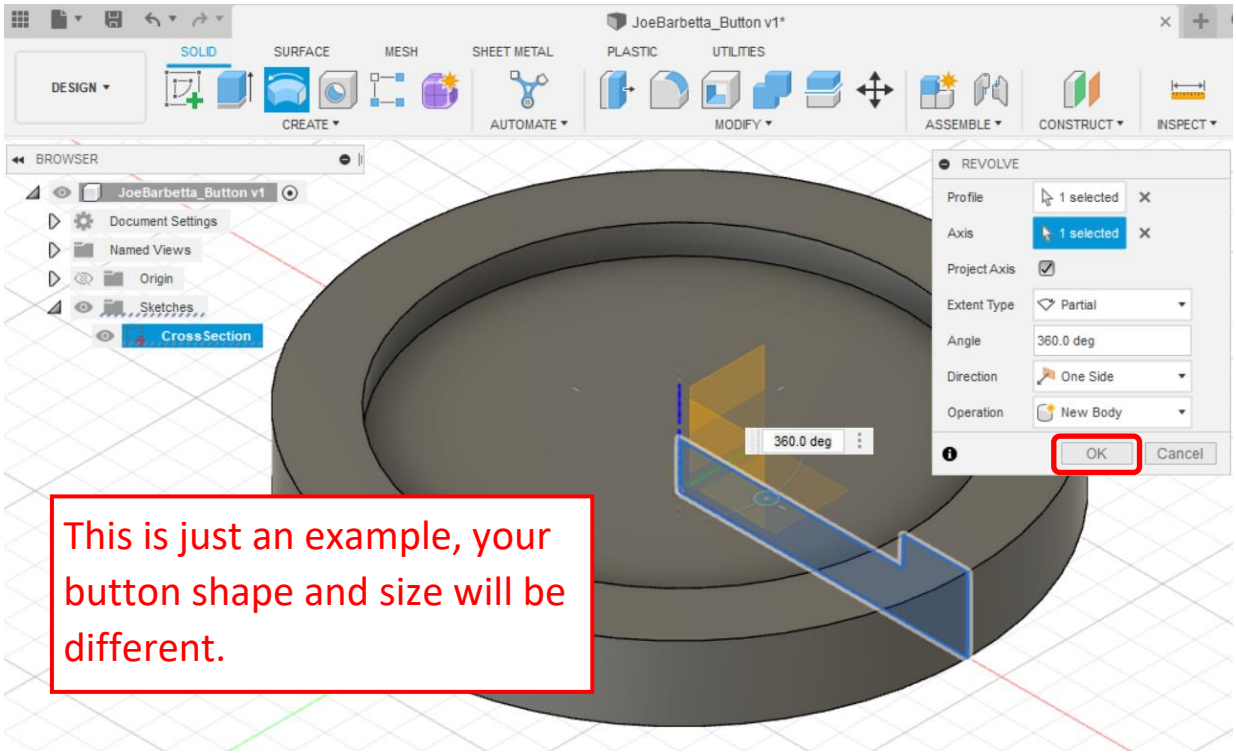


Using the Revolve Tool

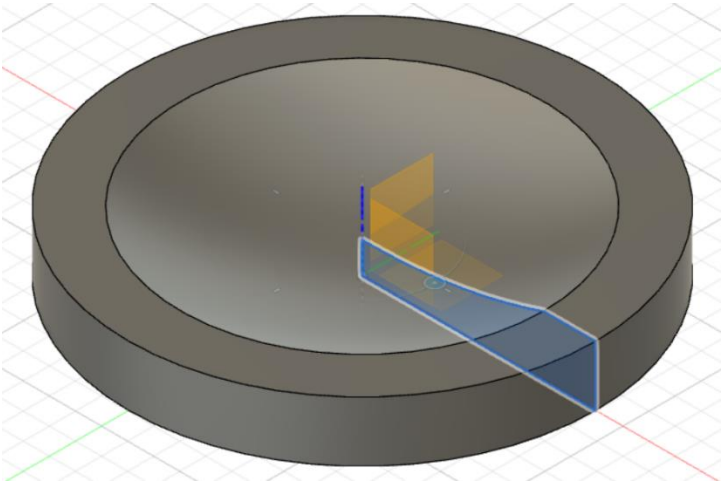
- select the **Revolve tool**. If it's not there find it in te **CREATE** pull-down menu.
- ensure **Select** is highlighted in blue next to **Axis**
- click on the **edge** as shown

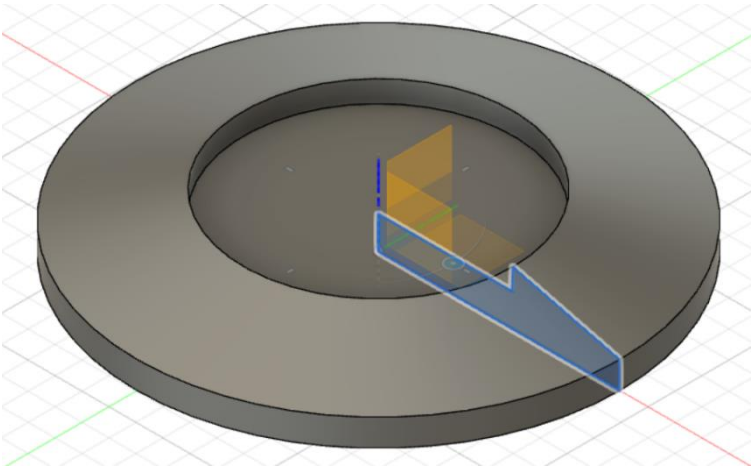


As you can see, this is a really neat tool. By default, the angle is 360 and thus the profile is fully revolved.
- click **OK**



Here are other examples of revolved Bodies for other button shapes.

