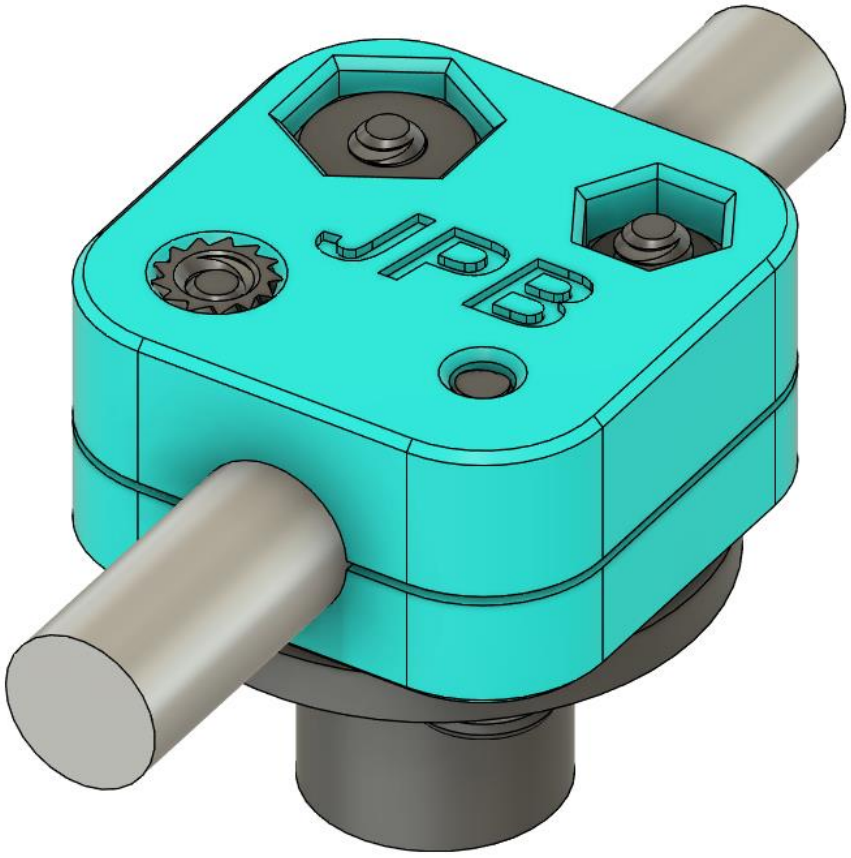
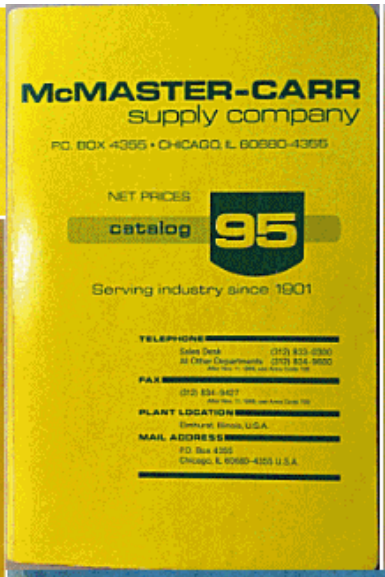
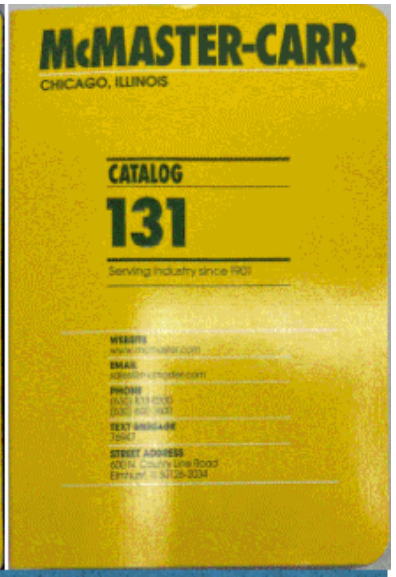
Mr. Barbetta's *"I don't care about 3D Printing. I just want the art credit."* class

