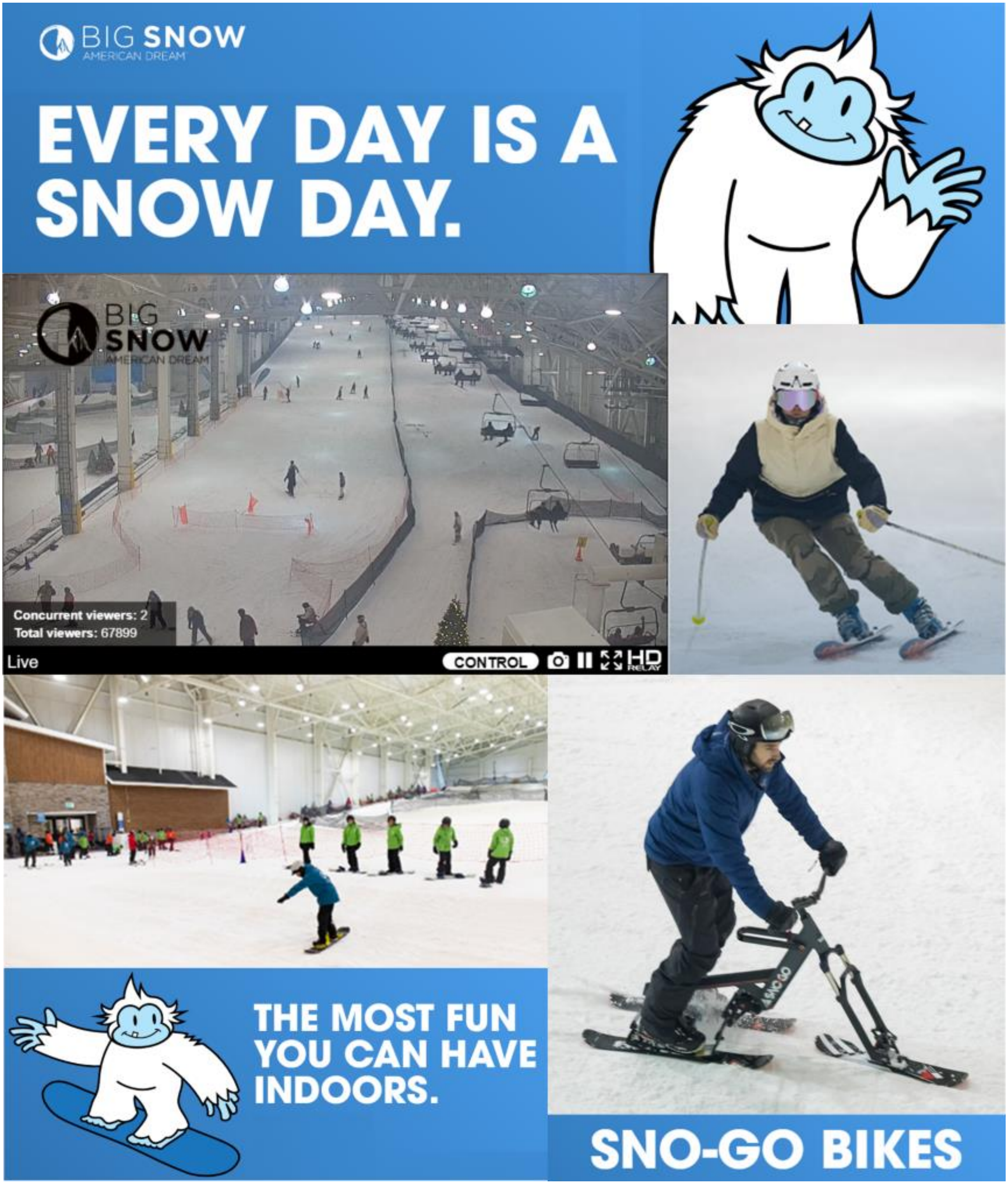
Mr. Barbetta's ***"I don't care about 3D Printing. I just want the art credit."*** class

Design a Low Taper Fade guide

Today's Lesson is Sponsored by Big Snow at the American Dream Mall

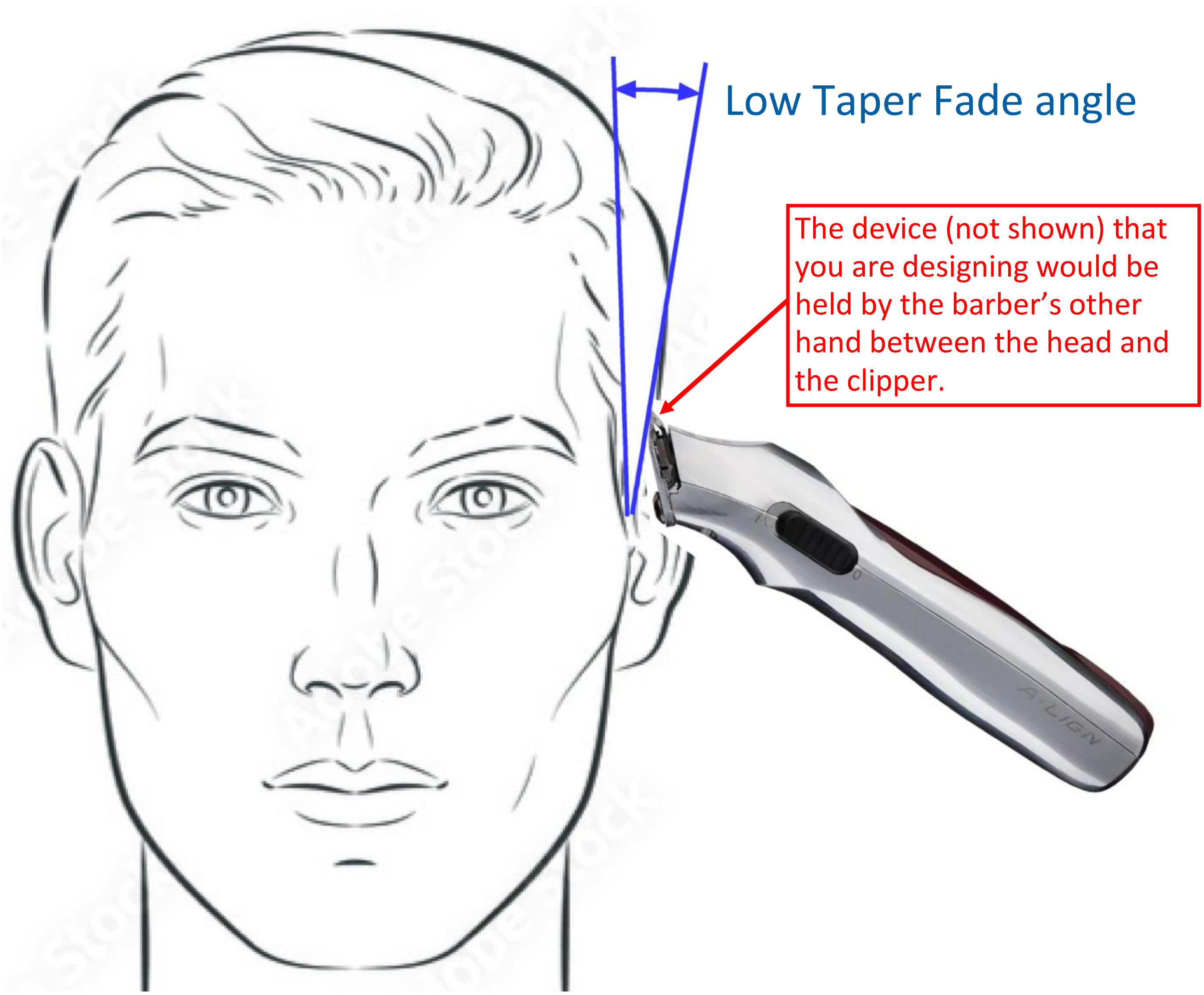


Contents

Product requirements

A device must be designed that can be placed against the head and act as a guide for a hair clipper to achieve an accurate Low Taper Fade angle. The device may resemble a comb comprising teeth with a side profile to define an angle with respect to the side of a head. This device must include a handle to allow the user to maintain its position against the head when using the hair clipper. Your initials should be extruded on the handle.

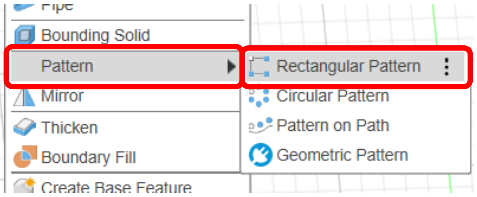
The instructions that follow are only to inform one of methods that can be used. Any objects depicted in these instructions is just a general object. YOUR DESIGN WILL NOT LOOK LIKE THIS GENERAL OBJECT!



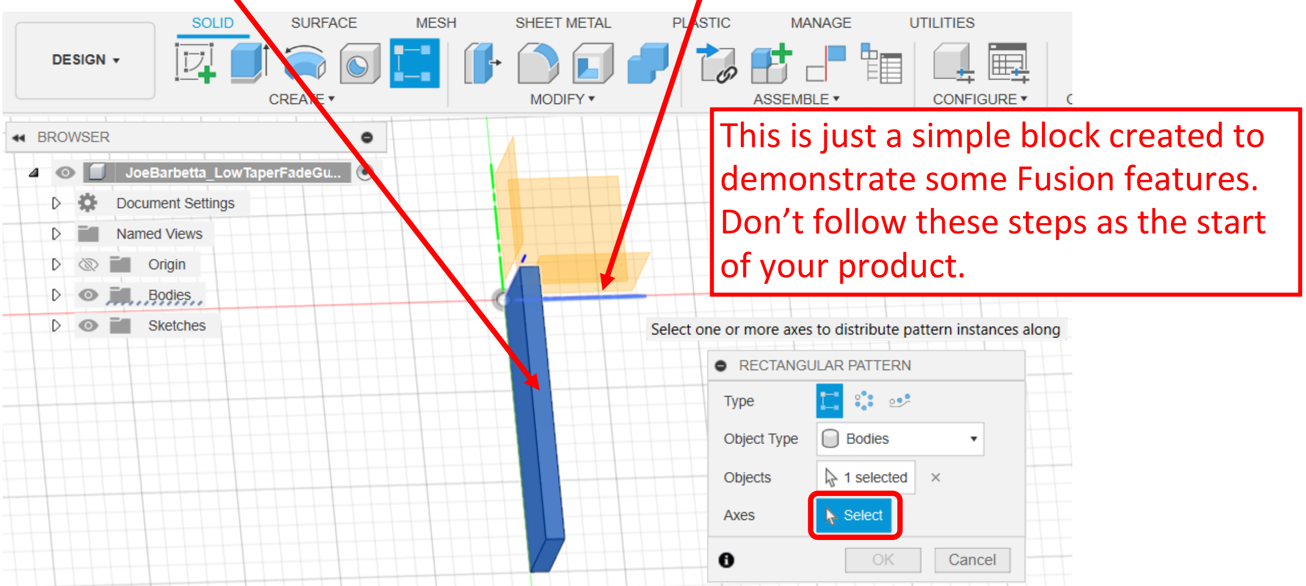
Design Strategy – Rectangular Pattern

The teeth of a comb or any repeated element can be designed once and then the **Rectangular Pattern** tool can be used. This is just an option. Your design strategy may use a different method. As an example, a simple block will be repeated.

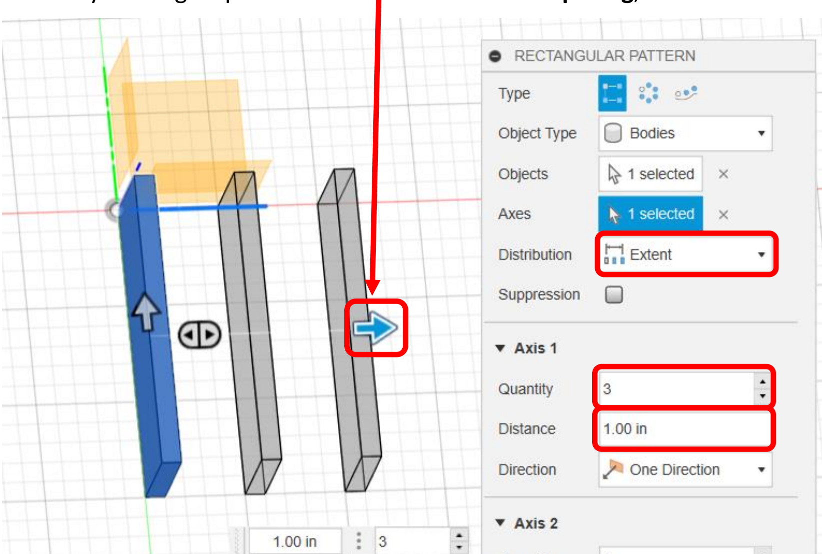
- from the **CREATE** menu select **Pattern** and then **Rectangular Pattern** near the bottom of the menu list



- click on the **body** to be repeated
- click on **Select** for **Axes** and then click on an **axis**. Here the **x-axis line** near the origin was clicked on. However, another axis, an edge of an object, or a Construction axis could also be used.



- drag the **horizontal** or **vertical arrow** in the direction where the repeated bodies are desired. Here the horizontal arrow was dragged to the right.
- change the **Quantity** from the default of 3 to the number of desired bodies
- change the **Distance** to that desired as well. If the **Distribution** is set to **Extent**, as shown, the Distance will be that from the original to the last body in the group. If the **Distribution** is set to **Spacing**, then the Distance is that of adjacent bodes. Click **OK** when done.



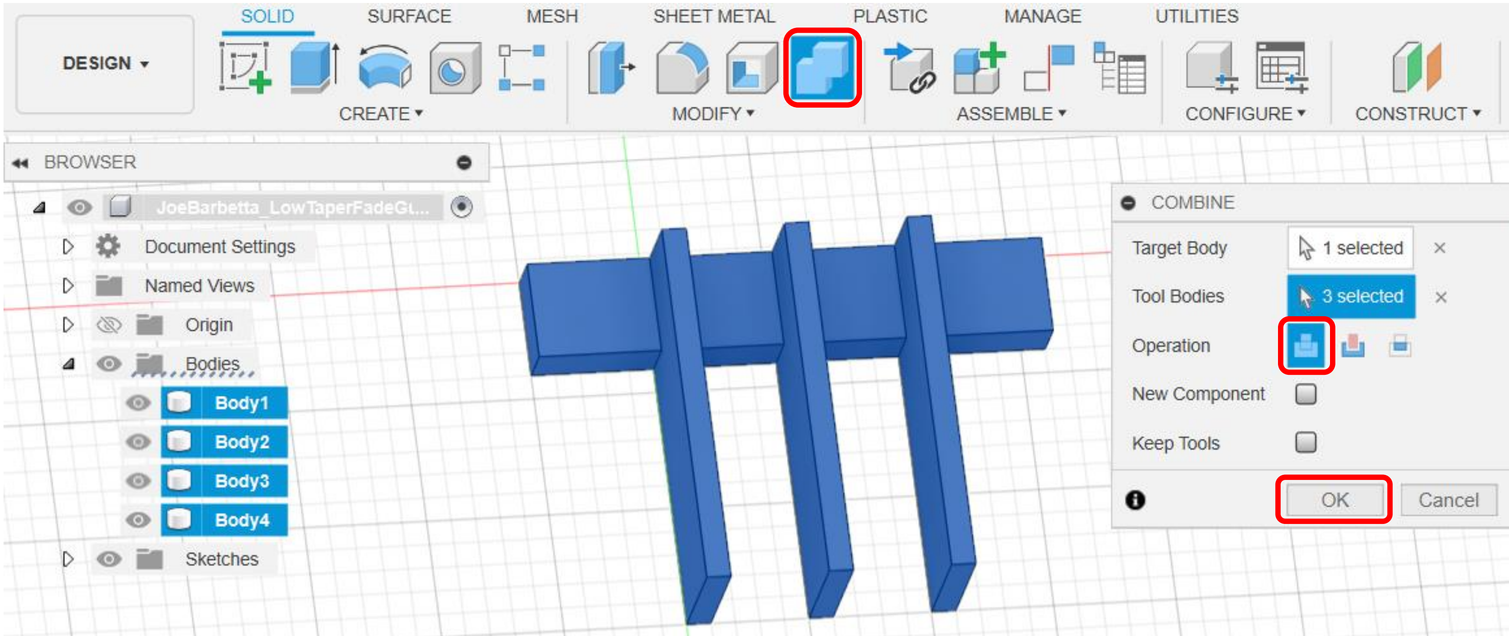
This is just a simple block created to demonstrate some Fusion features. Don't follow these steps as the start of your product.

Your starting body will be totally different and you may not even need to use the Pattern feature.

Design Strategy – Combining Bodies

If a **Sketch** is extruded so that the new **Body** will intersect with other **Bodies** and the extrude **Operation** is set to **Join**, the Bodies will automatically be combined into a single Body. However, if a Body is moved to a position where it intersects with other Bodies, the **Combine** tool must be used to combine them.

- select the **Combine** tool. If it is not visible, select it from the MODIFY menu.
- click on the bodies to be combined into a single body. Here all 4 bodies were clicked on. However, not all bodies have to be selected.
- by default the **Operation** should be **Join**
- click **OK** to complete the combining of the selected bodies into a single body



These are just some simple bodies created to demonstrate some Fusion features. Don't follow these steps as the start of your product.

Your product will look nothing like this and you may not even need to use the Combine feature.

DFM (Design for Manufacturing)

DFM (Design for Manufacturing) is an engineering term for the optimization of a design for the intended manufacturing process.

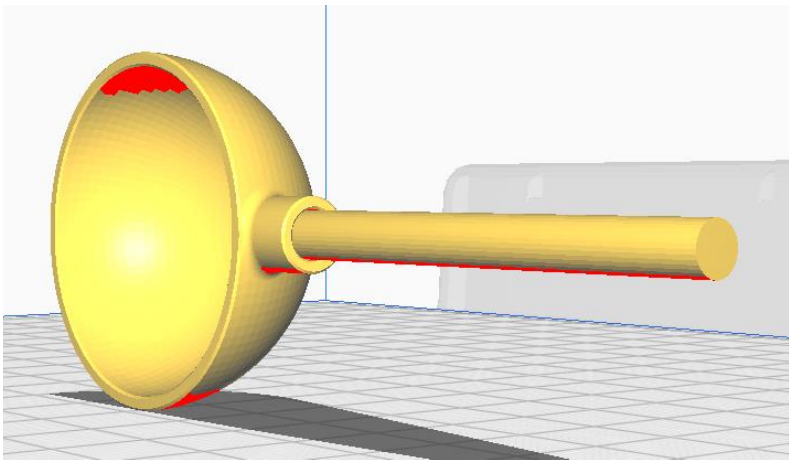
At first *Low Taper Fade LLC.* can start with a small production run using 3D printing. If the product starts to take off then *Low Taper Fade LLC.* can invest $5,000 in a single cavity aluminum mold. If sales start to skyrocket then $25,000 can be invested in a multi-cavity steel mold to really crank them out.