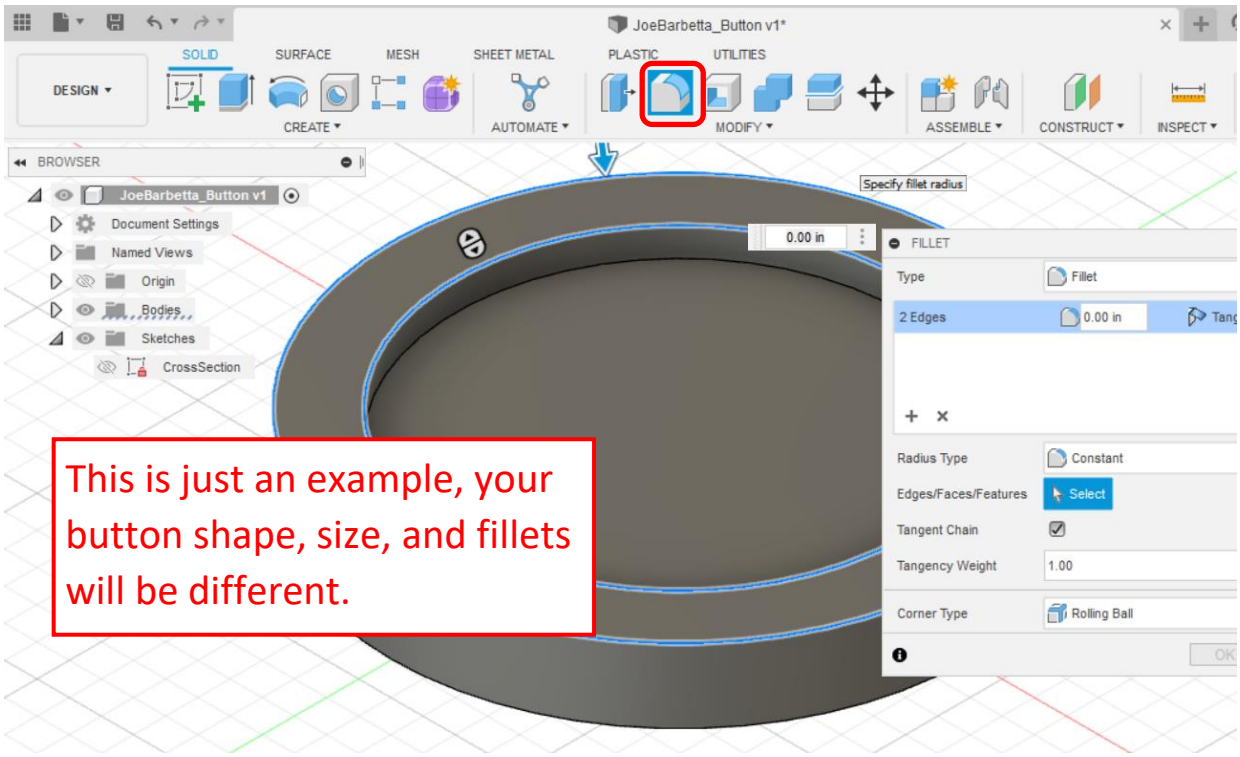




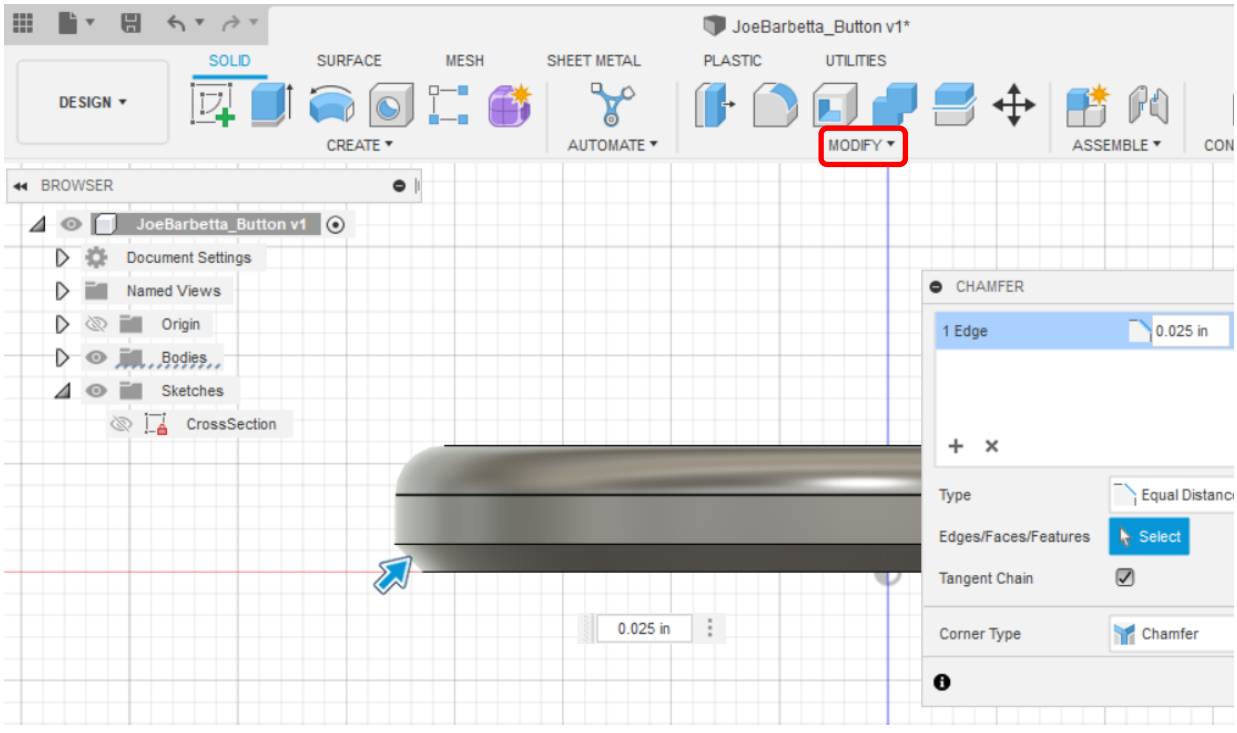
Adding Fillets and Chamfers

- select the **Fillet tool** (pronounced with a hard T and not like the cut of fish)

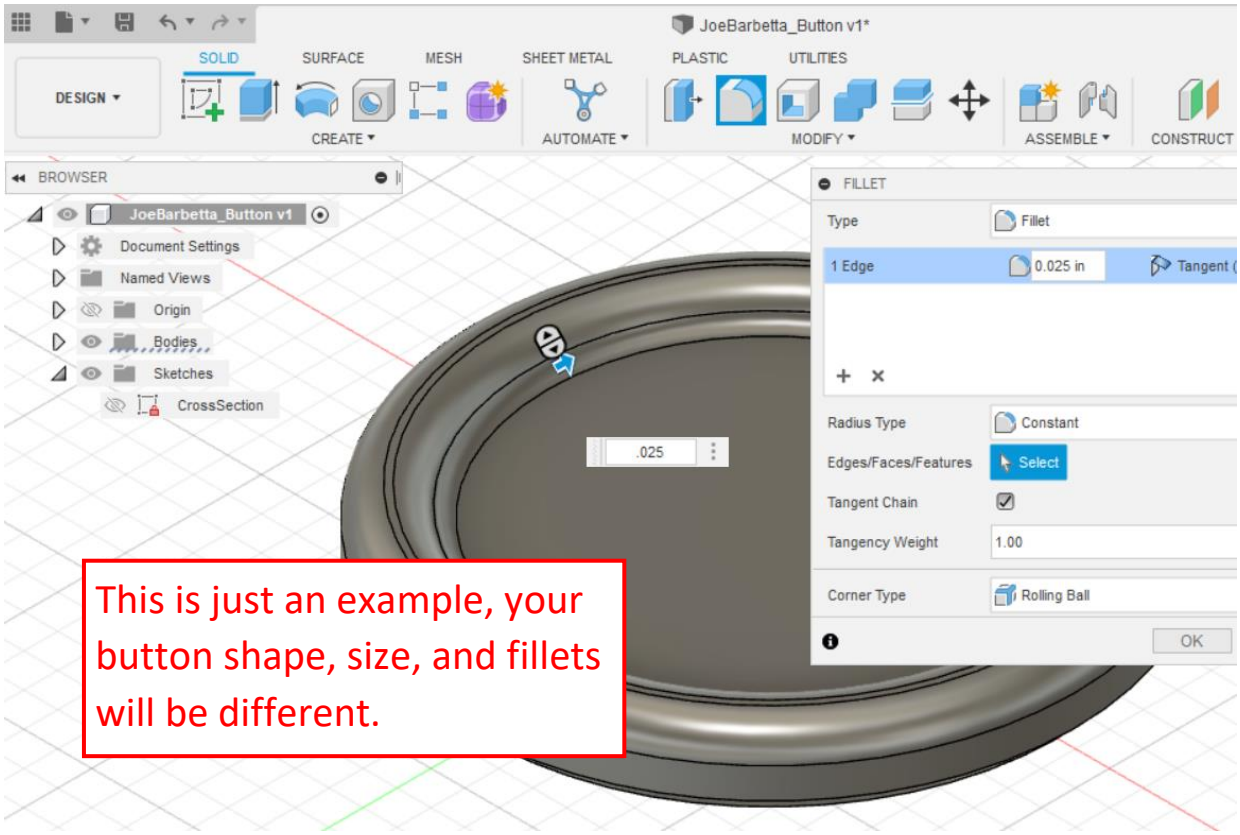
- click on one or more **edges** and type a value, e.g. 0.05, to match that of your button. Different values can be tried until the fillets resemble that of your button. If edges should have different fillet values, a single edge can be selected, a value entered, and OK clicked and then repeated for the other edge(s). Do Not fillet the bottom edge.