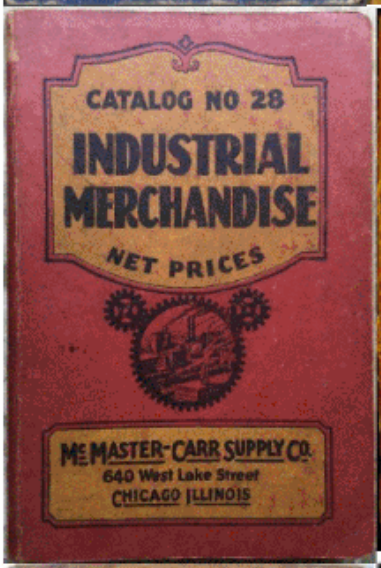
Fastening

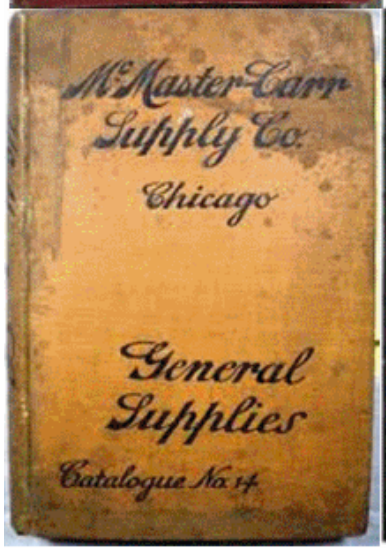


Today's Lesson is Sponsored by McMaster-Carr







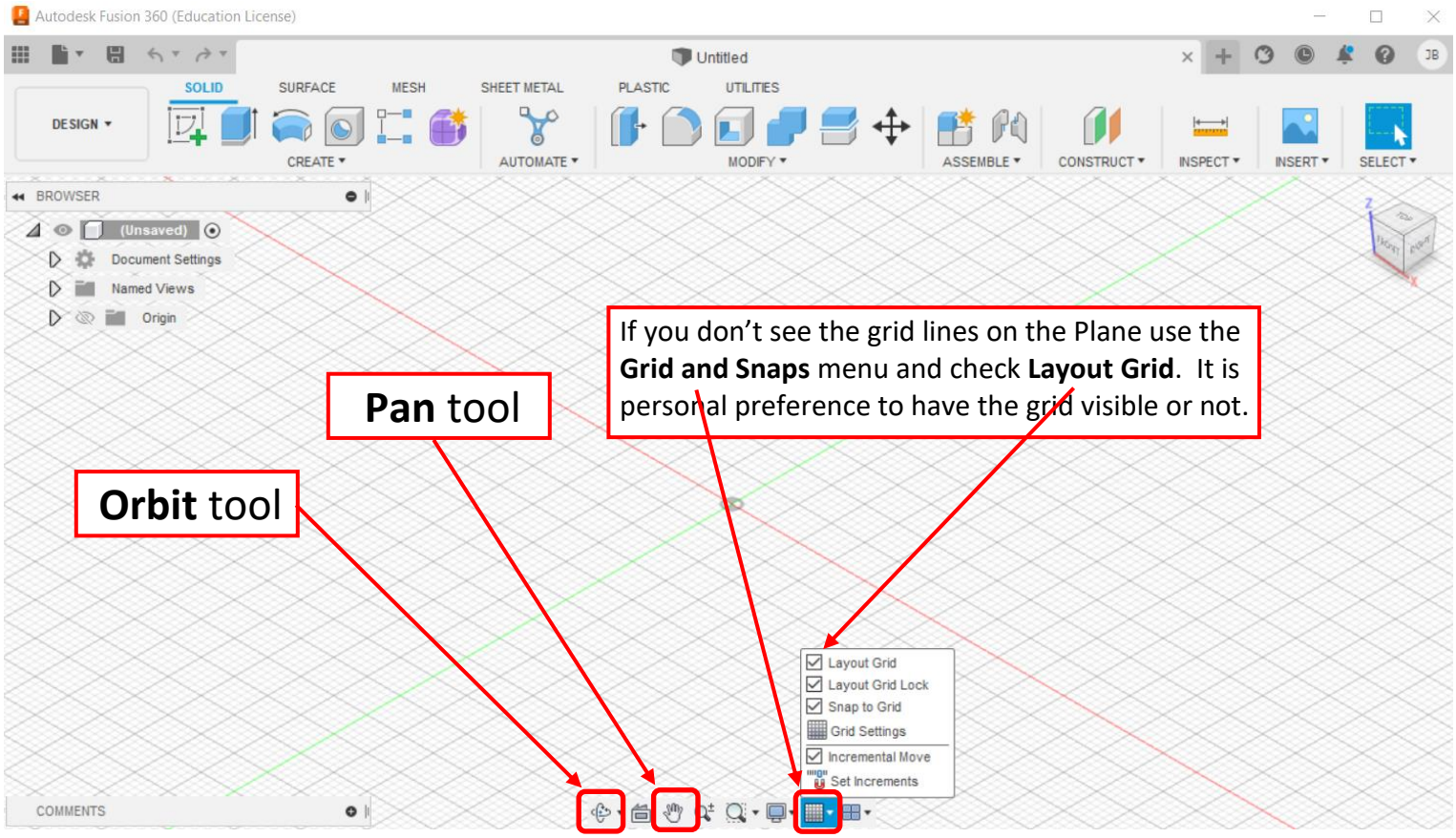


Contents

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