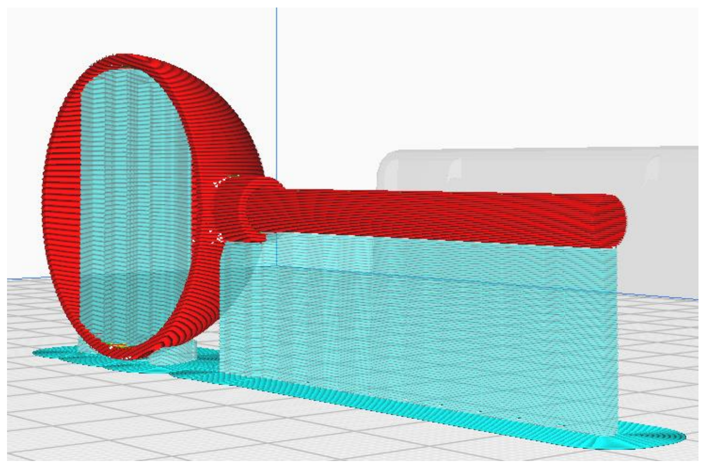
Whatever process we plan on using, we want to optimize the design for that process. For 3D printing, we will use two strategies. One involves adhering to the **45 degree** rule to eliminate the need for supports and the other will be a **brim block**.

Of course for production (high quantities) injection molding would be used and the issue of supports would not be relevant. However, if one wishes to use 3D printing for a *low production run*, it makes sense to optimize the process. One way to implement good DFM is to create a design that doesn't need supports.

Supports relate to extra support material to print elements of the part that are not on the build plate. One cannot print over air. There must be lower layers. Every slicer program, such as Ultimaker Cura, can generate these supports. The downsides of doing so include a longer print time, the use of more material, additional labor, and resulting rough surfaces where the support material connects to the part. Sometimes it is required for these rough surfaces to be sanded or scraped to remove remnants of the supports and to reduce roughness.

Below is an example of an anemometer arm (used to measure wind speed) imported into Cura. On the right side is the result of added support material, which Cura shows in blue. It should be noted that there are many options to creating supports. For example, an alternate support option is the use of *tree supports*.

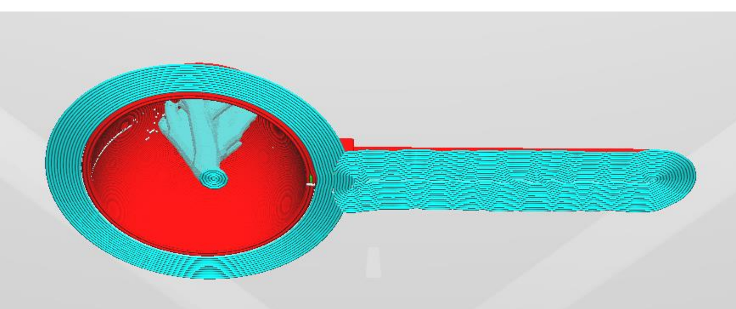


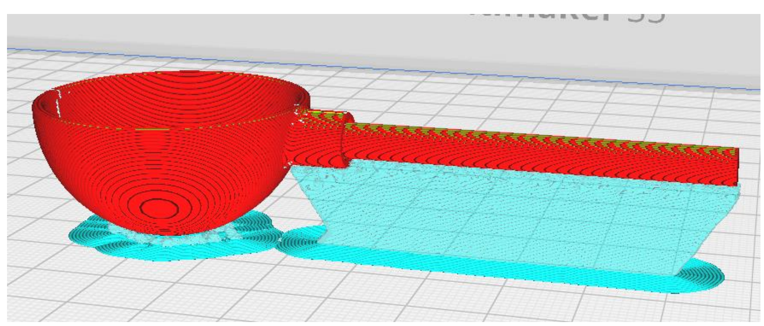


Below are the results of repositioning the cup.

On the left side the part has been rotated with the open end of the cup down on the build plate.   The view was be rotated in Cura to view the underside. Supports are needed inside of the cup and under the extension.

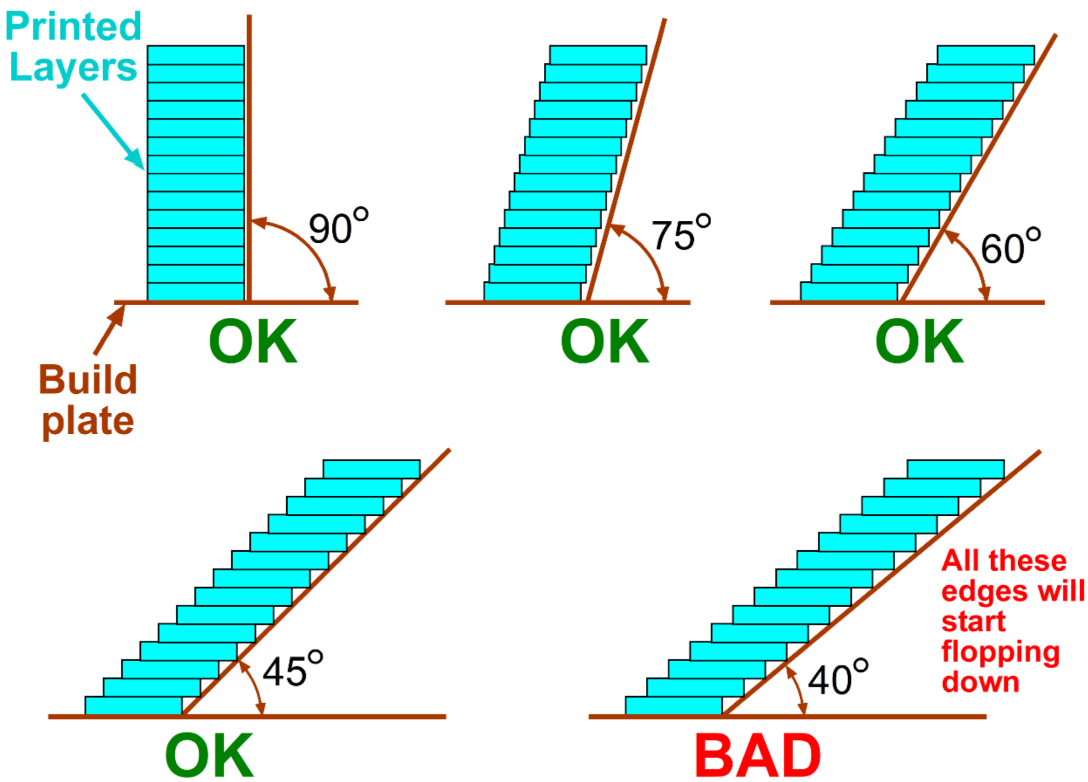
One the right side in another position of the cup with the necessary supports.

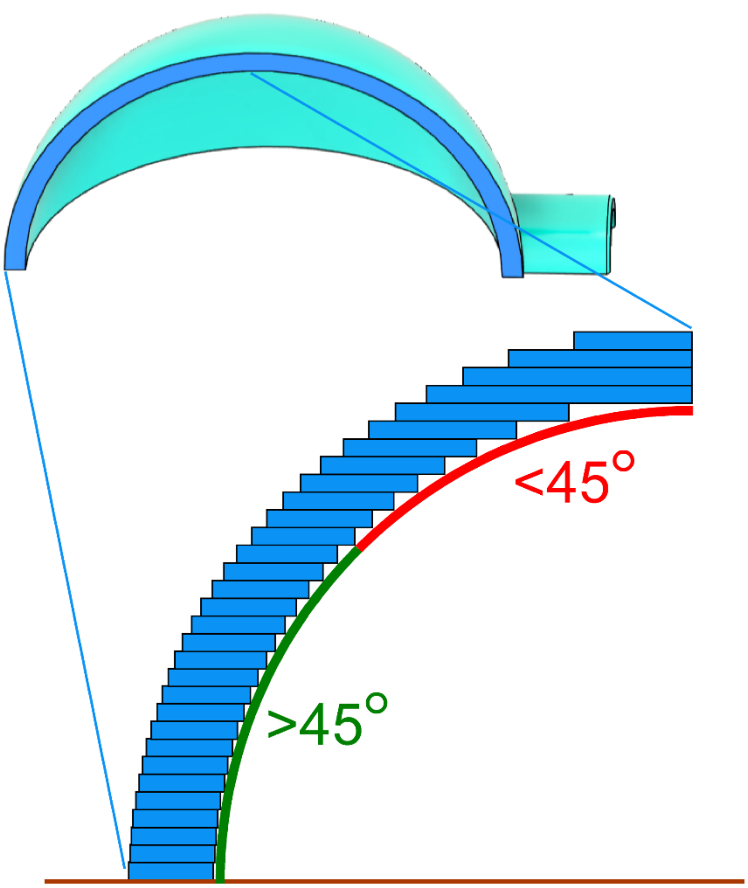




45 Degree Rule

A 3D Printer starts printing the 1st layer on the **build plate** and then prints layer upon layer. An overhang is any printing that is not over a lower layer and thus "overhangs". Overhangs can be printed as long as the surface they form is at an **angle >= 90 degrees**. Note that these illustrations can be considered as magnified view of a small section of the 3D printed object. **Each layer may only be 0.1 mm thick**.





The illustration below shows part of the anemometer cup cross section with exaggerated layer thicknesses.

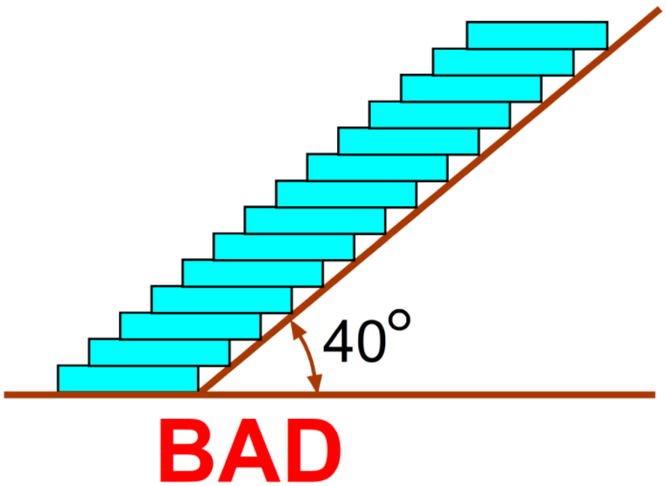
The lower portion of the inner wall has angles that are greater than 45 degrees, which can be 3D printed.

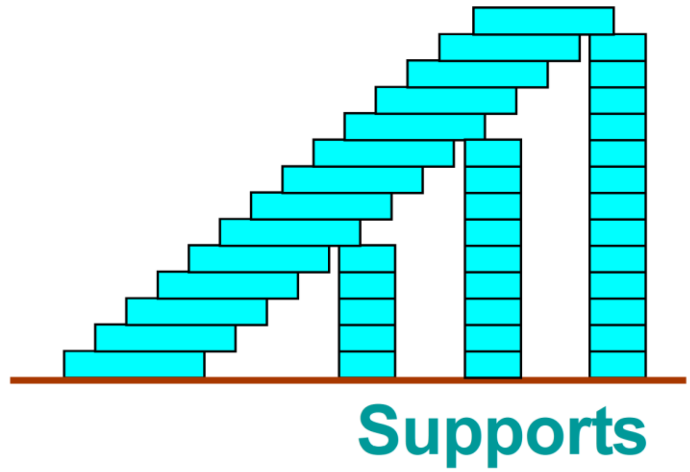
The upper portion, called out with a red line, has angles that are less than 45 degrees. This upper wall section will start flopping down causing the top of the cup to collapse.

3D Printed Supports

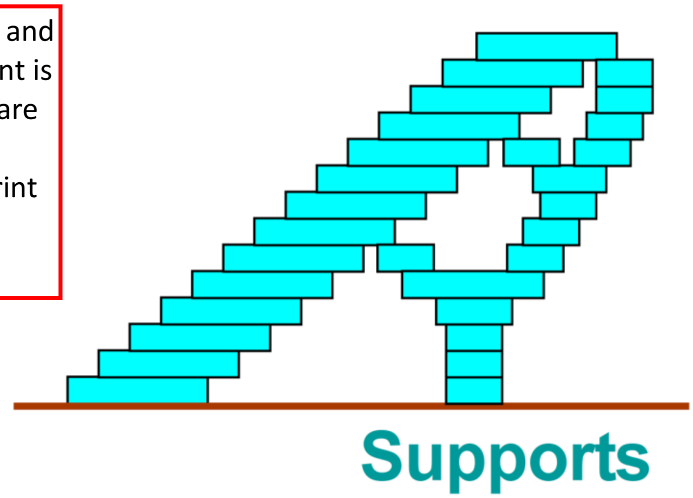
Note that we want a design that **does not need** *supports*. This page serves to explain what supports are.

The **slicer** software, which converts the STL file(s) to the printer format, can **automatically generate supports** to allow the printing of overhangs that violate the 45 degree rule. Slicer software, such as Cura, often has many **settings to configure the support generation**. There can also be a setting for the type of supports. Shown here is an illustration of simple vertical and tree supports.

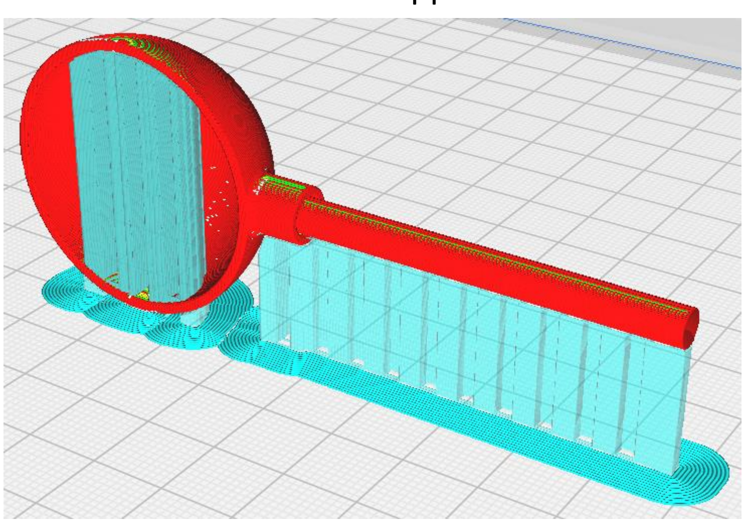




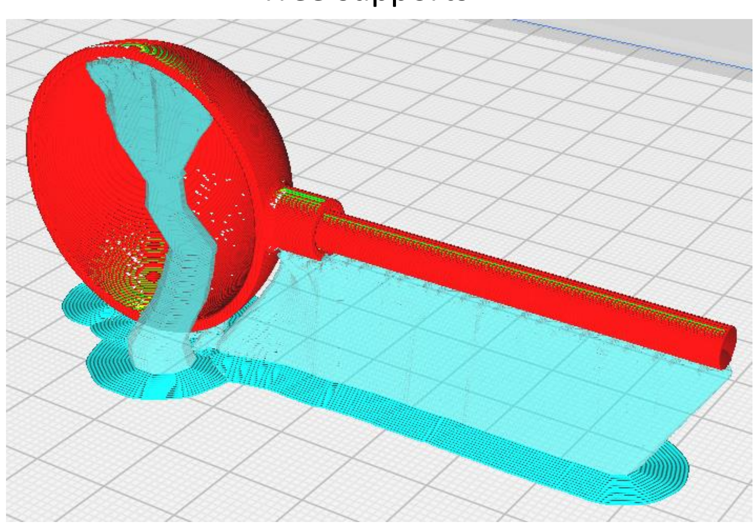
Supports are typically printed with the same material and at the same time as the printed object. When the print is finished and removed from the printer, the supports are then broken or cut away.
Alternatively, a 3D printer with a dual extruder can print the supports with a different material that can be dissolved away with a second process.



Normal supports



Tree supports

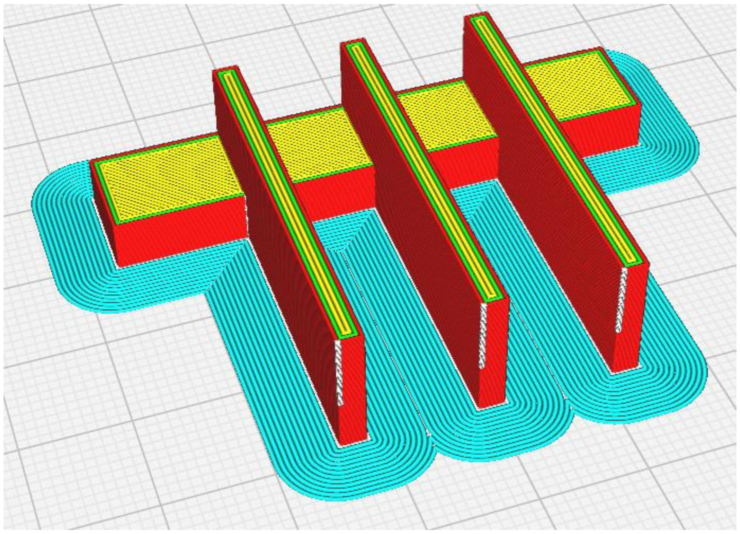


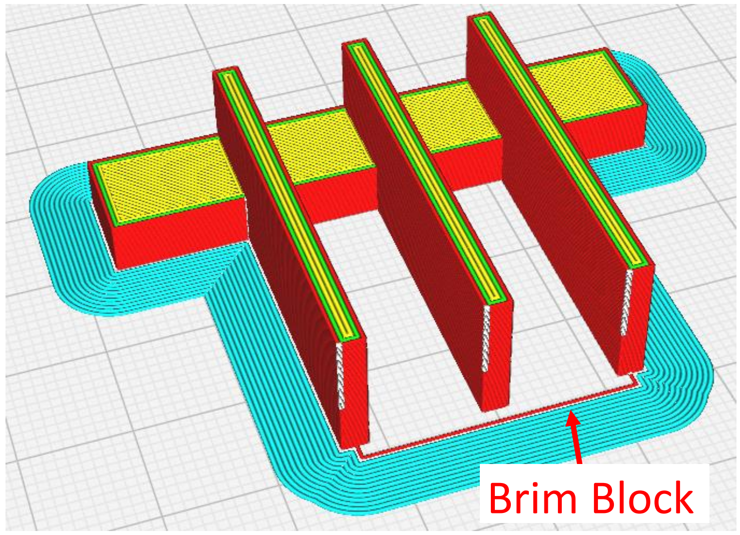
Adding a Brim Block

Slicer programs, such a Ultimaker Cura, have settings to add extra material for the initial layer at the start of a print to enhance Build Plate Adhesion. Especially if the design doesn't have a large bottom surface, the initial layer can detach from the build plate during printing, which results in a mess. This extra material helps prevent this.