- within the **MODIFY** pull-down menu select the **Chamfer tool**
- click on the **bottom edge** and enter a small value, such as **0.025**, and click **OK**. Leaving a sharp edge may make it more difficult to slide the button into a button hole. A Chamfer results in a 45 degree angle that doesn't violate the 45 Degree Rule of 3D Printing.



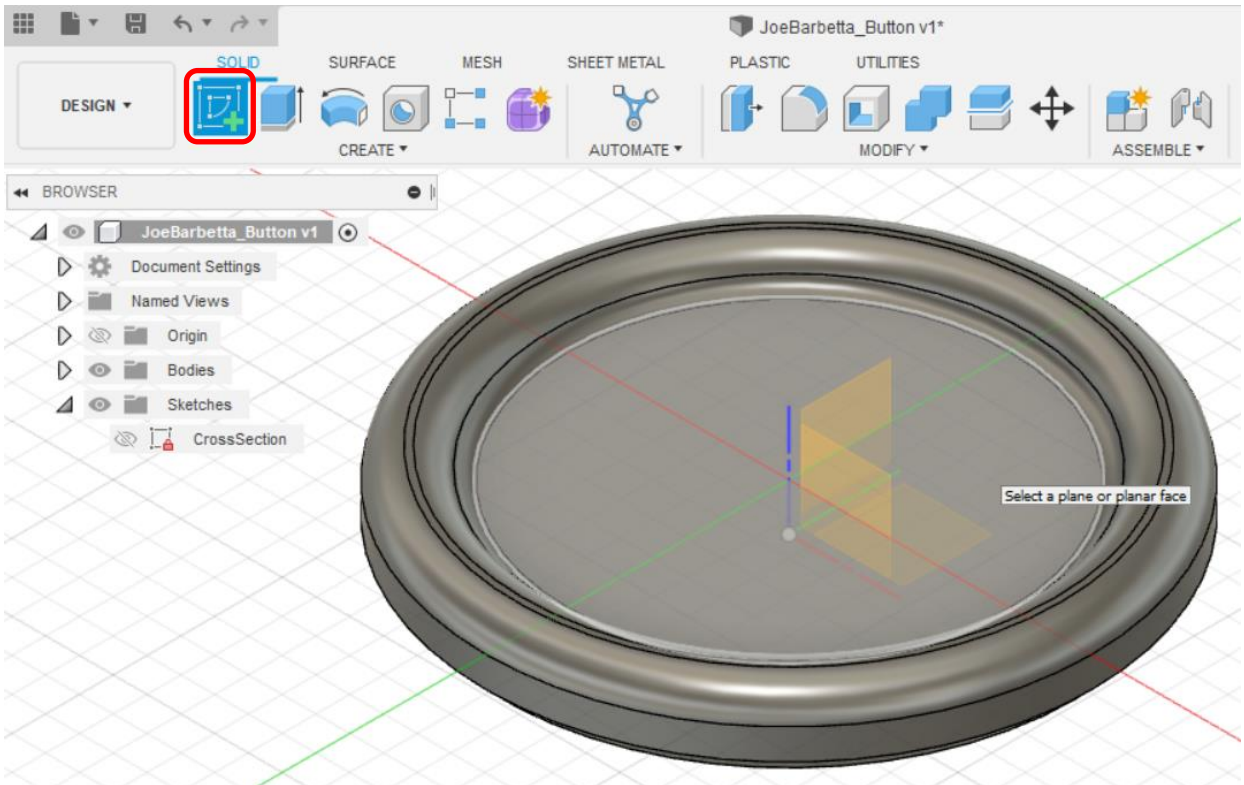
For aesthetics, any internal angles can be filetted also.