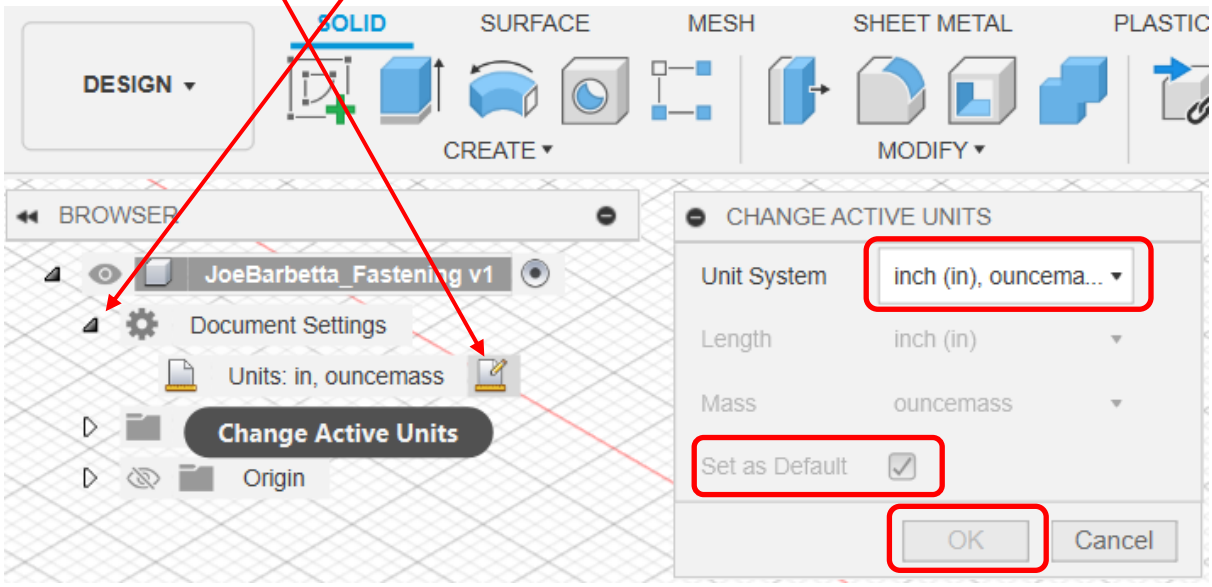
Starting a Design in Fusion (START HERE)

- 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 **_Fastening** e.g. **JoeBarbetta_ Fastening** (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.