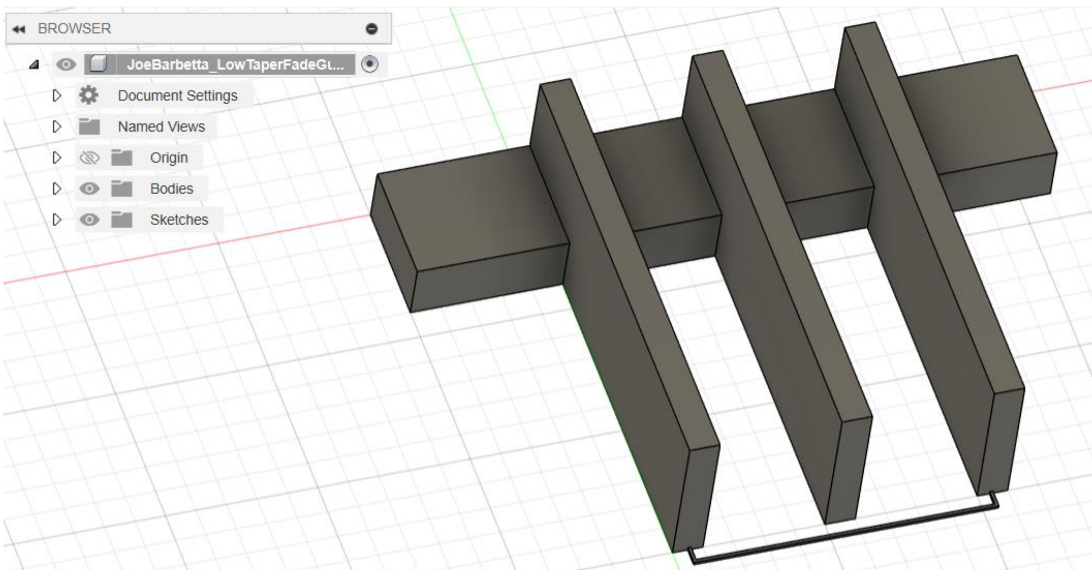
The left screenshot shows the Brim (shown in blue) created by Cura. It is the same plastic used to create the entire print and the final color of the print depends on the filament loaded into the printer. Cura uses different colors for the outer wall (red), inner wall (green), interior (yellow), and brim (blue). When the print is complete and removed from the build plate, the brim material is peeled off of the part. Sometimes small sections may be left on the print, which can later be *deburred*.

Especially, when elements are close together, it can be difficult and slow to remove the brim between adjacent elements. One can add a small strip of plastic in the design to act as a Brim Block, to limit the use of the brim. The right screen shot demonstrates this and one can see that this prevents brim material from being added between the elements.

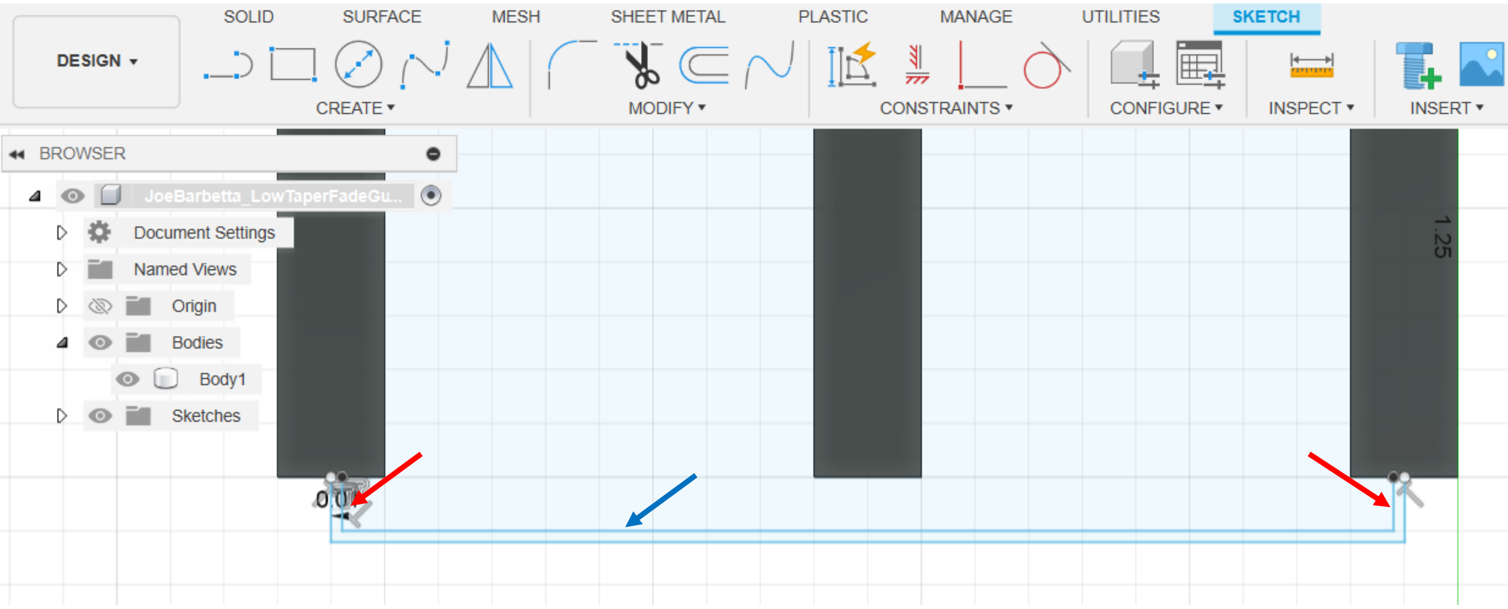




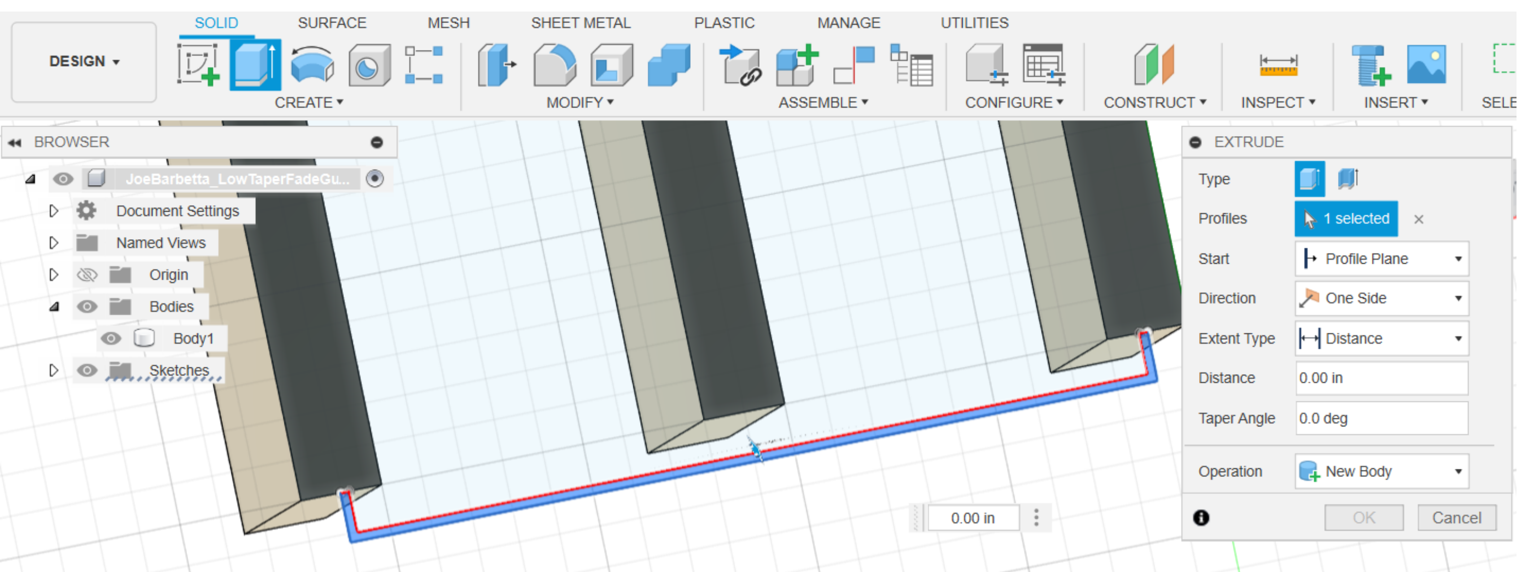
This screen shot shows the Brim Block that was added in Fusion. The next page discusses how it was created.



The Brim Block can be created by creating a sketch on the **bottom surface** of the part. In this case **two short (0.05" long) lines** (indicated with red arrows) were drawn down from the edges. Then a line (blue arrow) was created between the two endpoints. Then the Offset tool (in the MODIFY menu) was used to create parallel lines with a distance of **0.01"**. Note that these values are not critical.

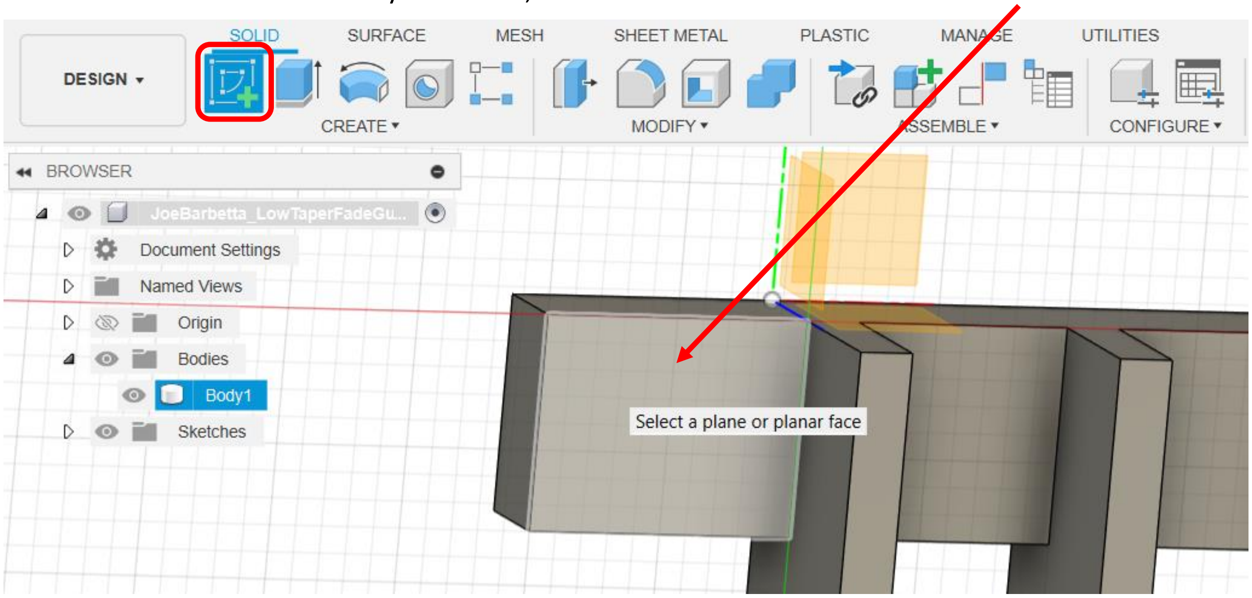


After clicking **Finish Sketch**, the view can be changed and the **Extrude** tool can be used to extrude the profile by **-0.01** (note the minus sign). A negative value was used because the sketch was created on the underside of the part. Note that is value is not critical. It is so thin, it will break off with the brim.



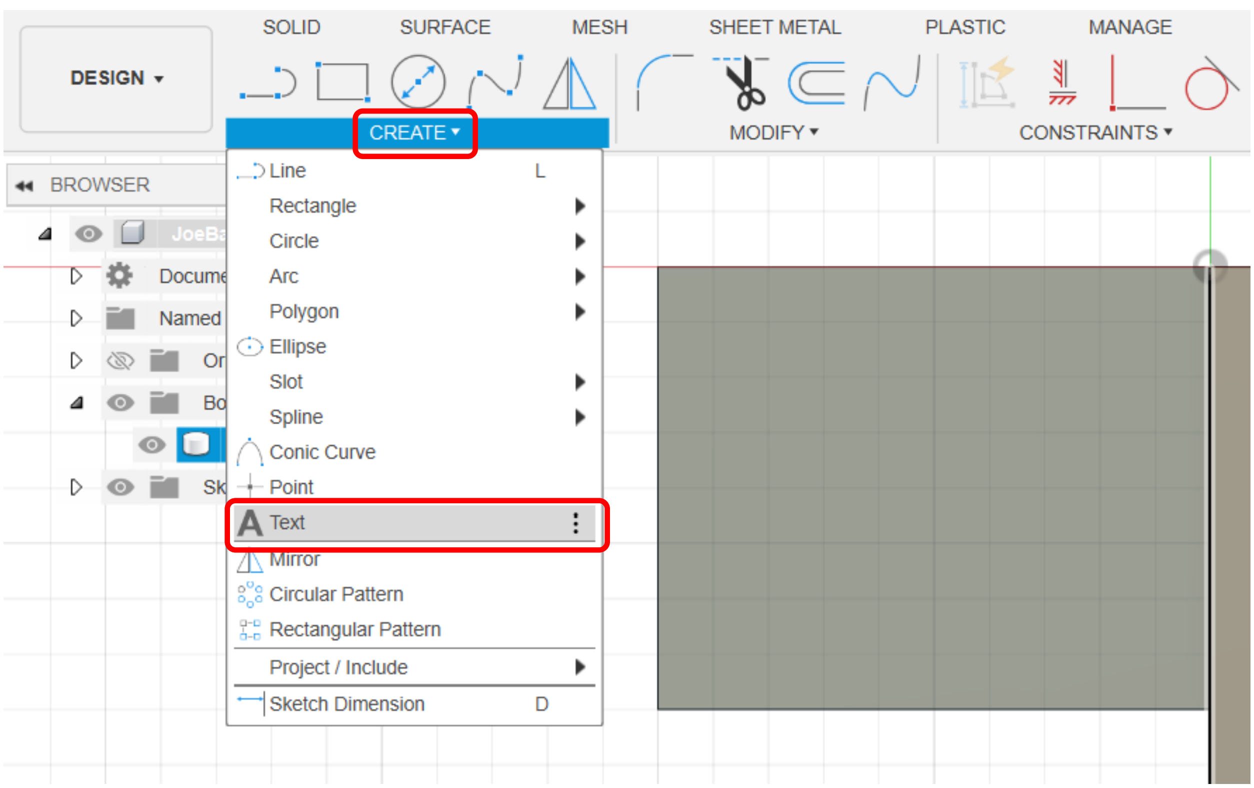
Adding Text

- decide on a flat surface to add your initials, select **Create Sketch** and click on that **surface**

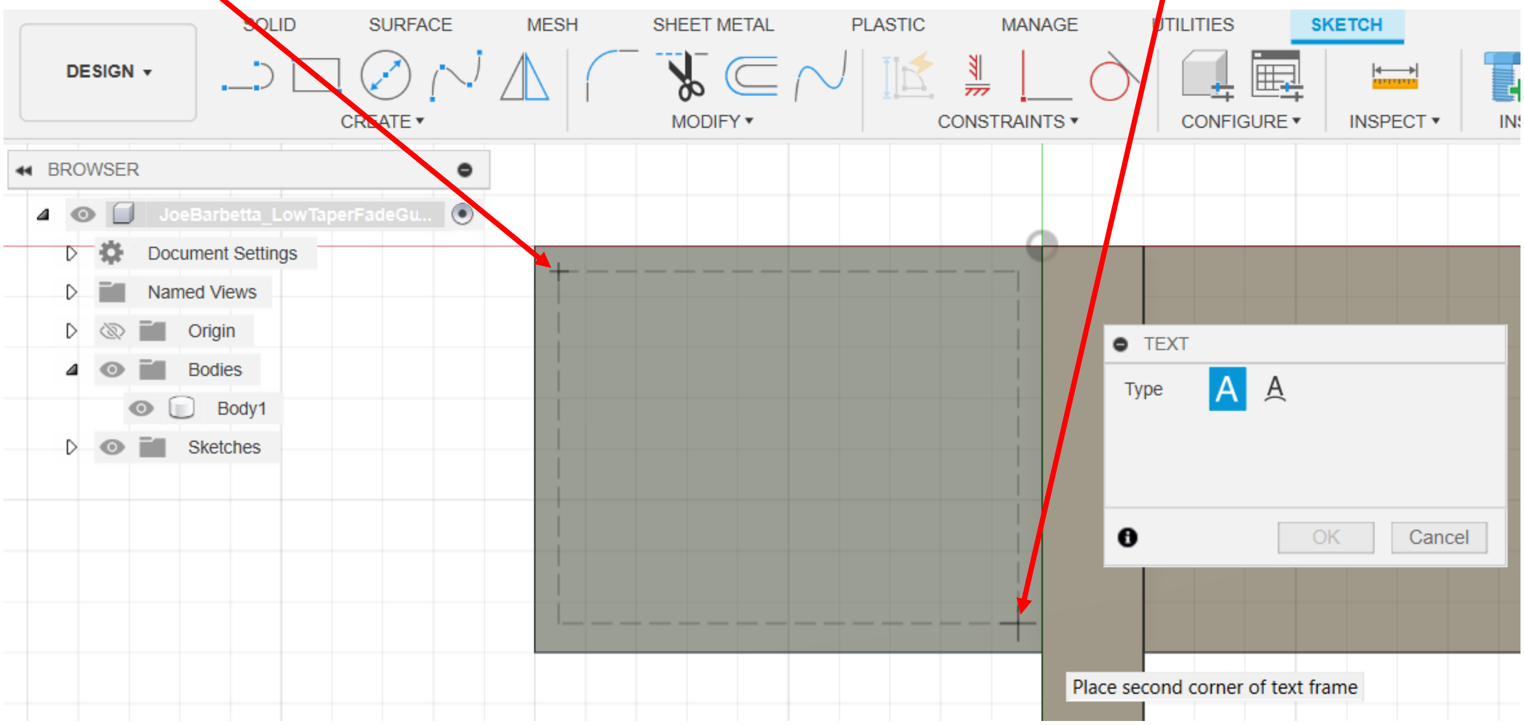


As stated previously, these steps show a simple sample object. Your product will look nothing like this.