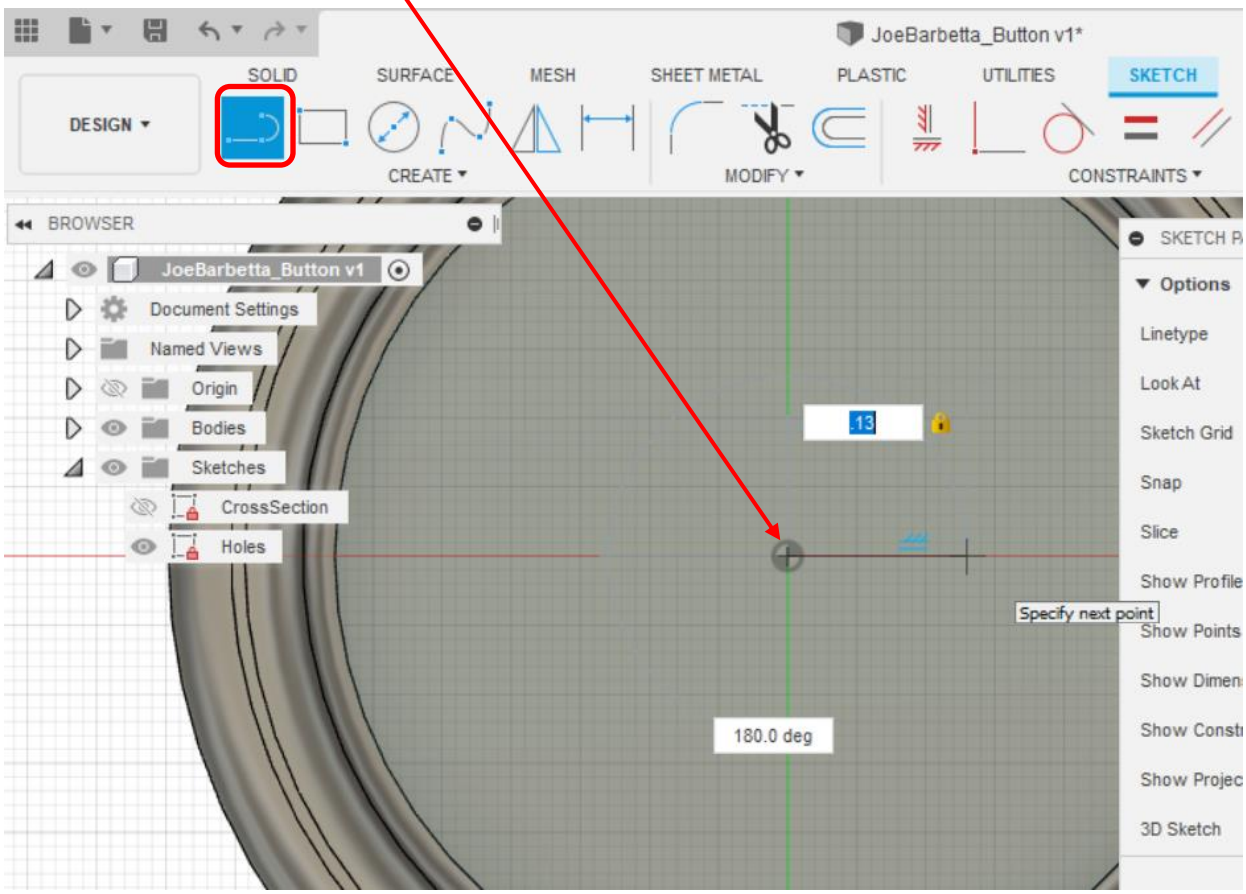
Creating the Thread Holes

- select the **Create Sketch** tool and select the top center Face of the button

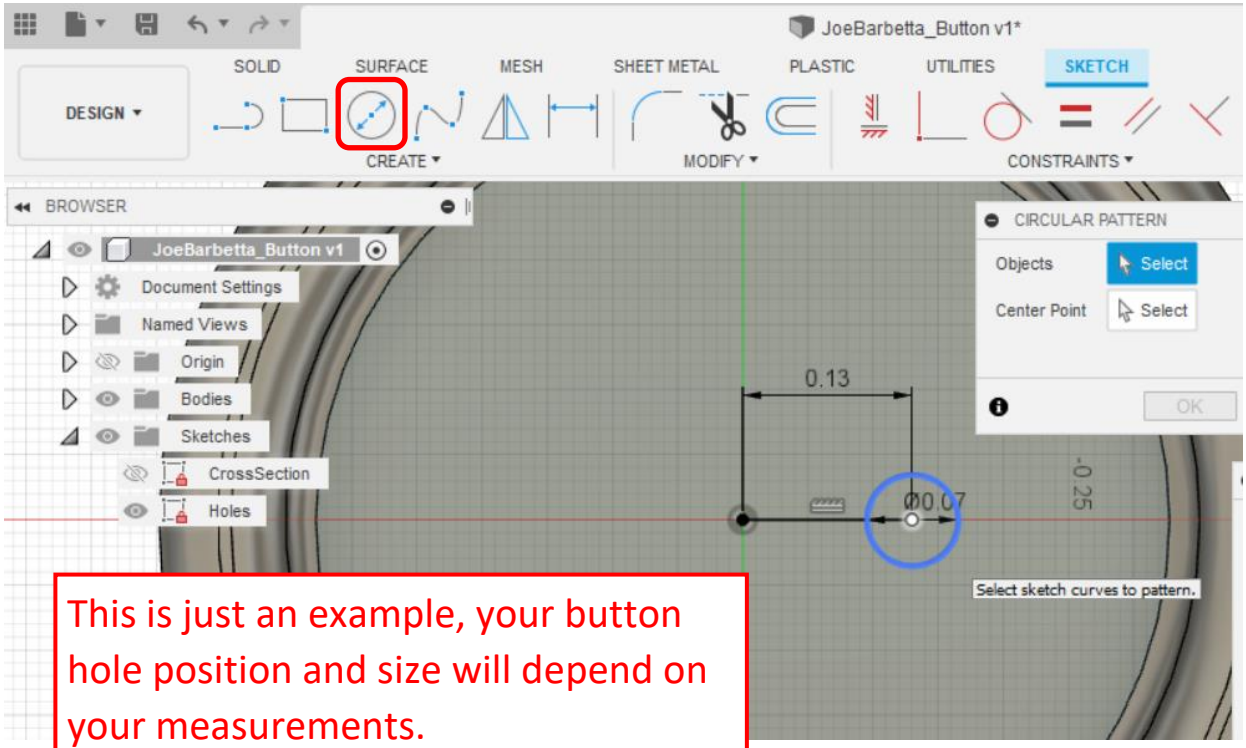
If your button has a rounded top, you may have to create this Sketch on the bottom of the button.



- zoom into the center of the face
- create a **Line** from the **center (origin)** to the right and enter a value equal to the **Button Center to Hole Center** dimension that was determined from measurements.



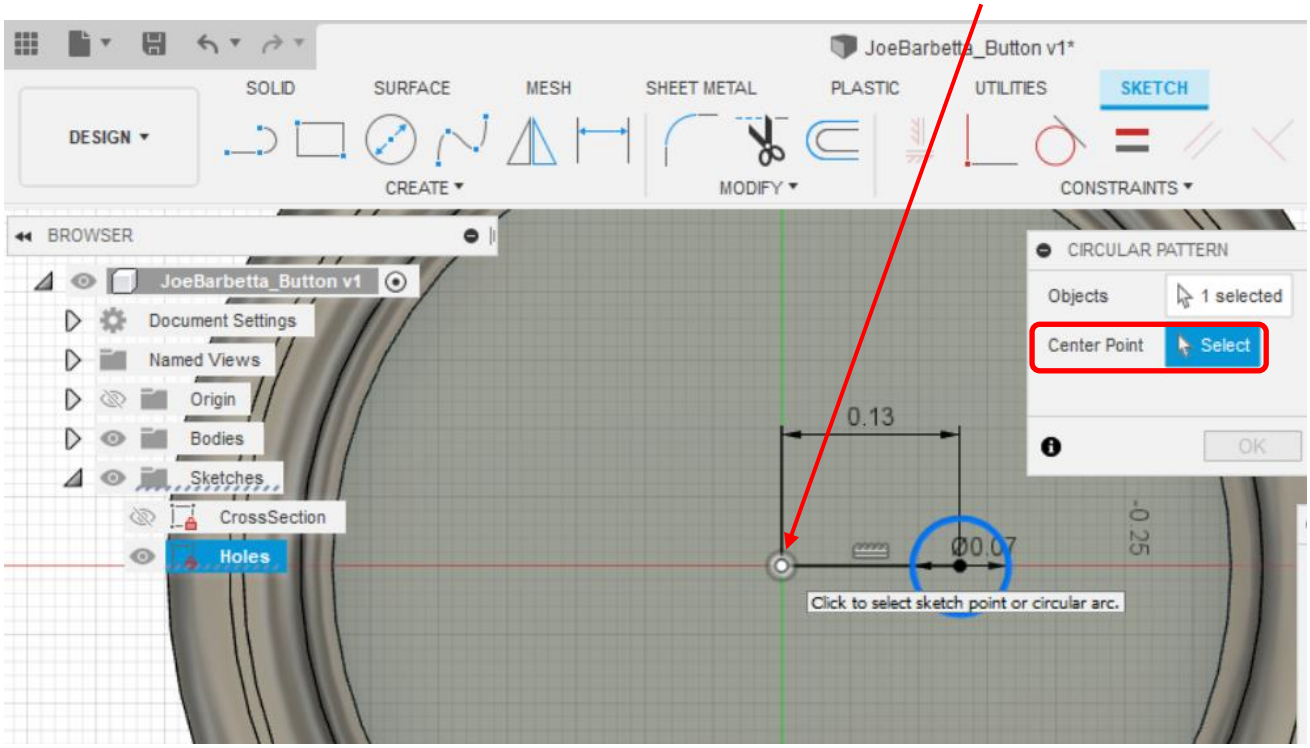
- select the **Circle tool** and click on the end of the line just created, enter the measured **Hole Diameter**, and click **OK**