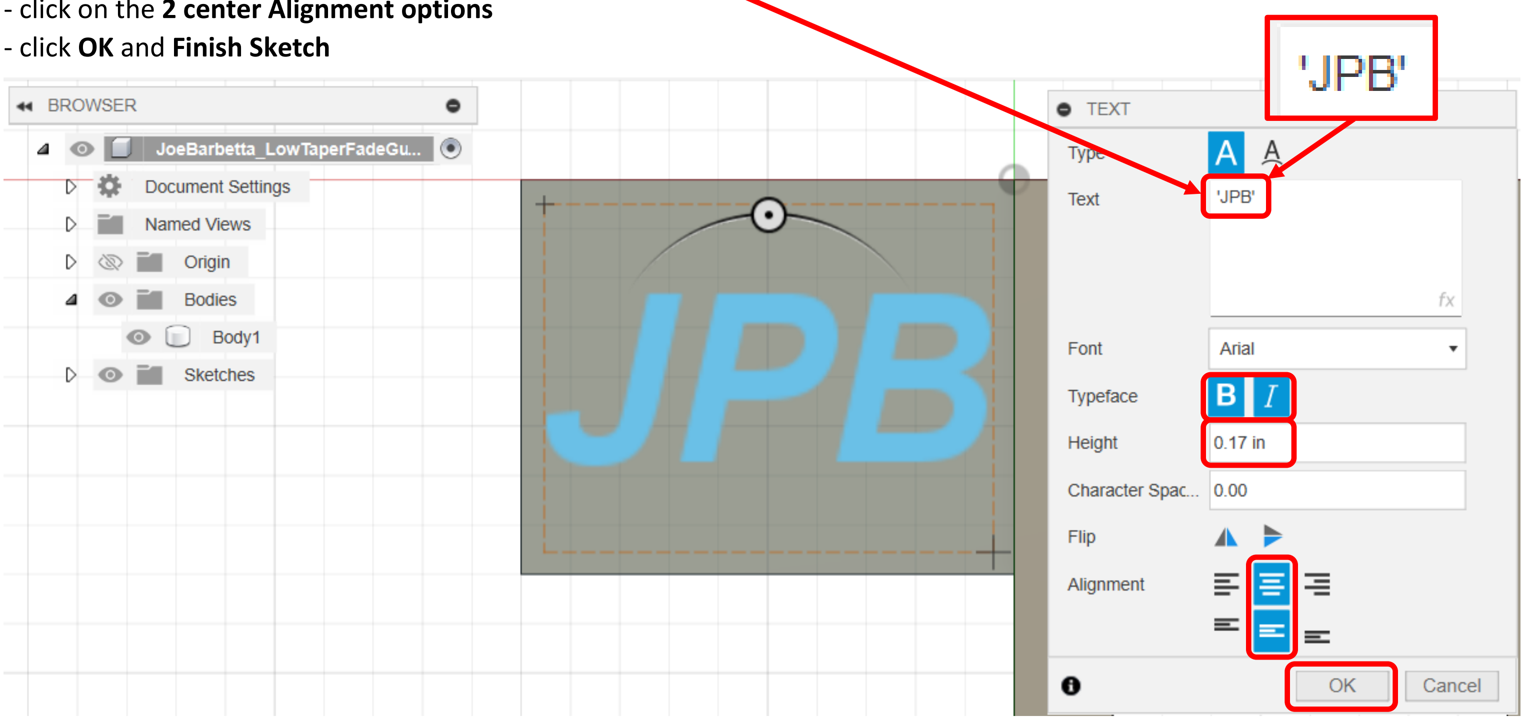
- zoom into the area where the text will be added
- from the **CREATE** menu select **Text**. If a window pops up about Parametric Text, click its **OK** button.



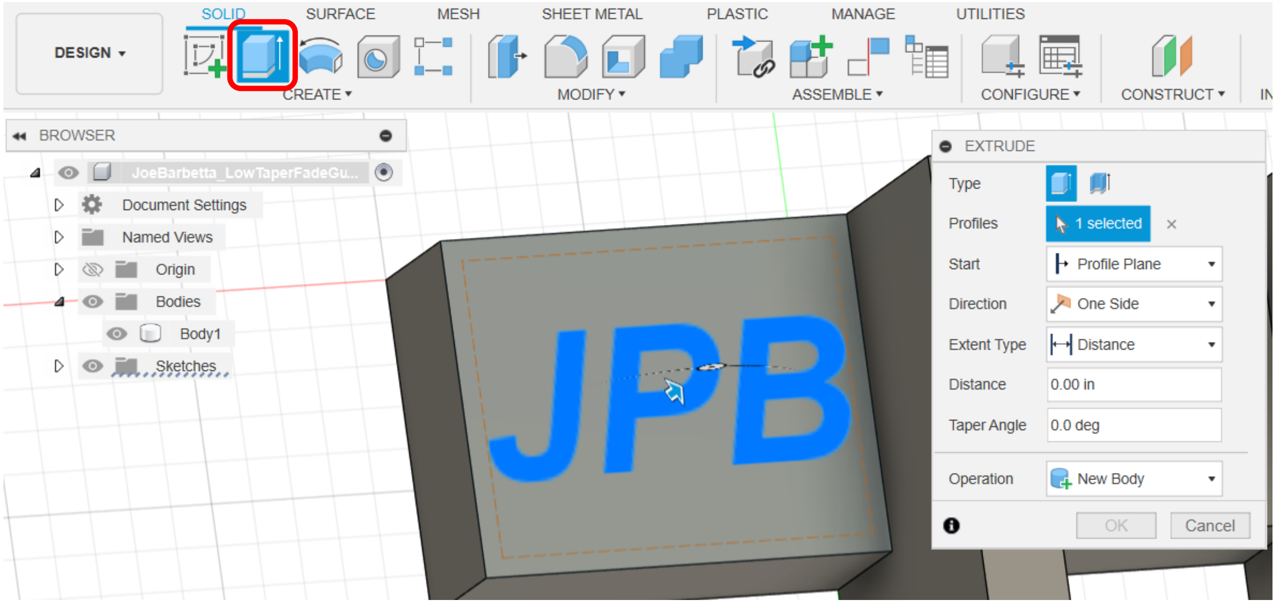
- zoom into the area where the text will be added
- from the **CREATE** menu select **Text**. If a window pops up about Parametric Text, click its **OK** button.
- click at a point to start a rectangle to define the text region and drag the other corner to a 2nd point and click. These locations are not critical. One wants to create a region for the text.



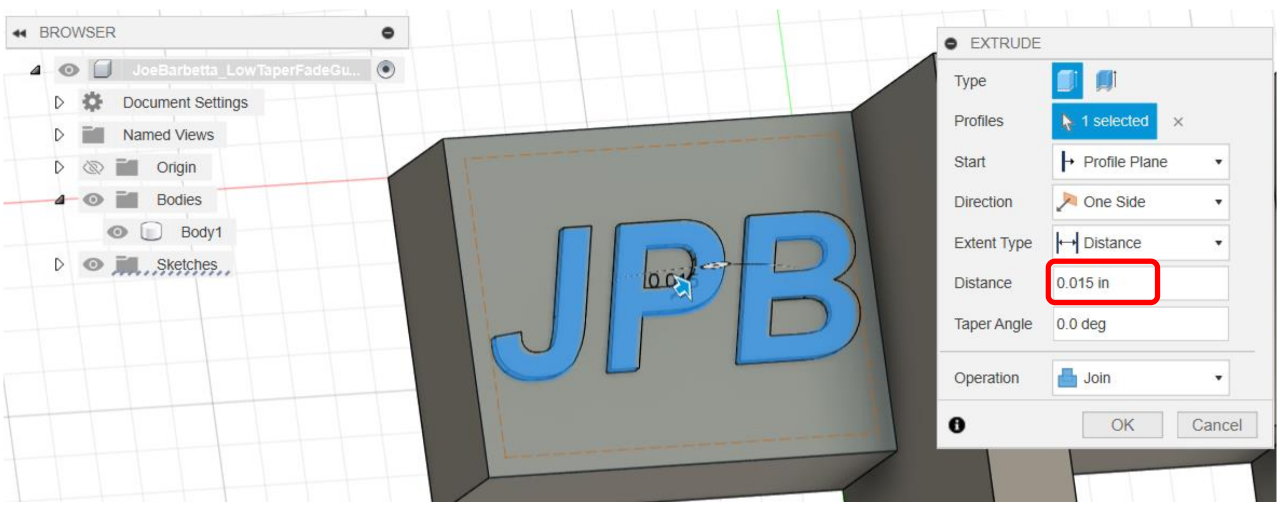
- in the Text box enter your **3 initials preceded by and followed by a single quote**
- change the **Height** value to allow your initials to fit on the surface and optionally click the Bold and Italic icons
- click on the **2 center Alignment options**
- click **OK** and **Finish Sketch**



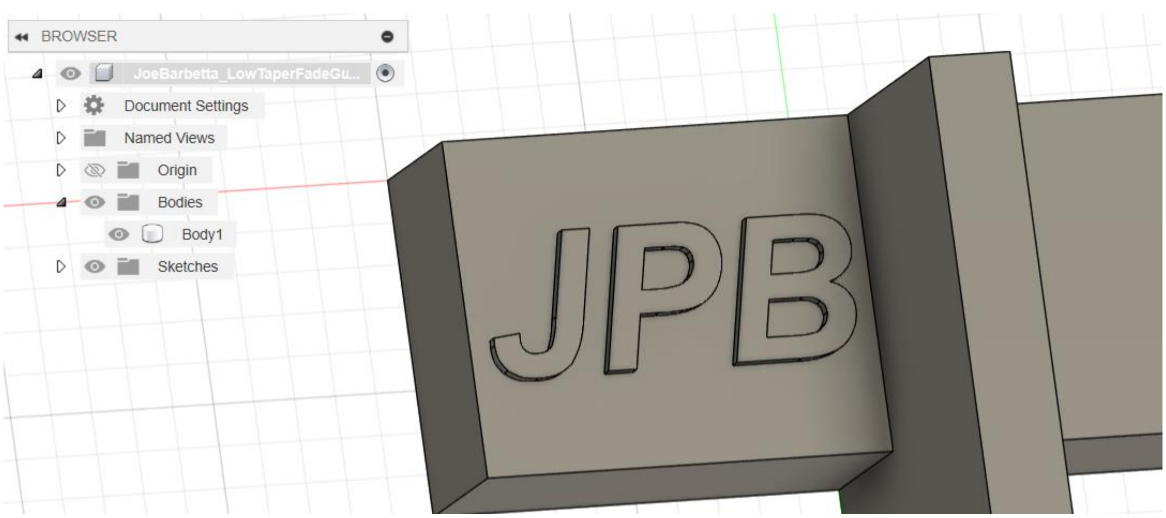
- zoom in to the text and click on it to select the text
- select the **Extrude** tool



- for **Distance** enter **0.015** and click **OK**. This will extrude the text out. If it is desired to cut the text, enter **-0.015** (note the minus sign)



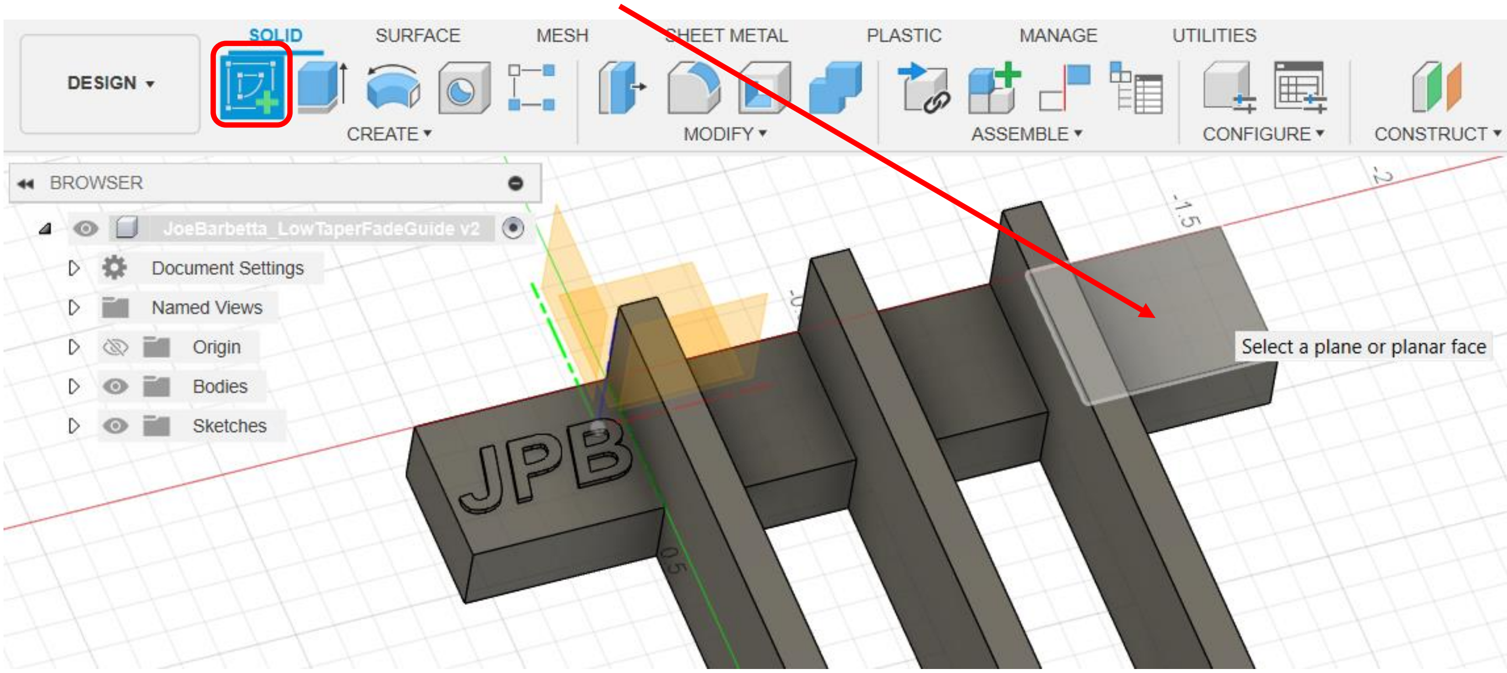
This is the result of text extruded outward.



Adding a Hole

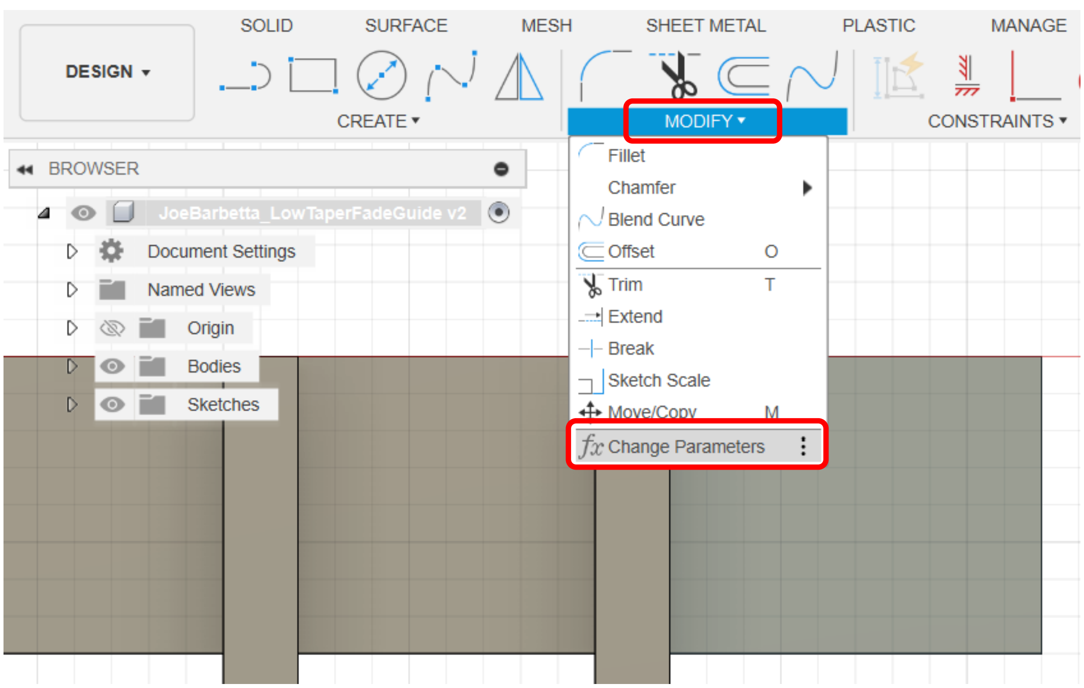
We want to add a hole someplace to allow the Low Taper Fade guide to be hung on a hook.

- select the **Create Sketch** tool and click on the **surface** where the hole should be



One could just draw a circle for the hole, but it can be useful to create a Parameter, which acts as a variable that can be used to define the diameter of the hole. A Parameter is most useful when a design will have multiple holes of the same diameter. Even though we may only have one hole in the design, it will be good practice to use it.

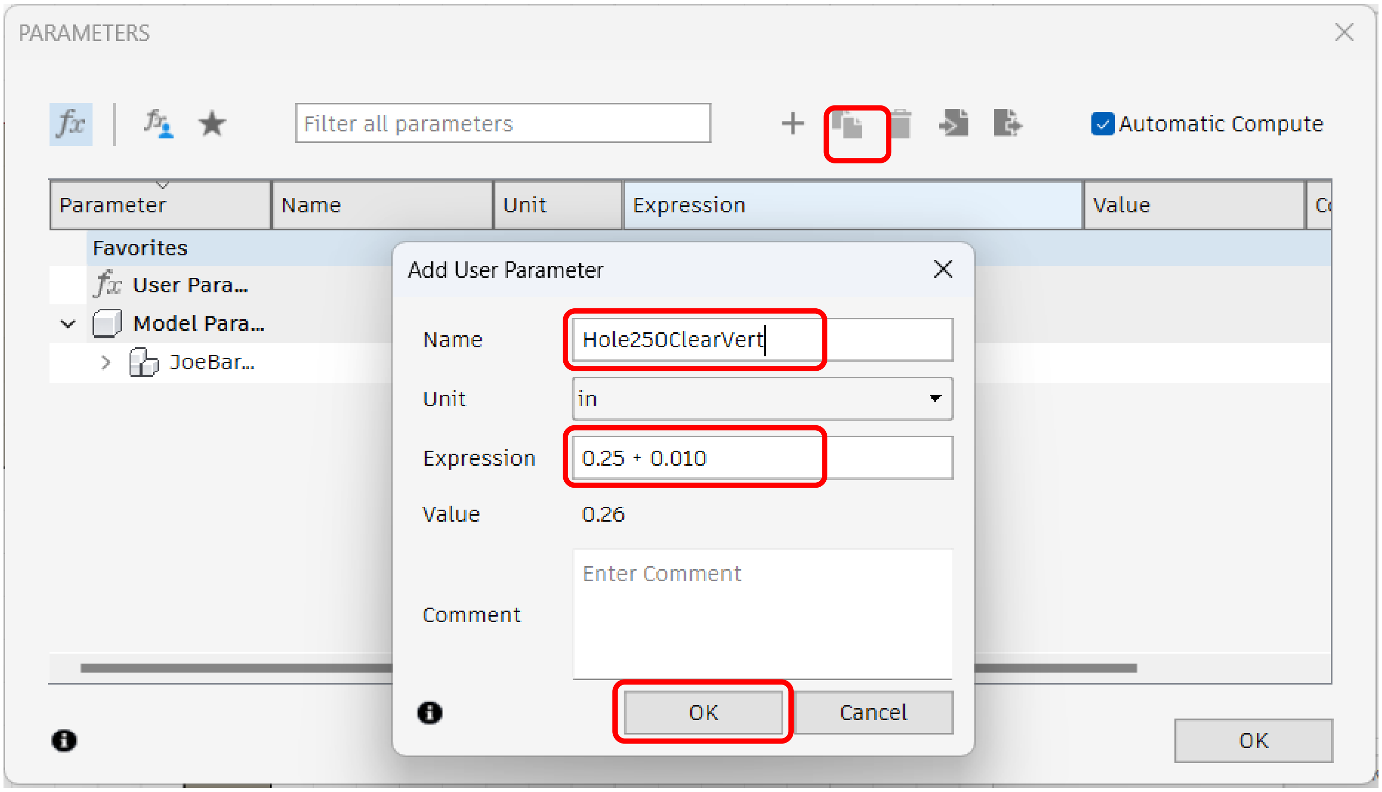
- from the **MODIFY** menu select **Change Parameters**. If a message window about **Parametric Text** appears, click its **OK** button.



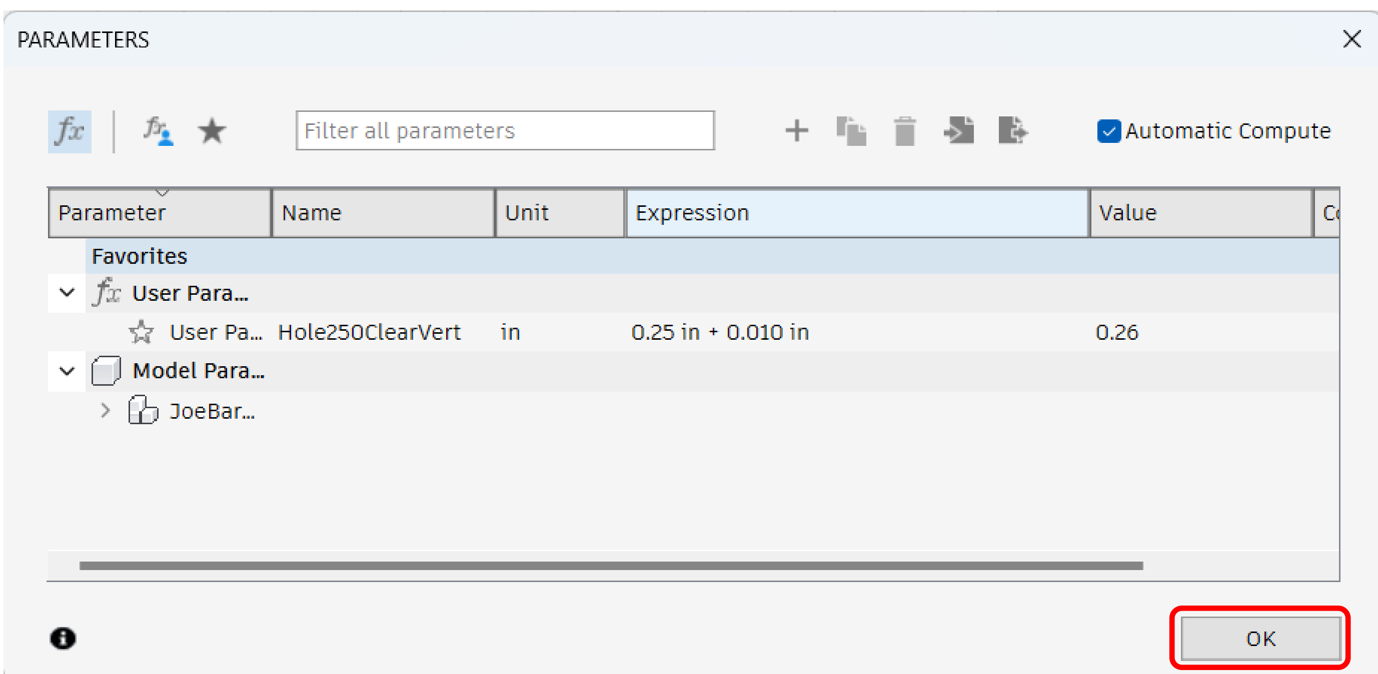
- click on the **+** icon and for **Name** enter **Hole250ClearVert** and for the **Expression** enter **0.25 + 0.010** and click **OK**.

Why that crazy name? It's just a convention that Mr. Barbetta made up for various designs. Hole designates that the Parameter is for a hole. **250** is for a 1/4" hole. The **Name can only have letters, numbers, and underscores**. A decimal point or forward slash cannot be used and 250 translates to 250 thousandths of an inch. **Clear** indicates it is a **clear hole as opposed to a pilot or tap hole**, which would be threaded. **Vert** indicates it is a **vertical hole with regard to the build plate**.

Why is there an **Expression** box instead of just a **Value** box? The Expression box can hold a formula based on other Parameters using mathematical operations. The use of Parameters for hole diameters allows one to adjust the diameter to account.

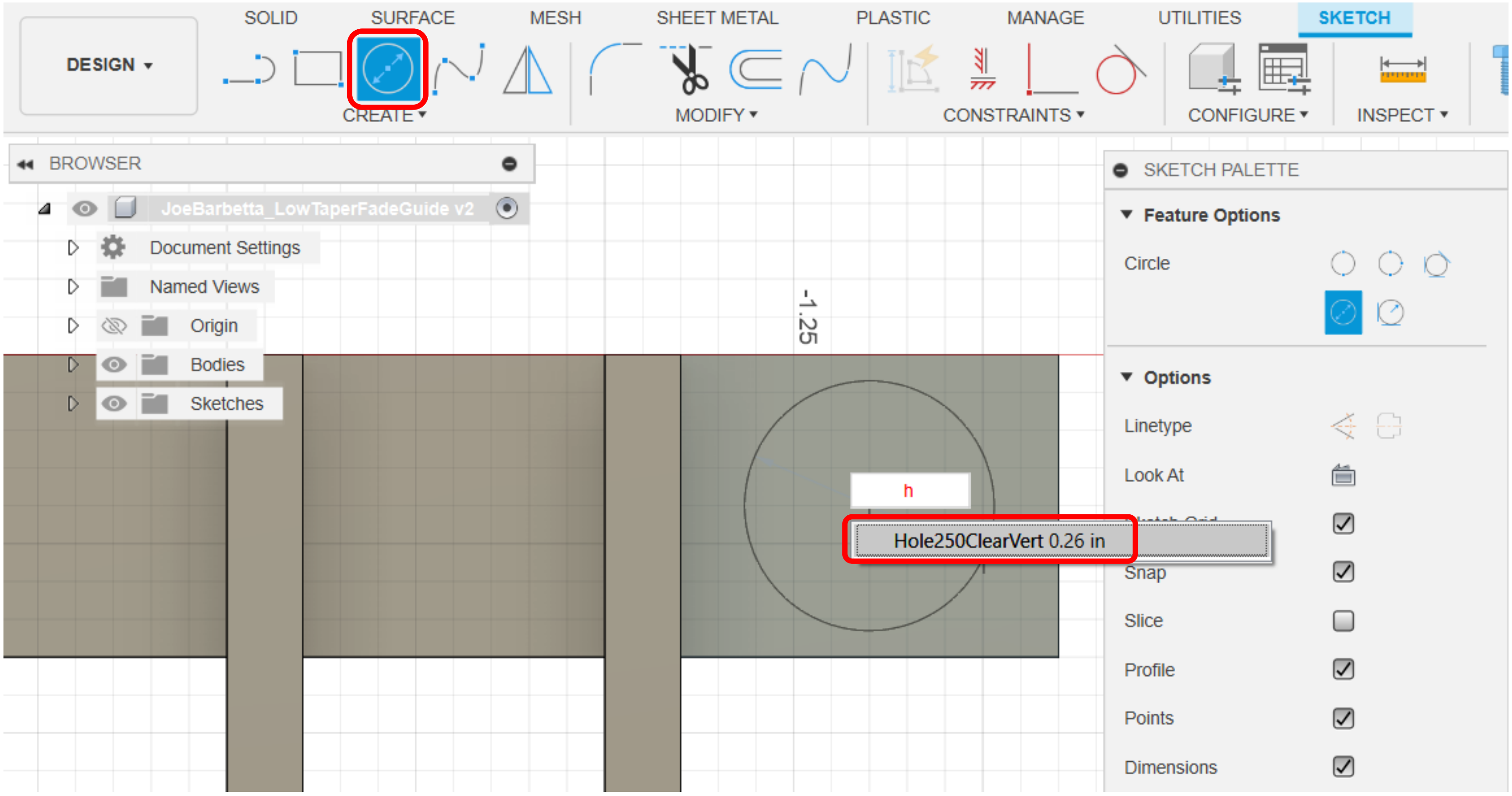


- click **OK**

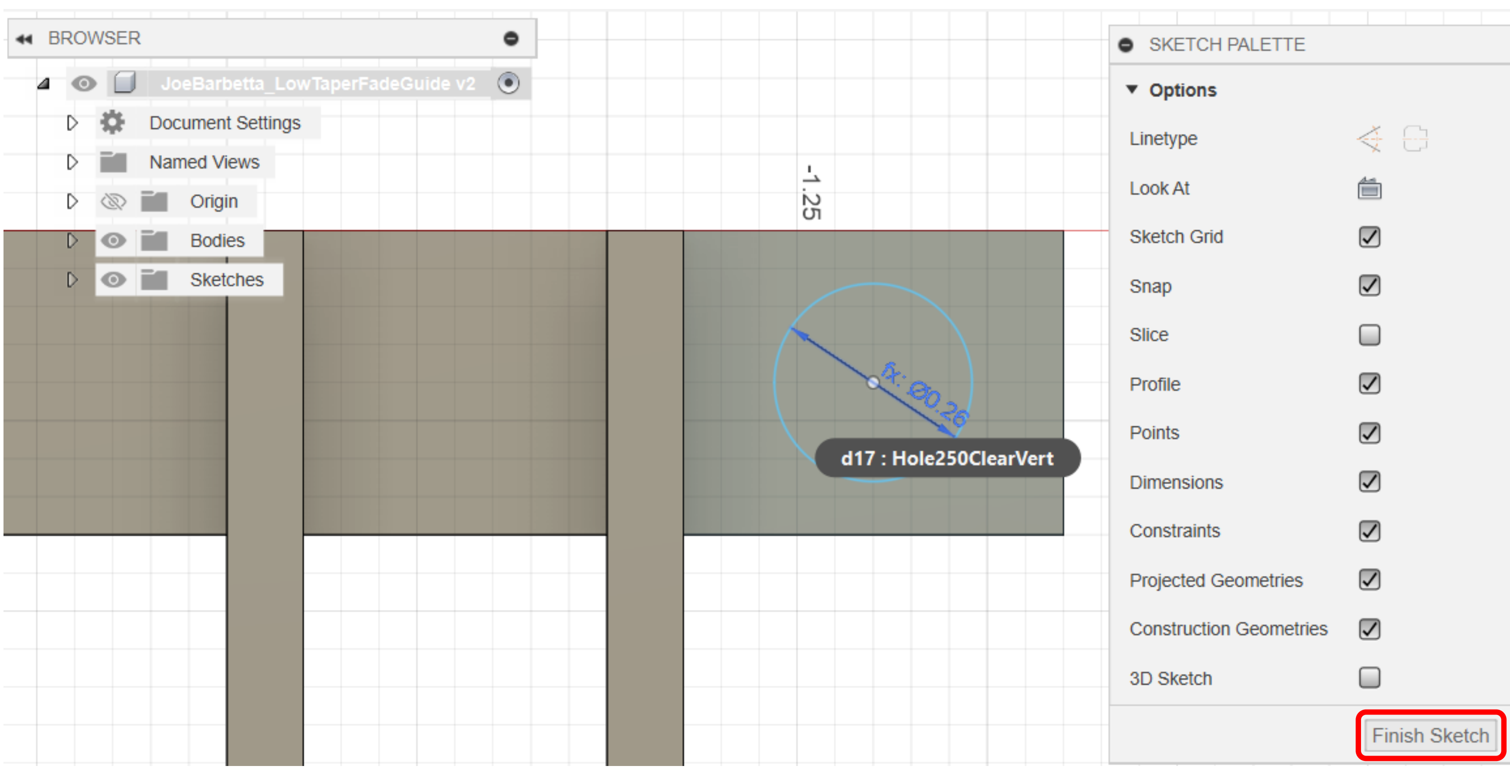


Note that Fusion does have a hole tool, however, to learn methods that will work with other CAD programs, we will sketch a circle and extrude it. This same method would also be applicable to creating a square or hexagonal hole as well.

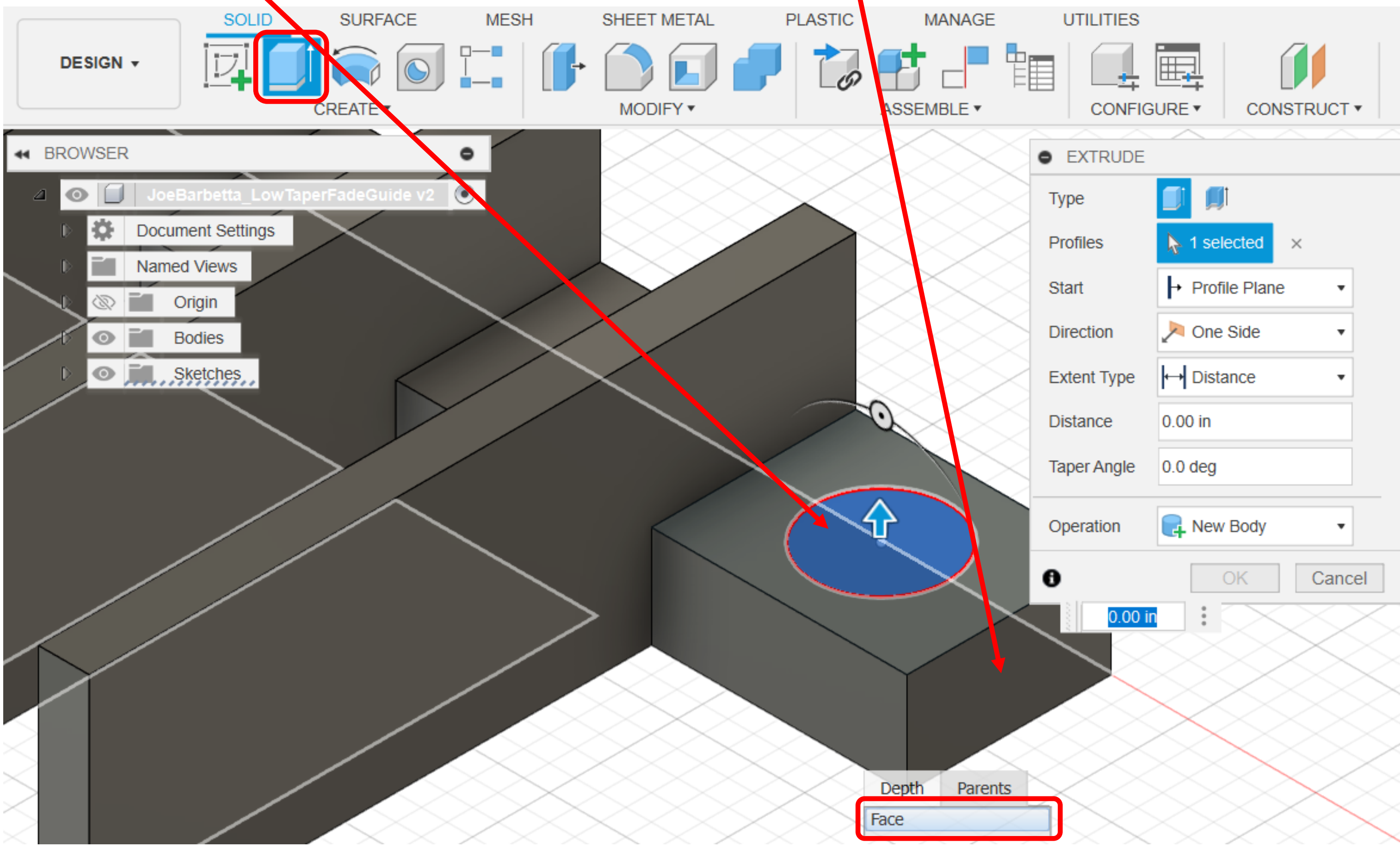
- select the **Center Diameter Circle** tool. If its icon is not visible, it can be found in the **CREATE** menu.
- click on a **point for the center** of the circle. For a hole to hang on a hook the hole placement is not critical, but try to get it centered by eye.
- **extend the circle edge out**, type **h**, select **Hole250ClearVert** from the menu, and press the **Enter key**.