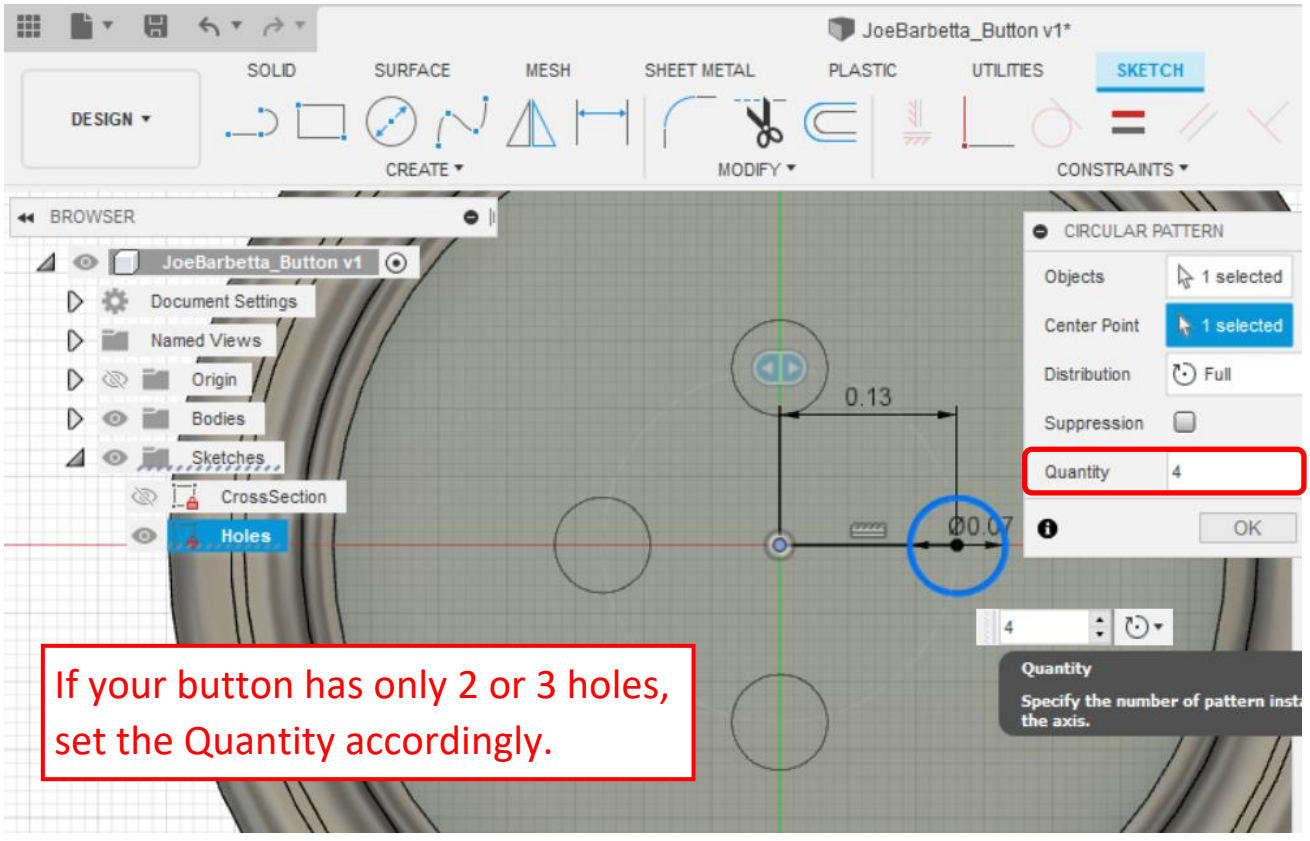
This is just an example, your button hole position and size will depend on your measurements.

Using the Circular Pattern Tool

- in the **CREATE** pull-down menu select **Circular Pattern**
- click on the Circle just created
- ensure **Select** next to **Center Point** is highlighted in blue and then click on the **Origin**

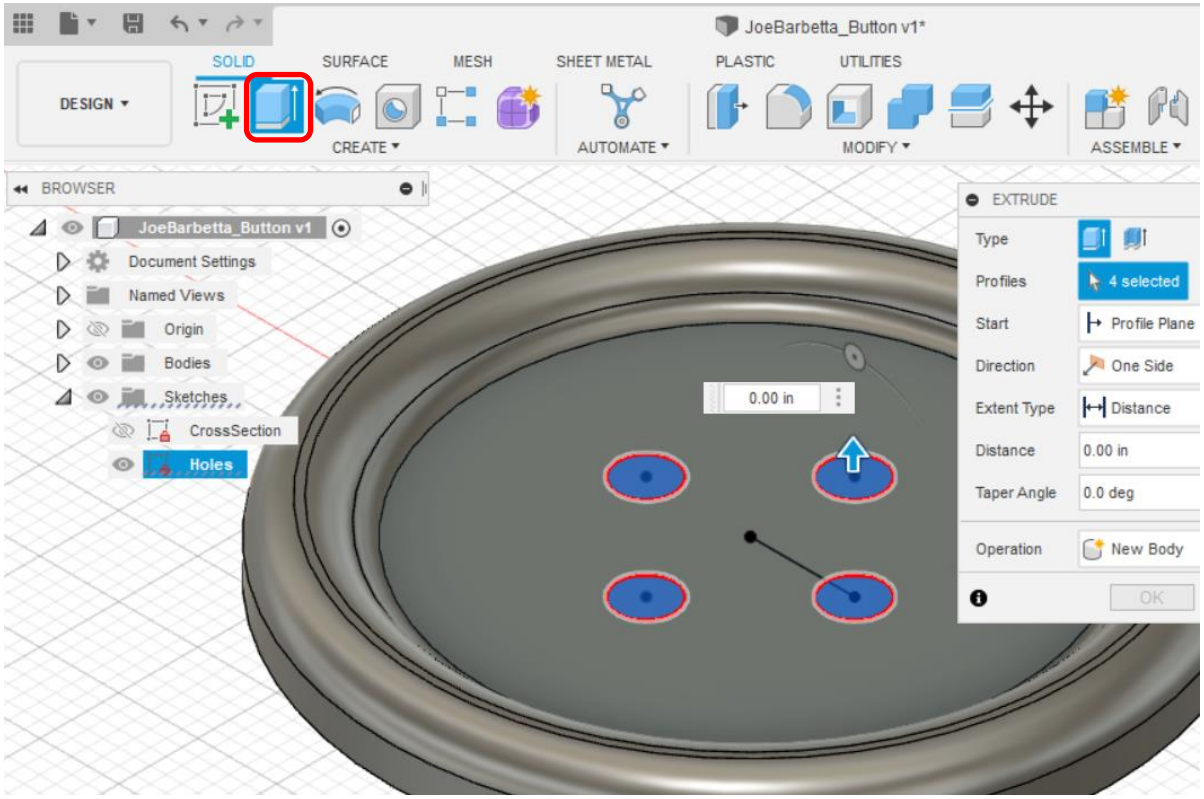


- enter the **number of holes** to create. Your button will likely have 4 holes, but it may also have 2 or 3.
- click **OK** and then **Finish Sketch**

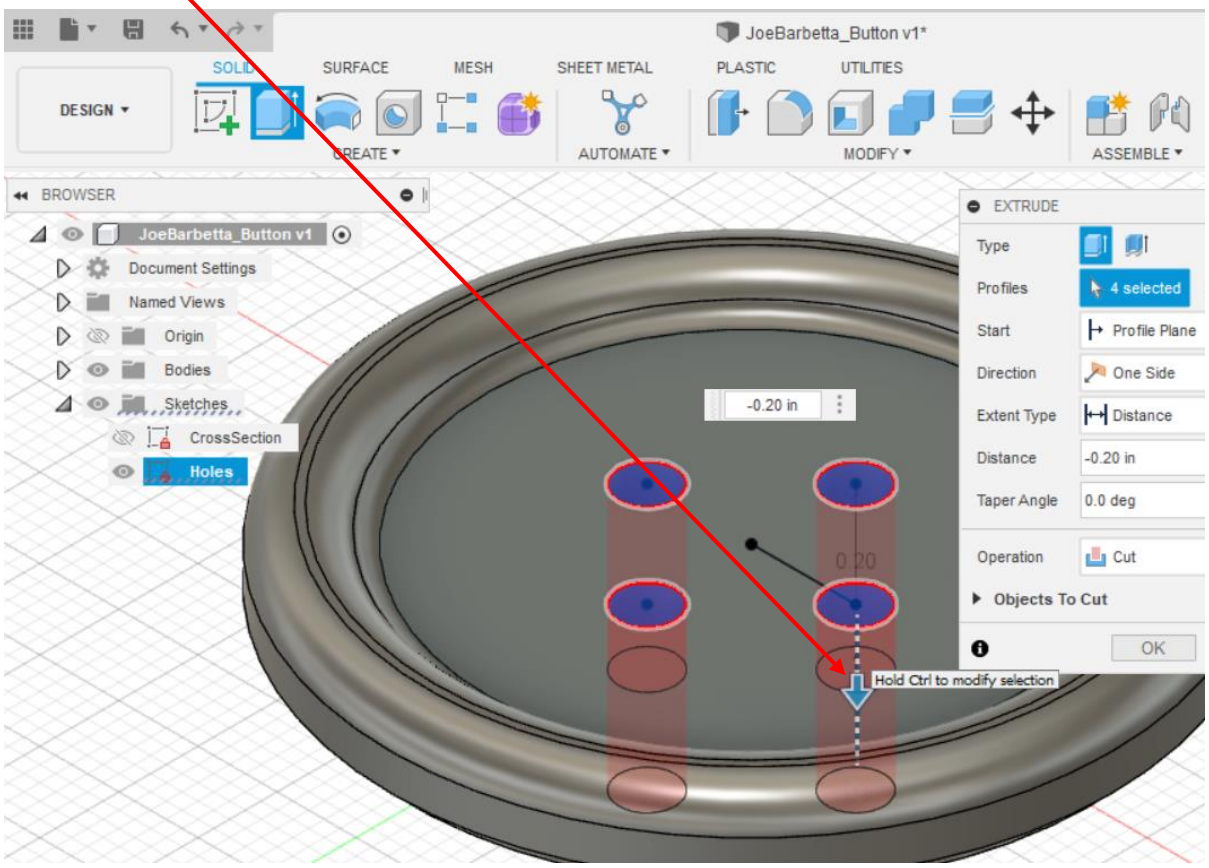


Extruding the Holes

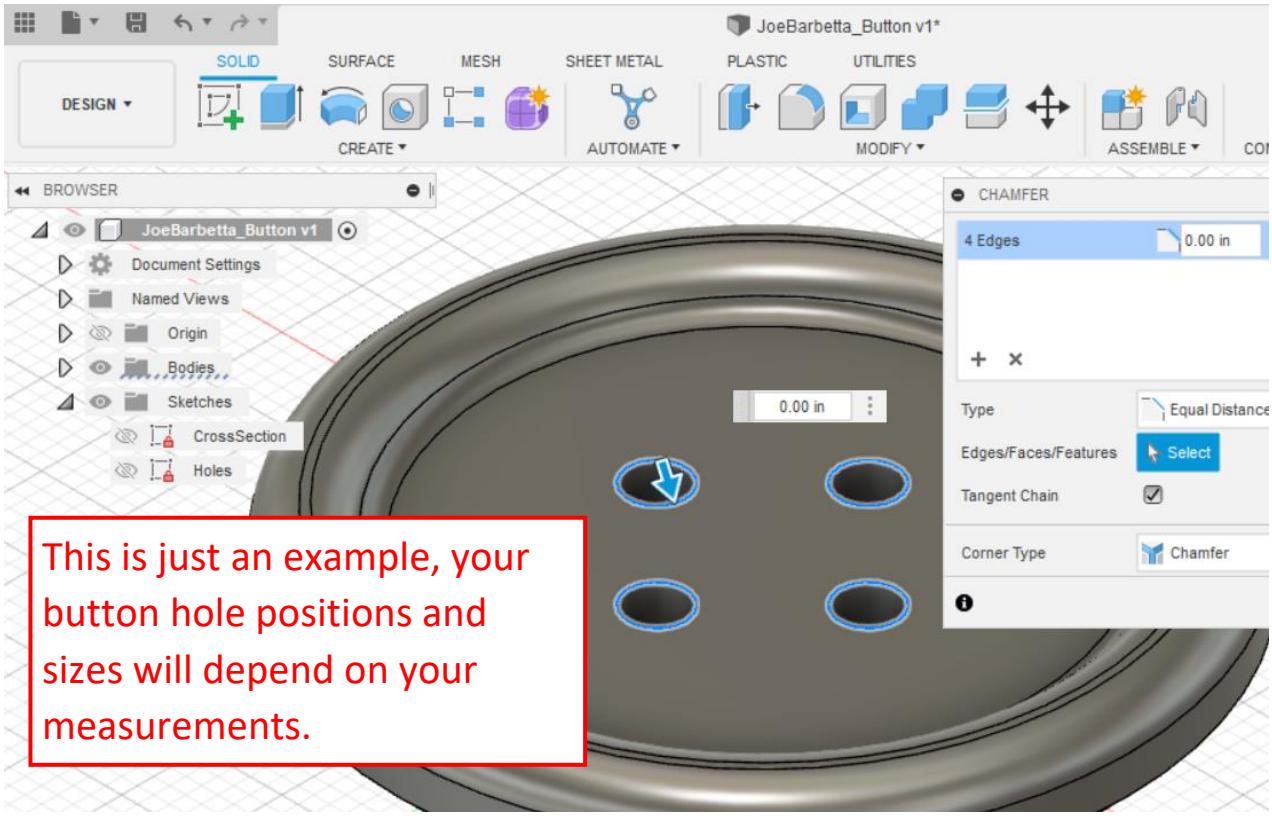
- click on the **Extrude** tool and then on the **interiors of the circles**