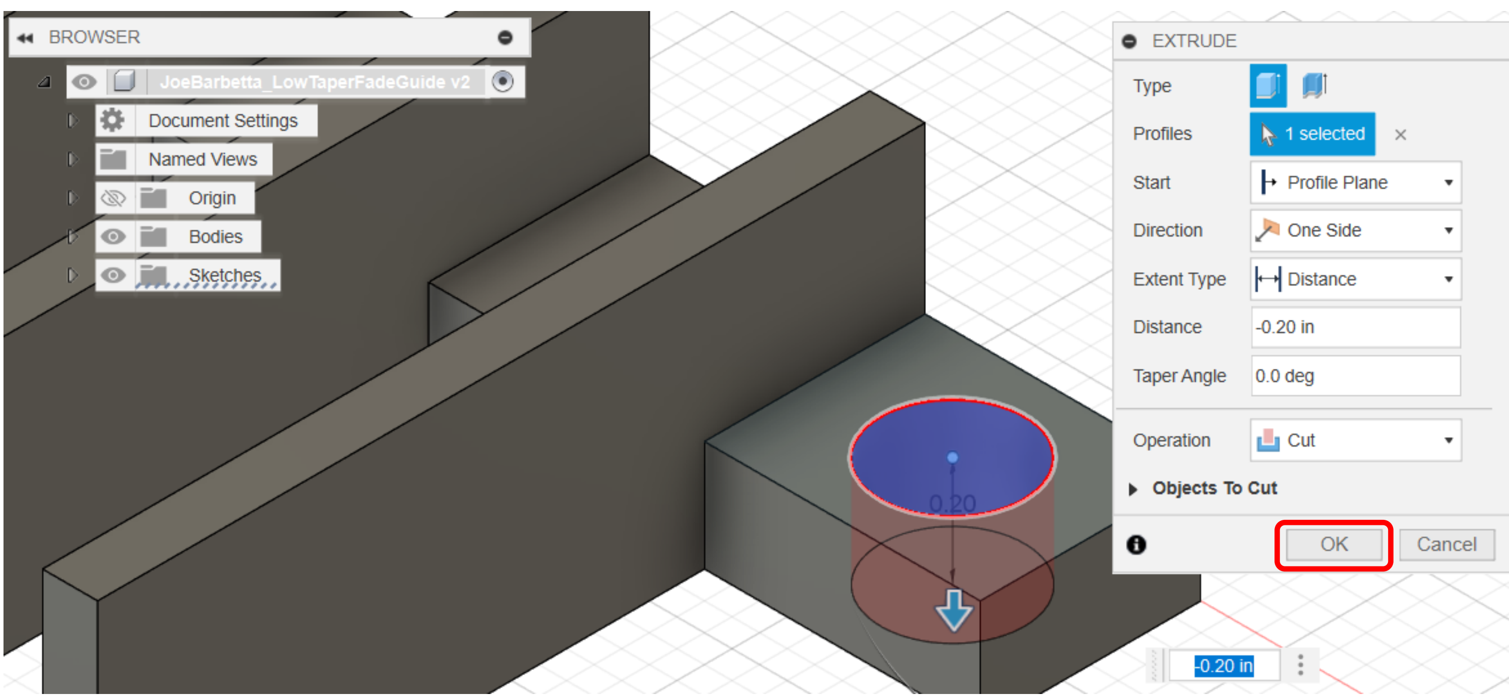
- click **Finish Sketch**



- select the **Extrude** tool
- click on the **interior** of the circle and then **hold the mouse down on the surface** indicated, which should cause the text Face to show



- select **Face** and it will show the circle cutting through the material. An option can be to pull the **blue arrow** until it shows the circle cutting all the way through the part.
- click **OK**

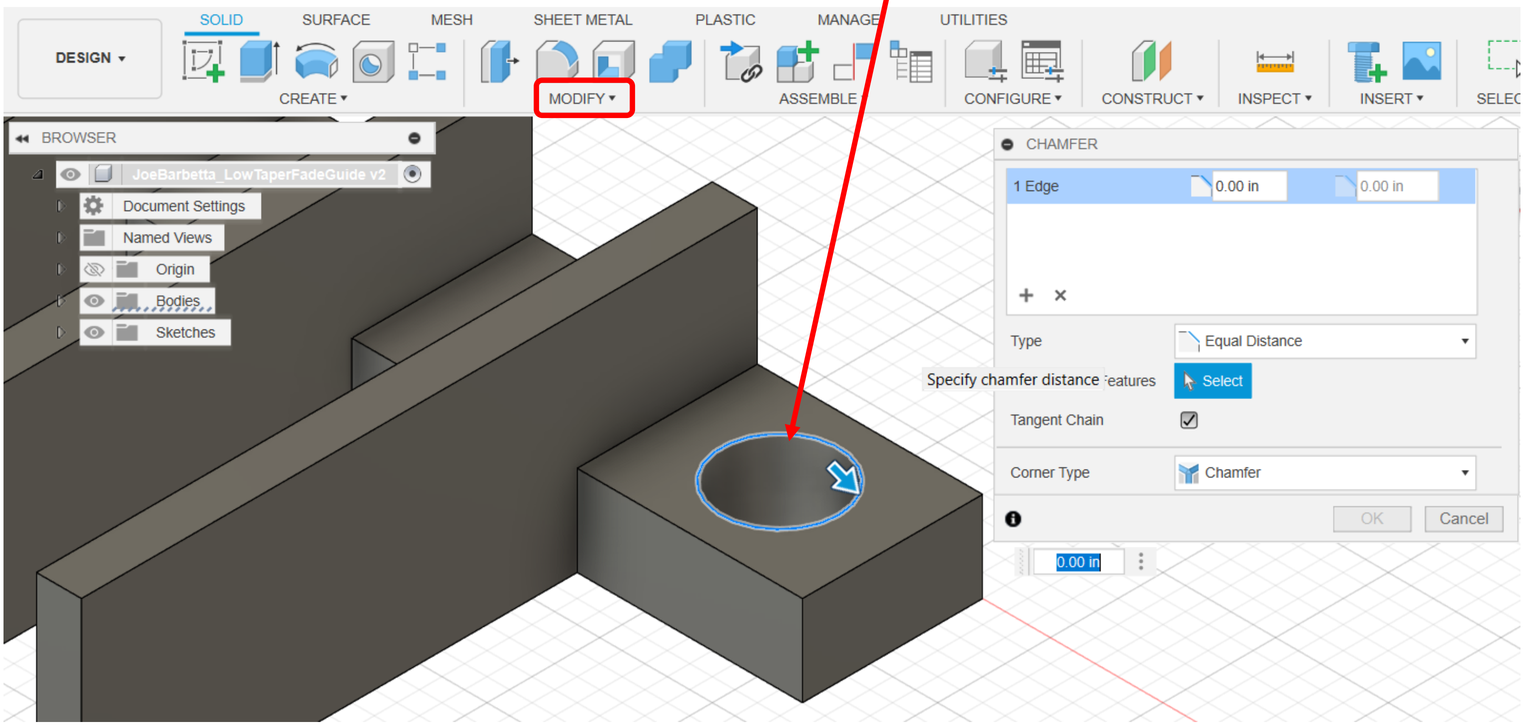


Adding Hole Chamfers

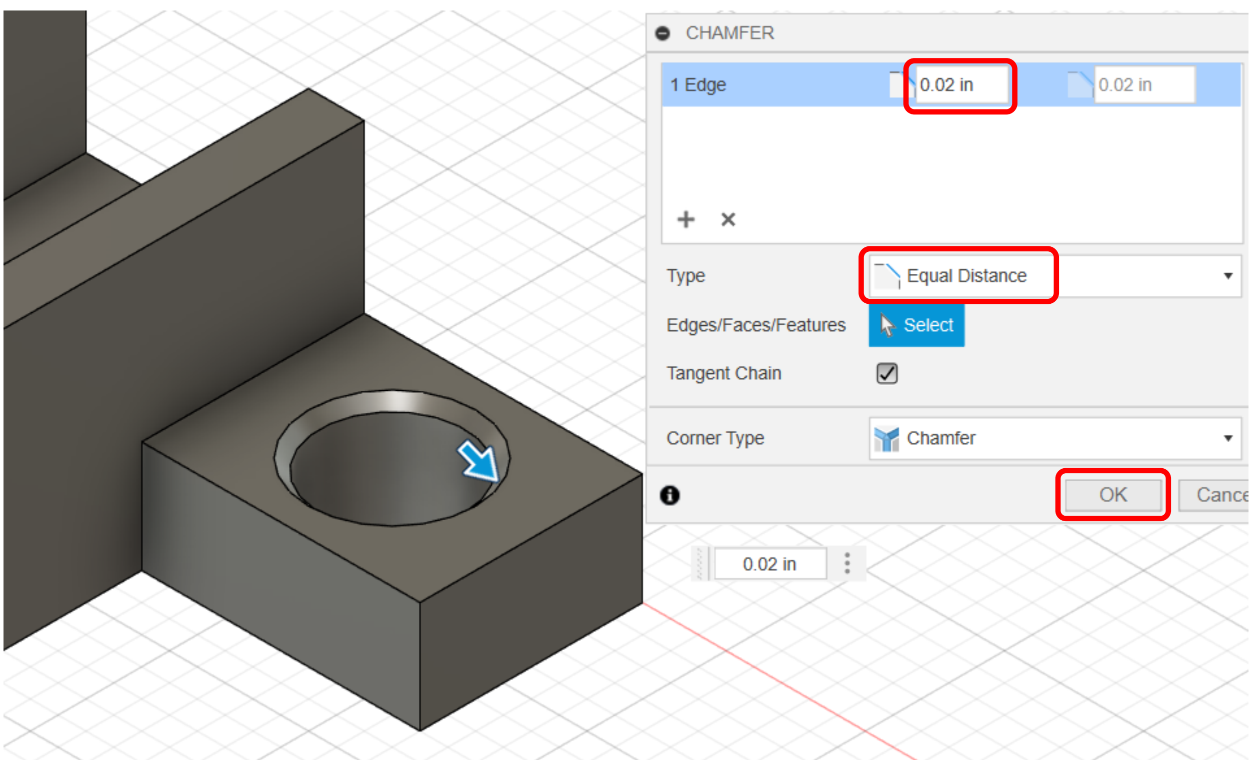
The edges of the hole were chamfered to allow a hook to more easily pass through the hole.

A fillet could be used instead on the top surface, but it's best to use a chamfer on the bottom surface, which will be on the build plate to adhere to the 45 degree rule.

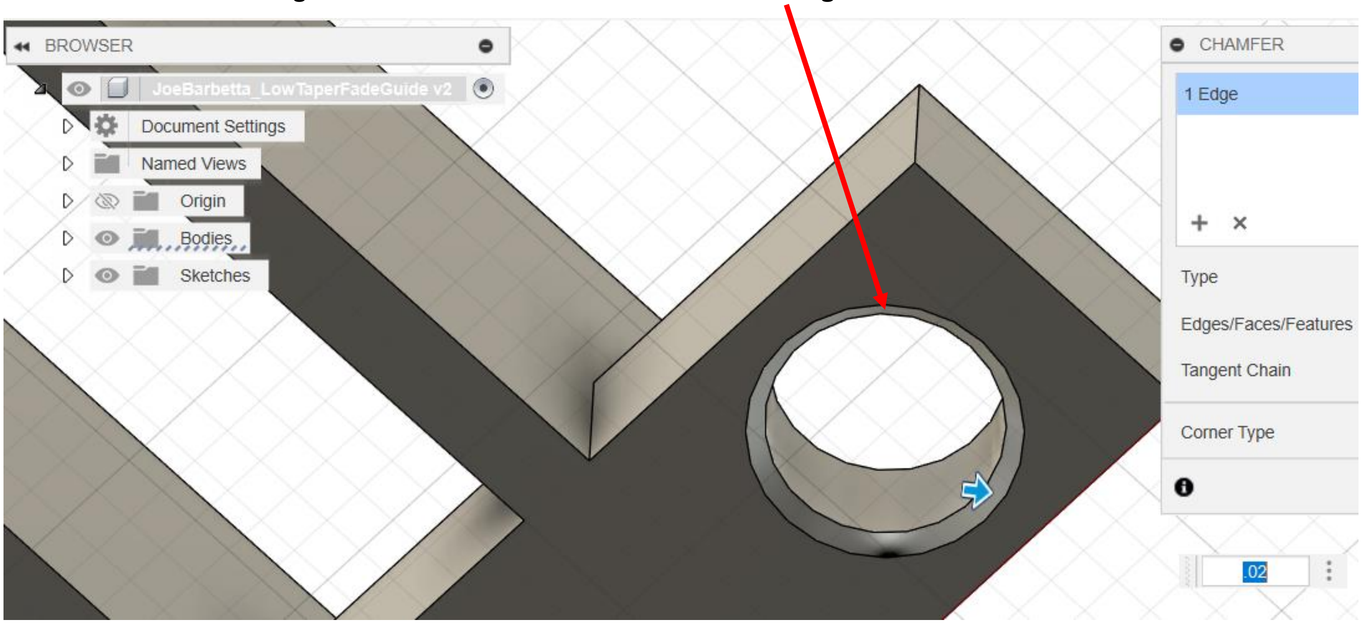
- from the **MODIFY** menu select the **Chamfer** tool and click on the **edge of the hole**



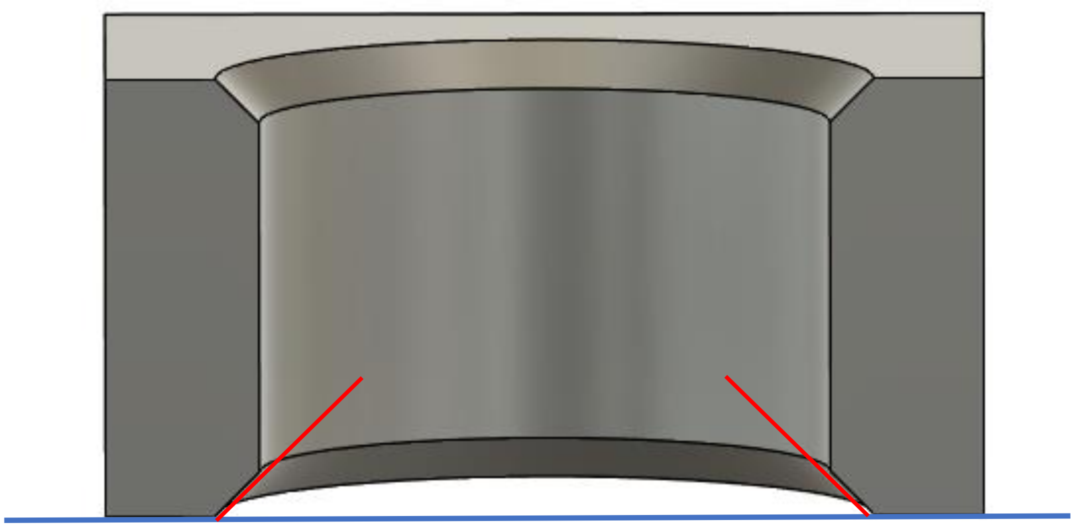
- ensure that **Type** is set to **Equal Distance**
- enter **0.02** and click **OK**

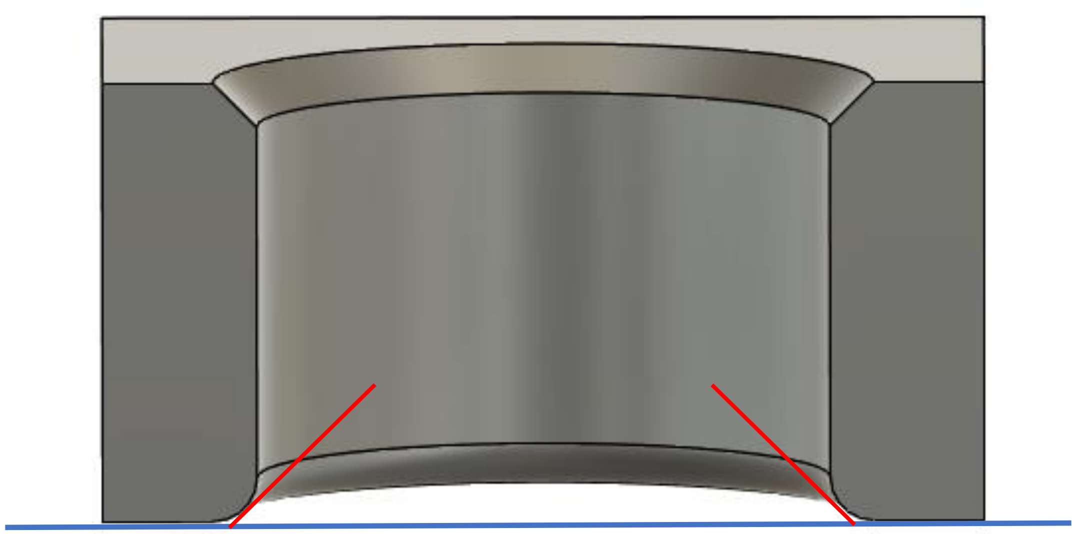


- use the **View Cube** to rotate the view to access the other side of the hole
- use the **Chamfer** tool again to create a **0.02** chamfer of this **hole edge** as well



Below are illustrations of the hole cross-section with **chamfers** and **fillets (lower picture)**. The lower part of the fillets have a **angle less than 45 degrees** close to the build plate (blue line) and thus **violate the 45 degree rule**.