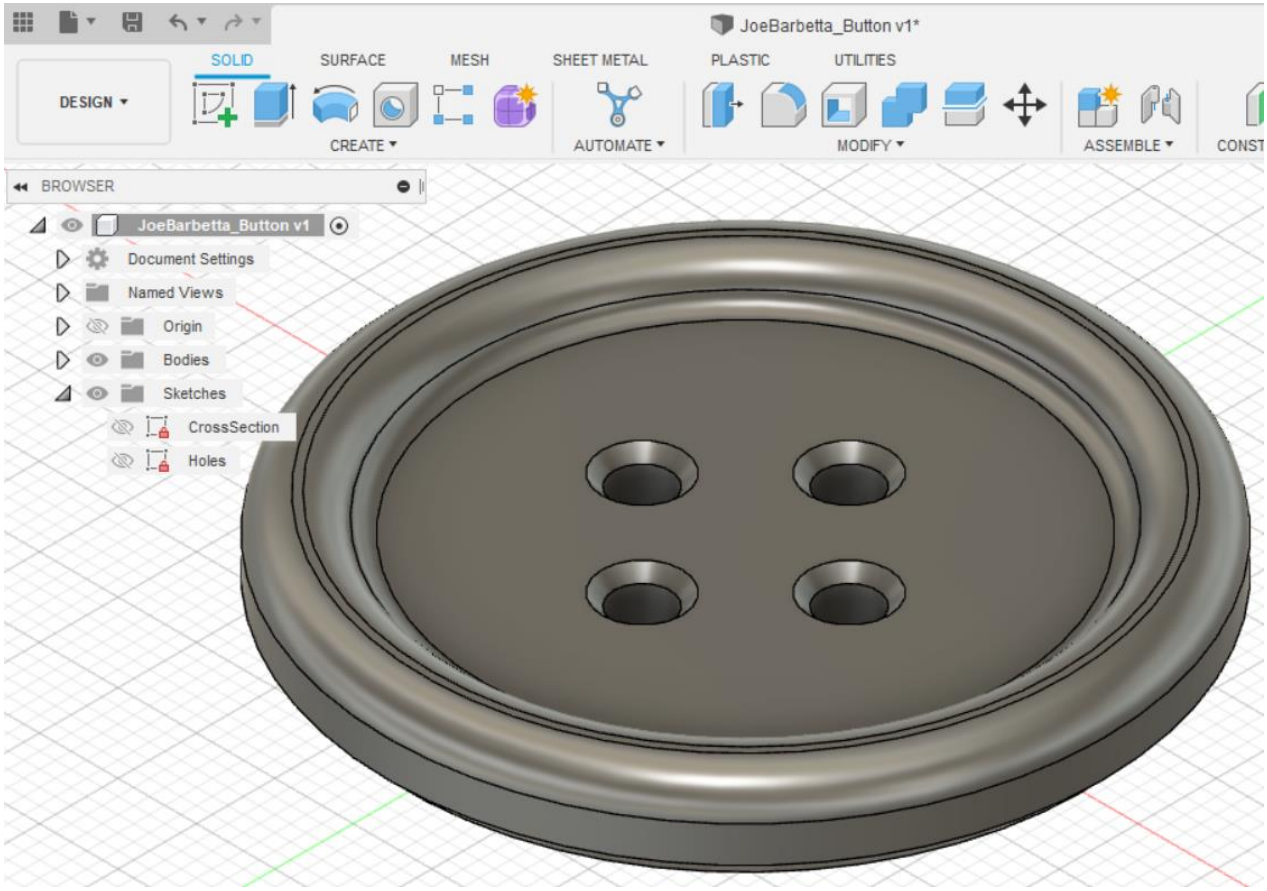
- pull the **blue arrow** down so that the holes are cut through the button and click **OK**



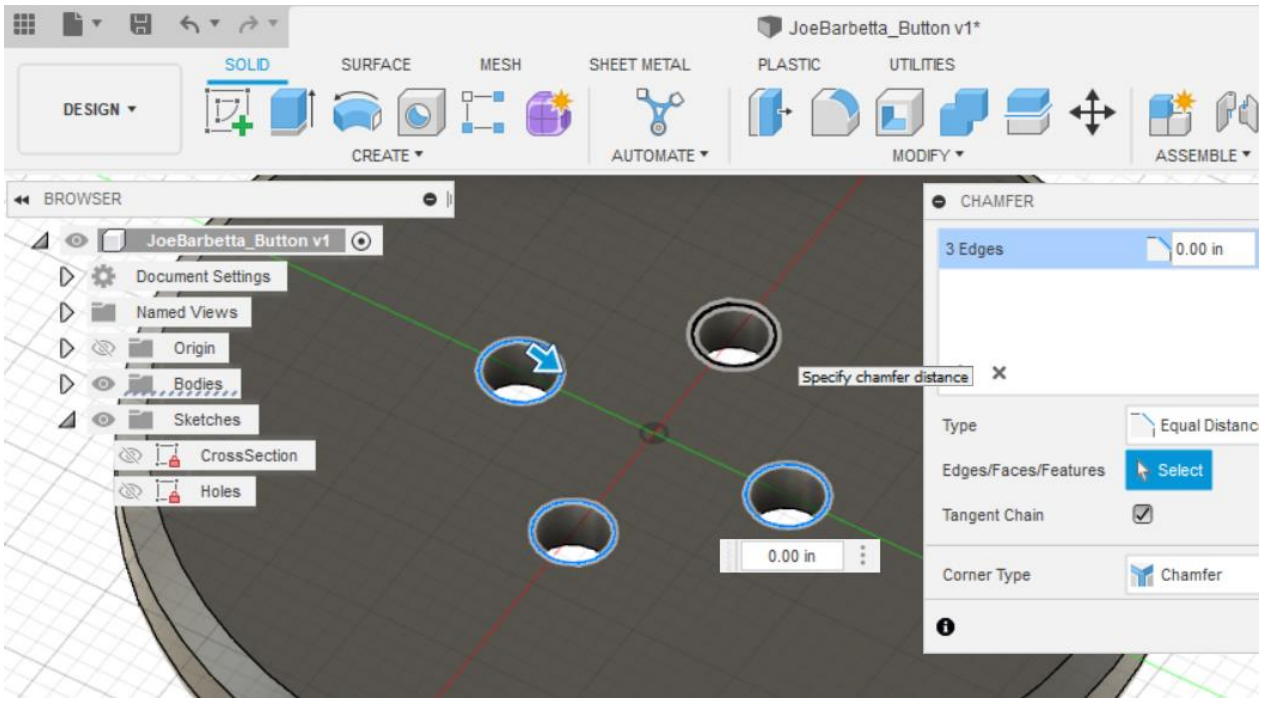
- in the **MODIFY** pull-down menu select **Chamfer** and click on the **edges** of the holes just created
- type **0.015** and click **OK**. This will make it easier to pass thread through the holes.



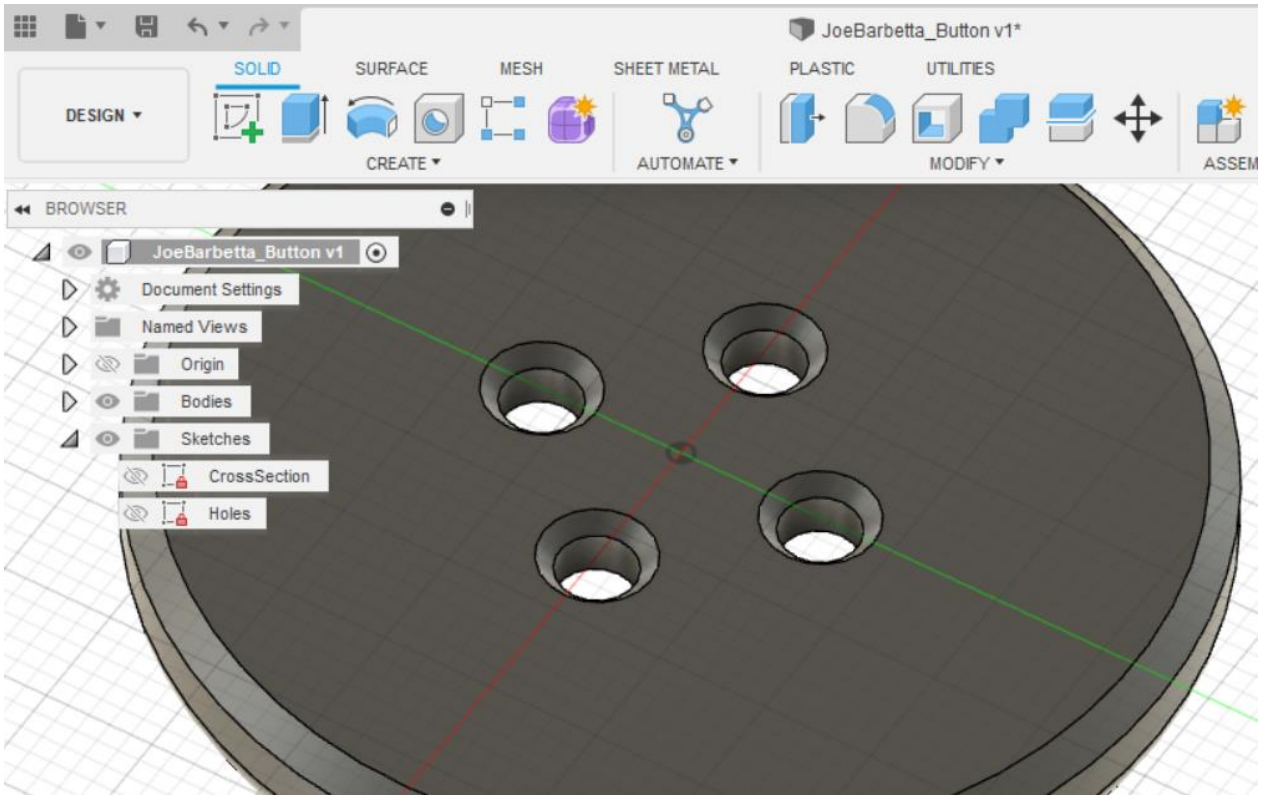
This is the result of the chamfered holes.



- change the view to access the bottom of the button and add a 0.025 chamfer to the bottoms of the holes.

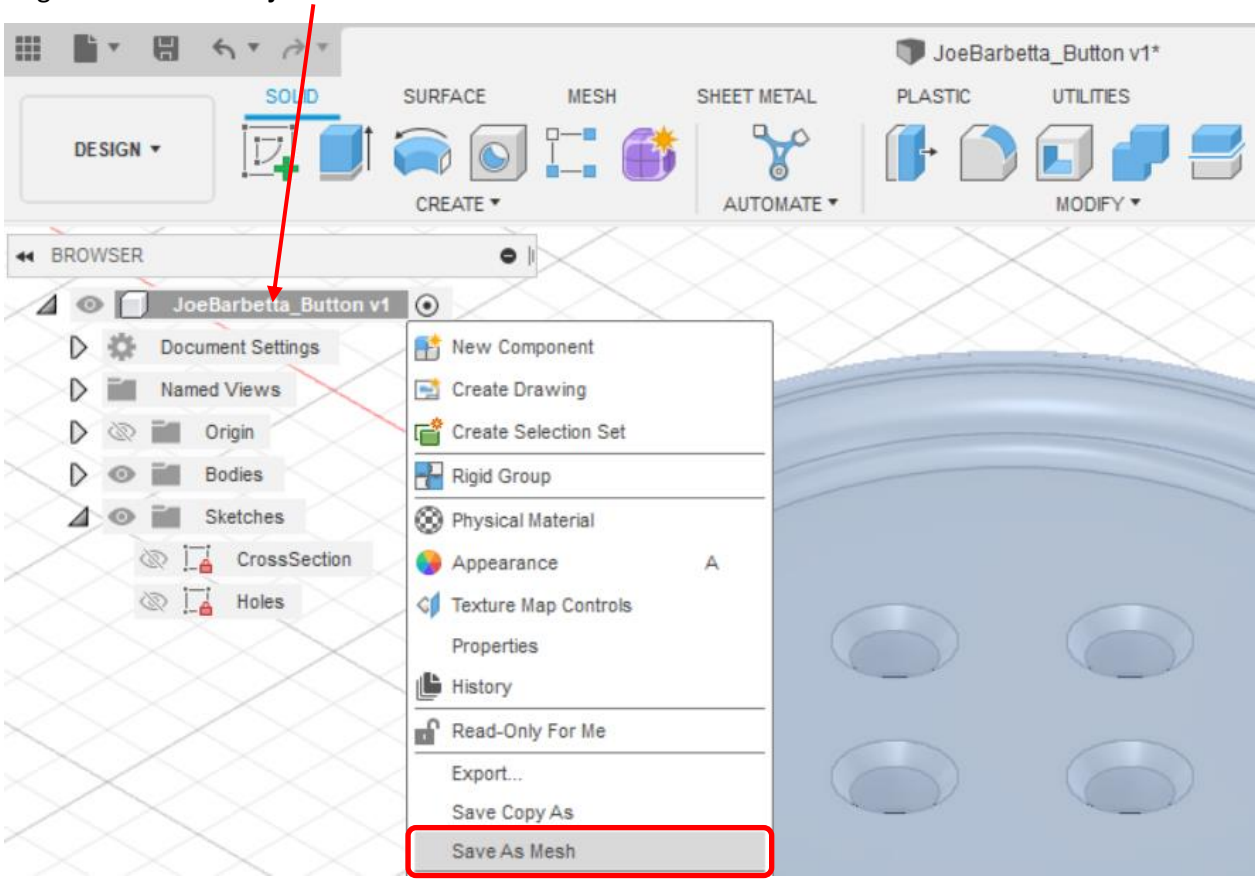


This is the result of the chamfered holes.

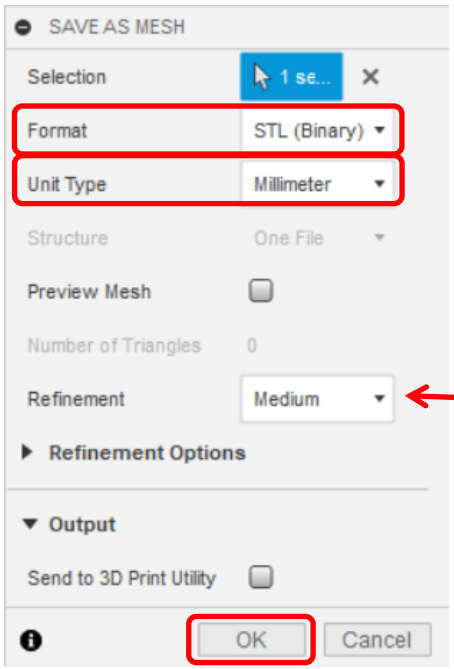


Exporting a STL File

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



This window will show each time Save As Mesh is used. Ensure that **Format is set to STL (Binary)** and **Unit Type is set to Millimeter** and then click **OK**. You will then be prompted to save the file. The default location is the **Downloads** folder.



Refinement can remain at Medium. If one wants the optimum printing for curved surfaces, High can be used. The difference is very subtle.

An STL file comprises coordinates in 3D space for thousands of connected triangles that define the outer geometry of the object and thus any curved surfaces are approximated. The High setting generates more triangles to better approximate curved surfaces at the expense of larger STL files and increased processing times.

- click **Save**. By default Fusion 360 will save the STL file to the **Downloads** folder.