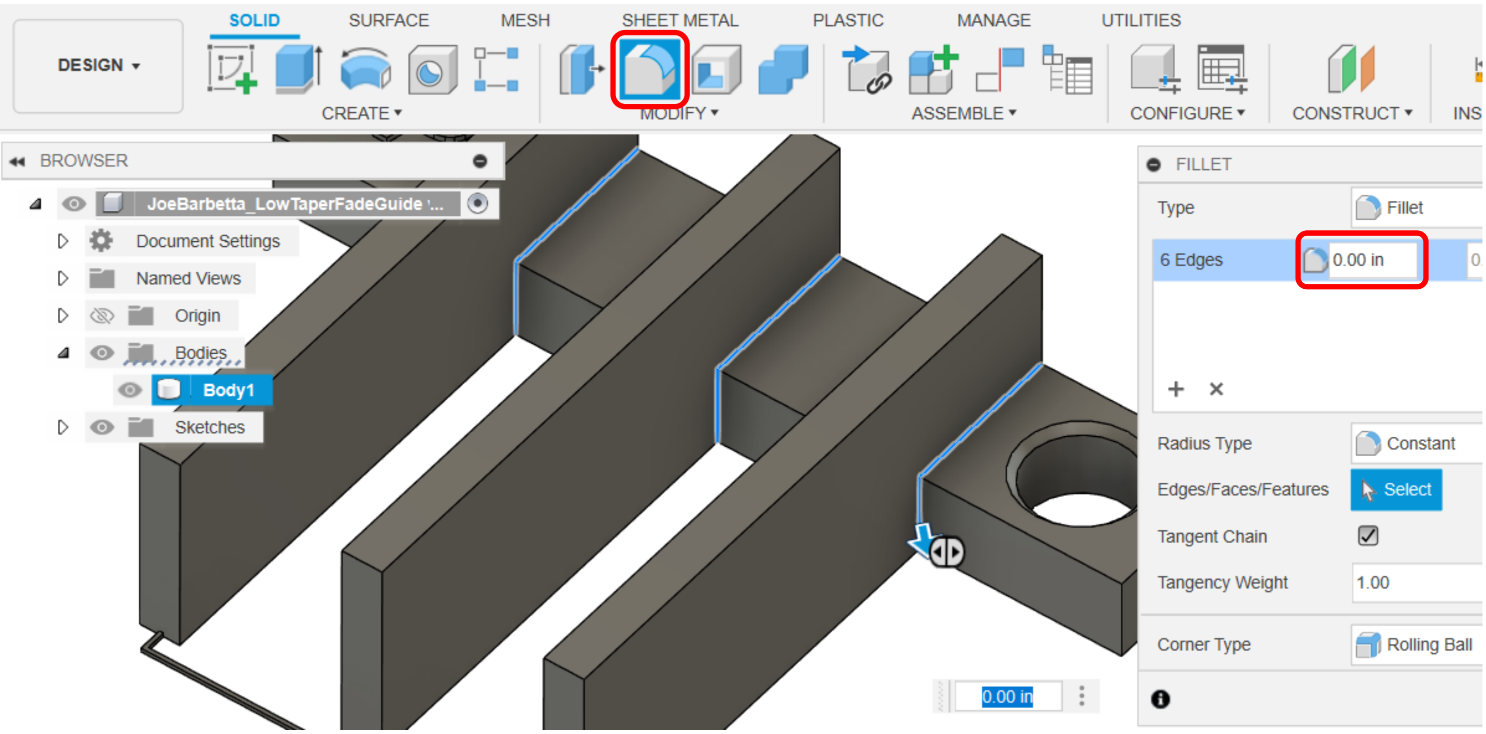


Adding Fillets

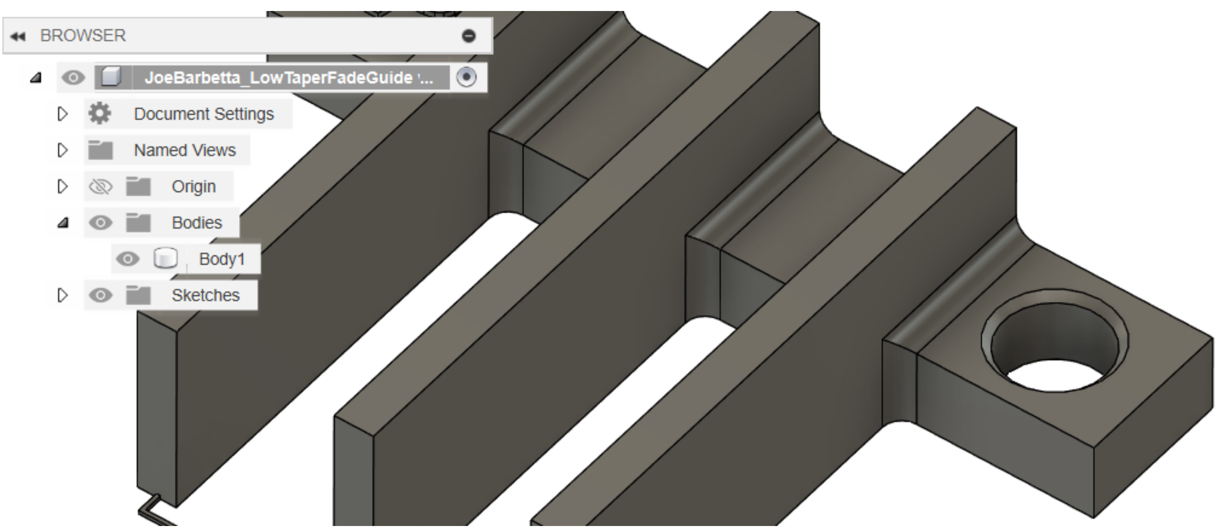
Adding fillets to a design add to the aesthetics of a design, but also strengthen the part and allow for better printing (a DFM consideration). Sharp internal corners result in stress concentrations upon bending of the parts. Adding fillets help to prevent cracking at those points

- select the **Fillet** tool and click on any edges of internal corners and use a value of **0.05**. Note that the view will likely have to be rotated to access internal corners that are hidden in one view.

This value is arbitrary and a value was chosen that doesn't interfere with features of the part or add much extra plastic.

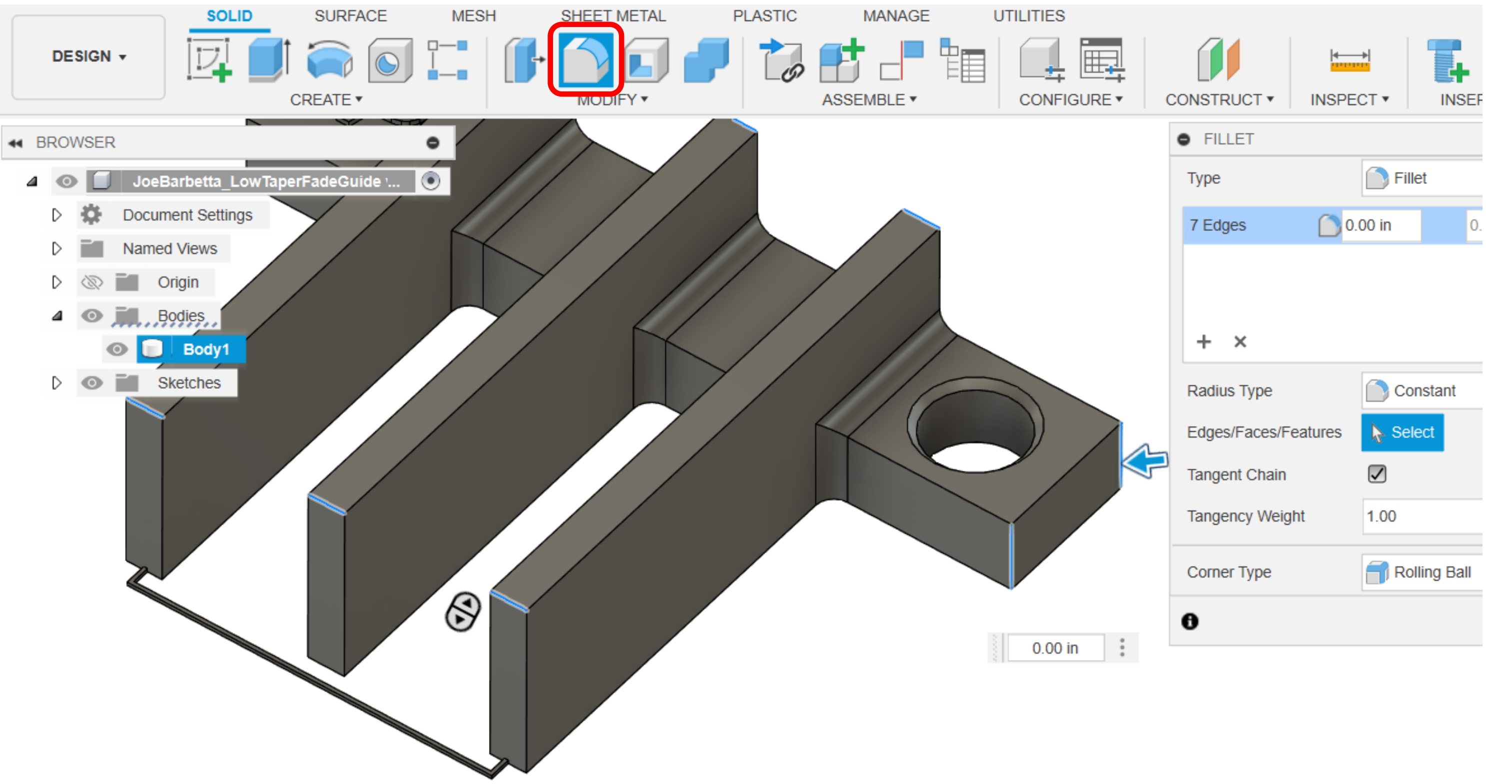


This is the result of the fillets.

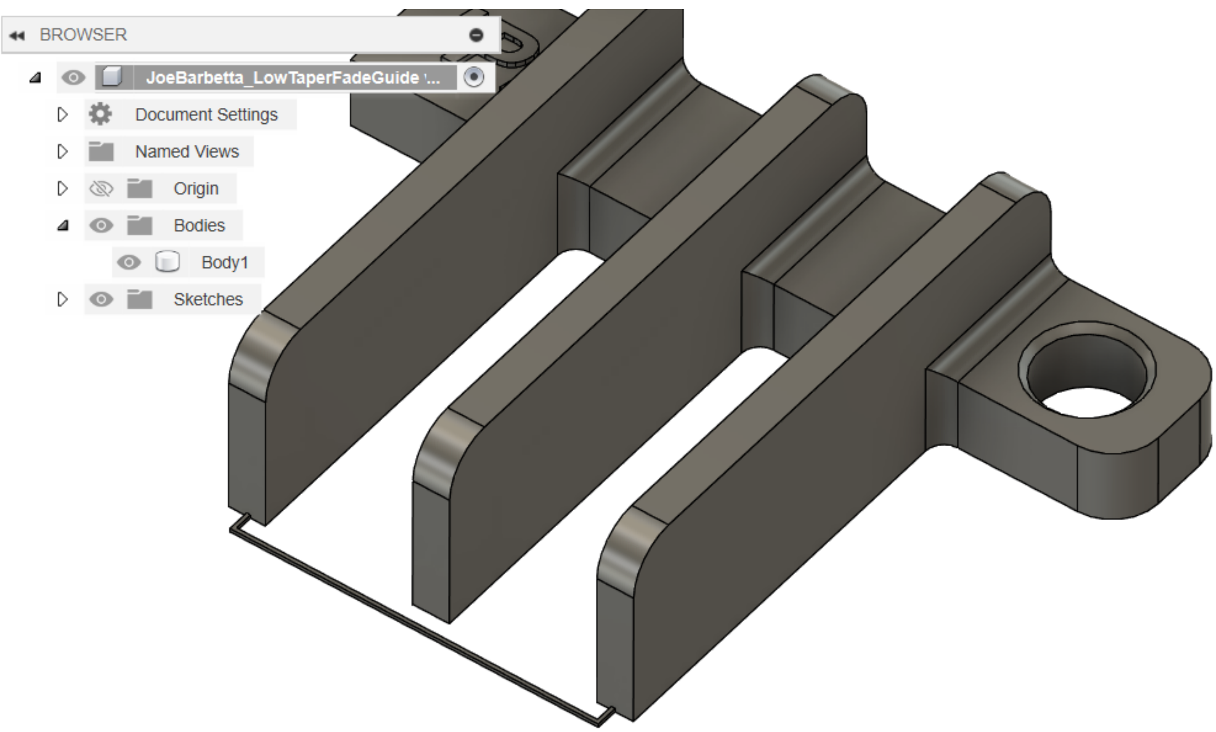


Adding fillets to external corners add to the aesthetics and prevent scratches from sharp edges. Adding fillets to corners for vertical edges at the build plate help prevent corners from curling up during printing.

- select edges, such as shown, where there is room for a large fillet, such as **0.125**



This is the result of these larger fillets.

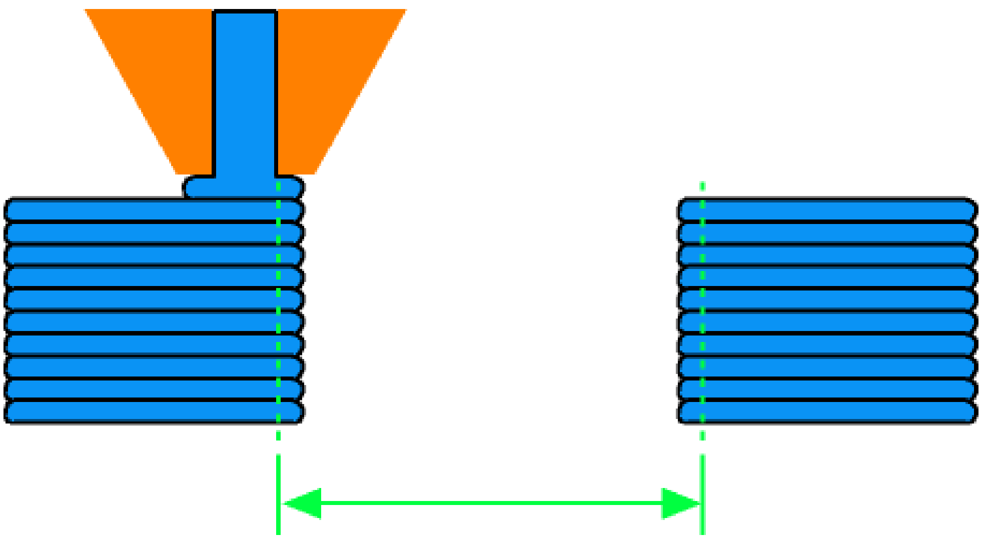


Creating a Test Print

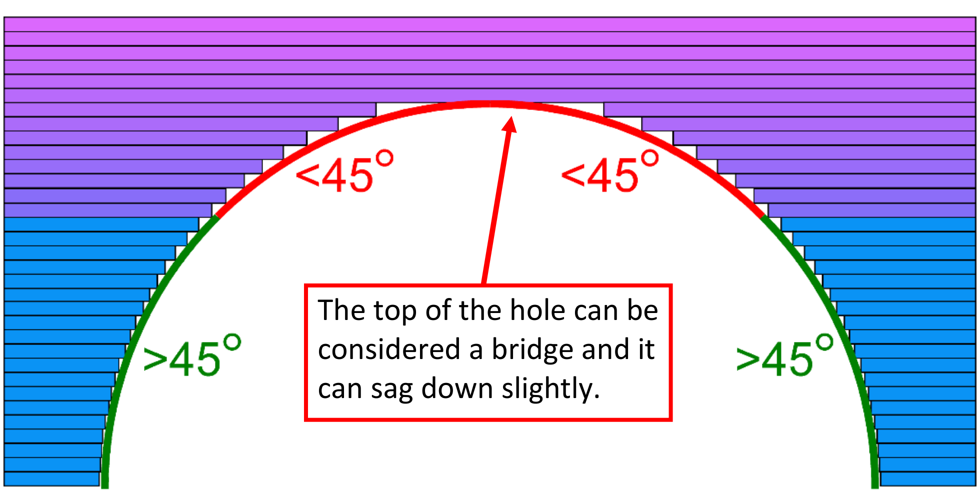
Whenever there are critical dimensions in a design for 3D printing, such as hole diameters, test prints should be done. We only have one vertical hole (with respect to the build plate) and a hole for hanging on a hook isn't critical, however, we will still go through this process.

Note that there are settings in slicer programs, such as Ultimaker Cura, that can make some corrections. One such setting is **Horizontal Expansion**, however this will only help for vertical holes. It can also be argued that for **DFM (Design For Manufacturing)** considerations, the actual design should be adjusted to prevent errors due to invalid slicer options. For example, one may forget to adjust these slicer settings before performing the print, or they may be set and then applied to a future print, for which the settings are not appropriate.

Below are illustrations of the mechanisms that can responsible for vertical and horizontal holes. The resulting dimensional errors can vary between printers and can also be influenced by variations in filament, ambient temperature, and slicer settings. Thus, one can appreciate the importance of performing test prints.



For vertical holes, one mechanism that can be responsible for holes being printed smaller than specified, is the bulging of the plastic upon leaving the nozzle.

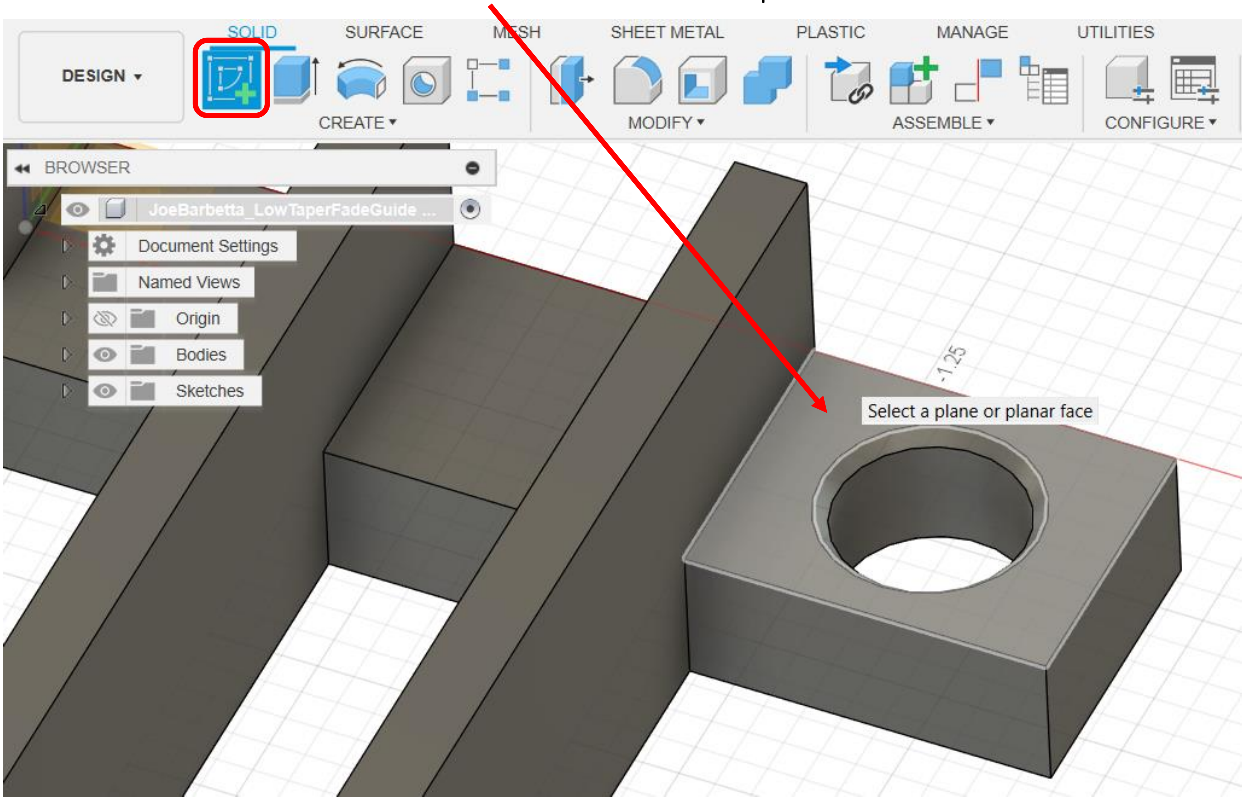


For horizontal holes, the top section of the hole has walls that violate the 45 degree rule. The top layers will "fall in" slightly, thus reducing the diameter when measured vertically.

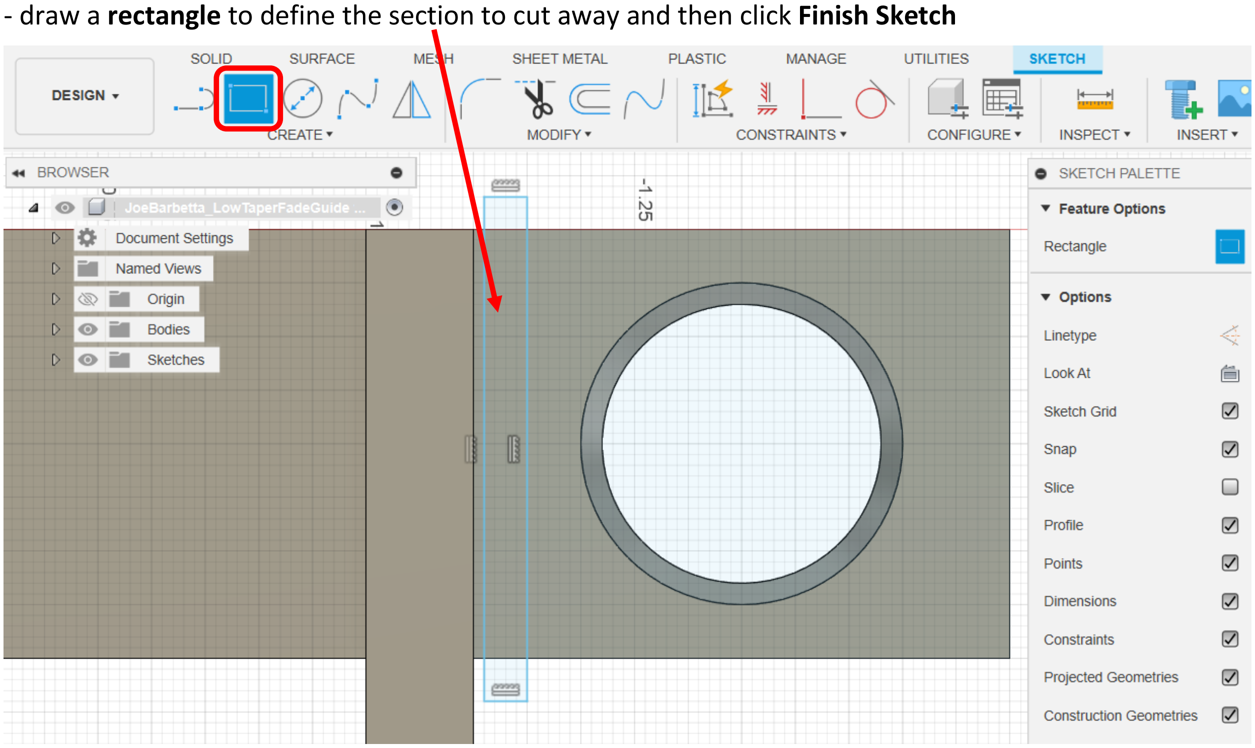
For the test print we will isolate one or more sections of the final design to verify dimensions. We want a small test print part so that time and material is not wasted. If dimensions need to be adjusted, this can be done in the design, and then either a 2nd test print or the final design can be printed.

There are multiple ways to isolate sections of a design for test prints. The method shown here involves creating a sketch and then extruding it the cut through the body. Another method could involve using the Split Body tool.

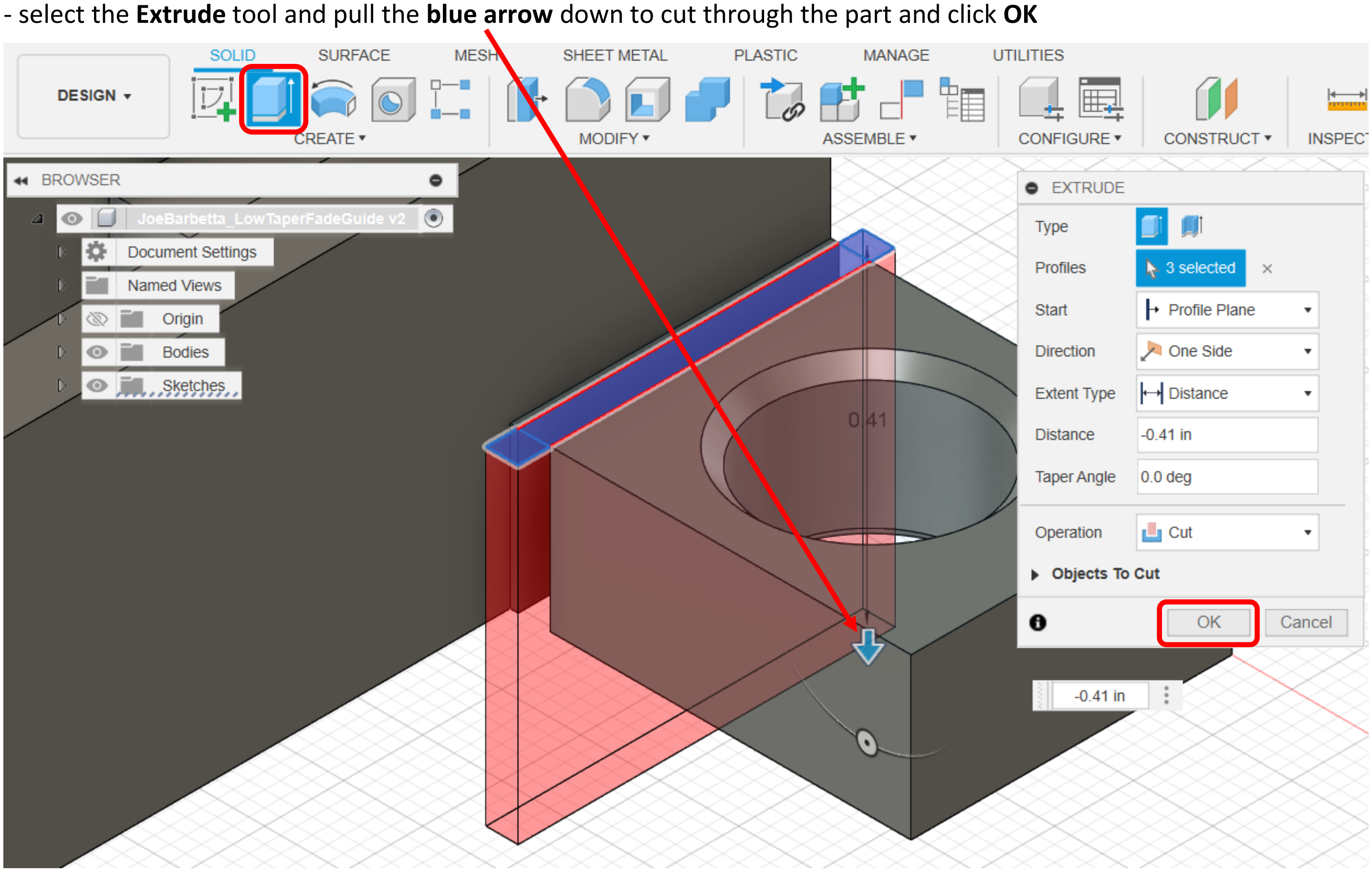
- **save your project**. If many changes are made one can revert back to the previous design.
- zoom into the hole that was earlier created
- select **Create Sketch** and click on a **surface** that includes an important feature



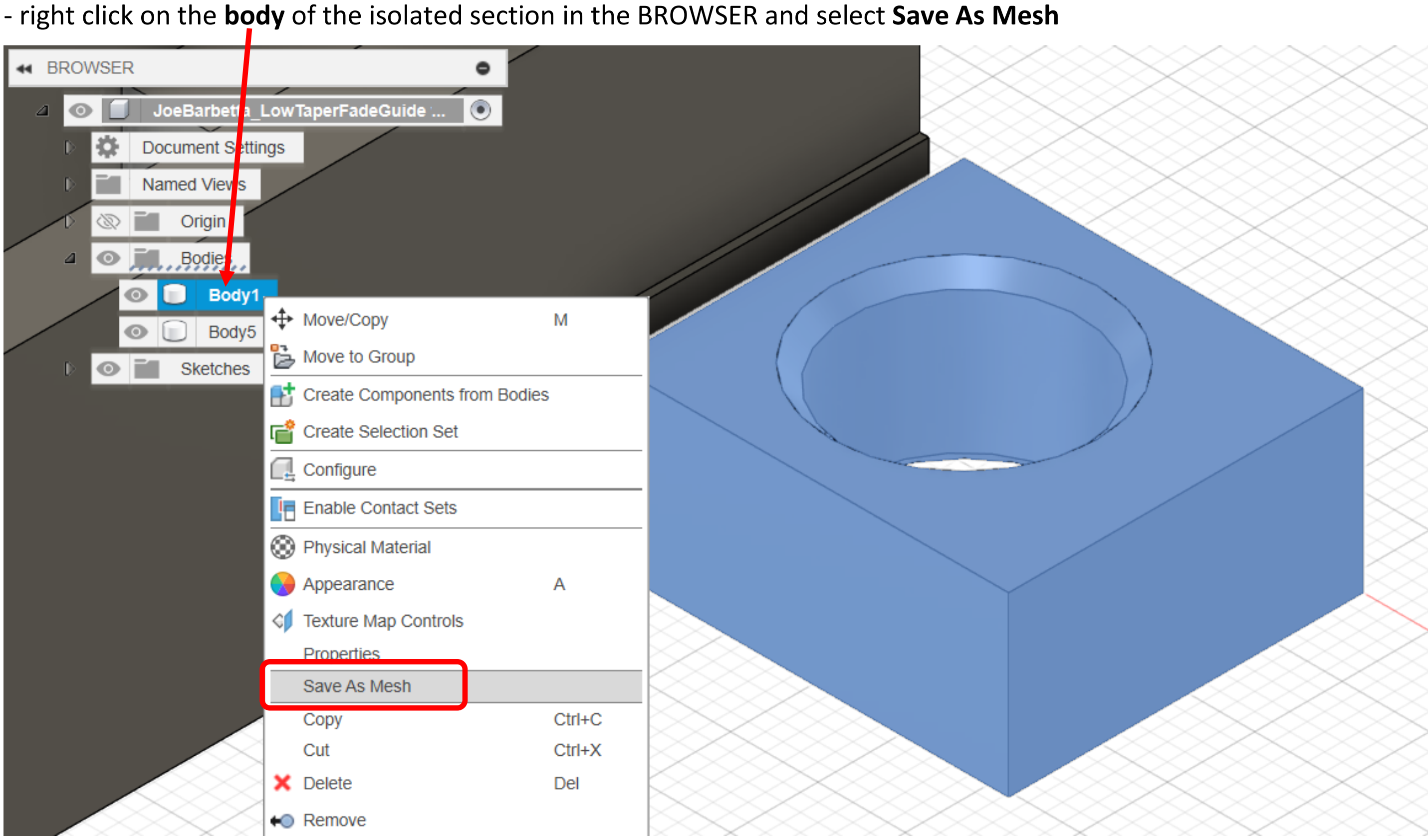
- draw a **rectangle** to define the section to cut away and then click **Finish Sketch**



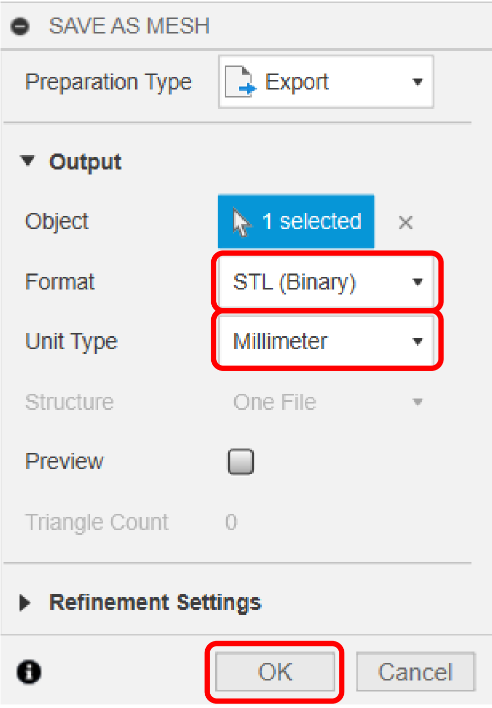
- select the **Extrude** tool and pull the **blue arrow** down to cut through the part and click **OK**



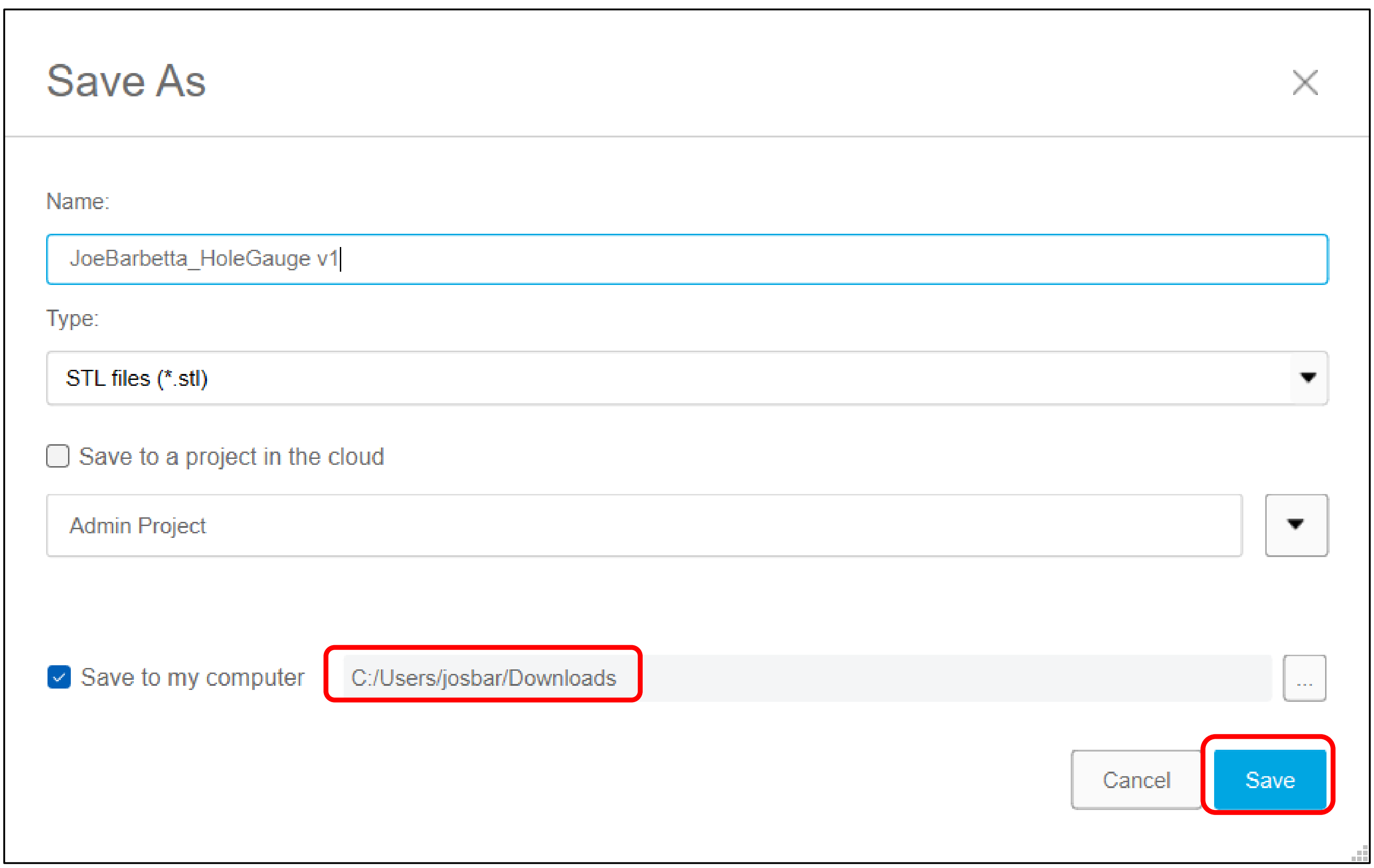
- right click on the **body** of the isolated section in the BROWSER and select **Save As Mesh**



- set the parameters as shown and click **OK**



- click on **Save**



Cura settings

If this is for an assignment, only a screen shot of your design in Fusion is needed. If you were to print your Low Taper Fade guide, the Cura settings for the Brim are shown below.

For Ultimaker Cura, one can scroll to and open the **Build Plate Adhesion** section. By default, Cura should have Brim selected