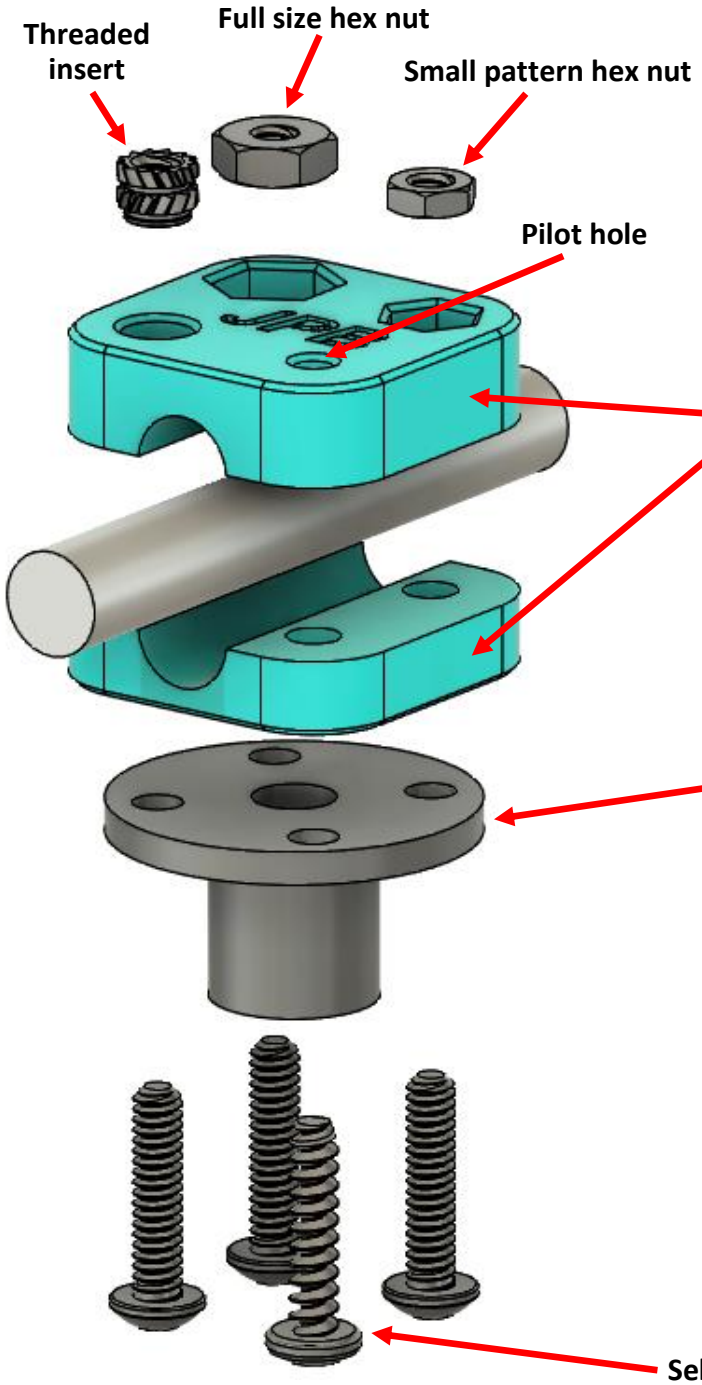
Did you know that the default units have changed over the years? The earliest version used cubits as the default unit.

Creating a Component

This assingment guides one to create a 3D printed mount (2 blue parts) to secure a 1/4" diameter rod to a flange that would be fitted to a shaft (not shown).

- yell “I will never need to secure a rod to a flange. This is a dumb assignment.”

Even if you never need to attach a rod to a shaft, this assignment teaches various methods for attaching parts, which can be applied to many different design scenarios. This can include “pure” 3D printed designs where 3D printed parts need to be connected together.



The top mount design demonstrates four different connection methods. Of course one would not employ four different methods in **one part**. One would select the method that is best for a specific application.

Note that hexagonal pockets are created for the nuts. This eliminates the need for a socket or wrench to tighten the nuts. It also provides a more professional look.

A **split mount** is implemented to best secure the rod. If the rod passed through a hole in a single block a set screw or other means would be needed to secure the rod. This also provides an accurate fit if there are dimensional variances in the rod or the printed part.

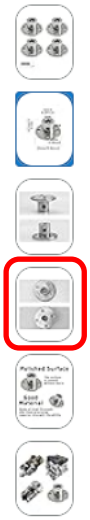
This is a purchased part for connection to a shaft. However, theses attachement means can be applicable to many diffeerent scenarios.

Three of these screws are machine screws that get fastened with nuts or an insert.

The screw with course threads is a self-threading screw, which is fastened right into a plastic body. This is the least expensive method because no insert or nut is needed. It is not a strong as the other methods, but could be ideal for attachments that are not subjected to large forces.

For this design scenario, the part the 3D prints are being attached to is a Flanged shaft coupling. Below is one such product sold on Amazon. Note that this listing shows several dimensions of the part. However, the most important dimension for our needs is missing. There is no dimension showing the positions of the hole. There could be a dimension showing the distance of each hole from the center or distance between holes.

Fusion has a Canvas feature where an image can be imported and scaled to digitally “trace” features.



4Pcs 5mm Flange Coupling Connector, Rigid Guide Steel Shaft Coupler, High Hardness Guide Model Accessory for DIY RC Motor Axis Fittings

Brand: Runseaway

4.6 ★★★★½ (3) | Search this page

$5.59

Price history

prime Overnight

Size: **5mm**

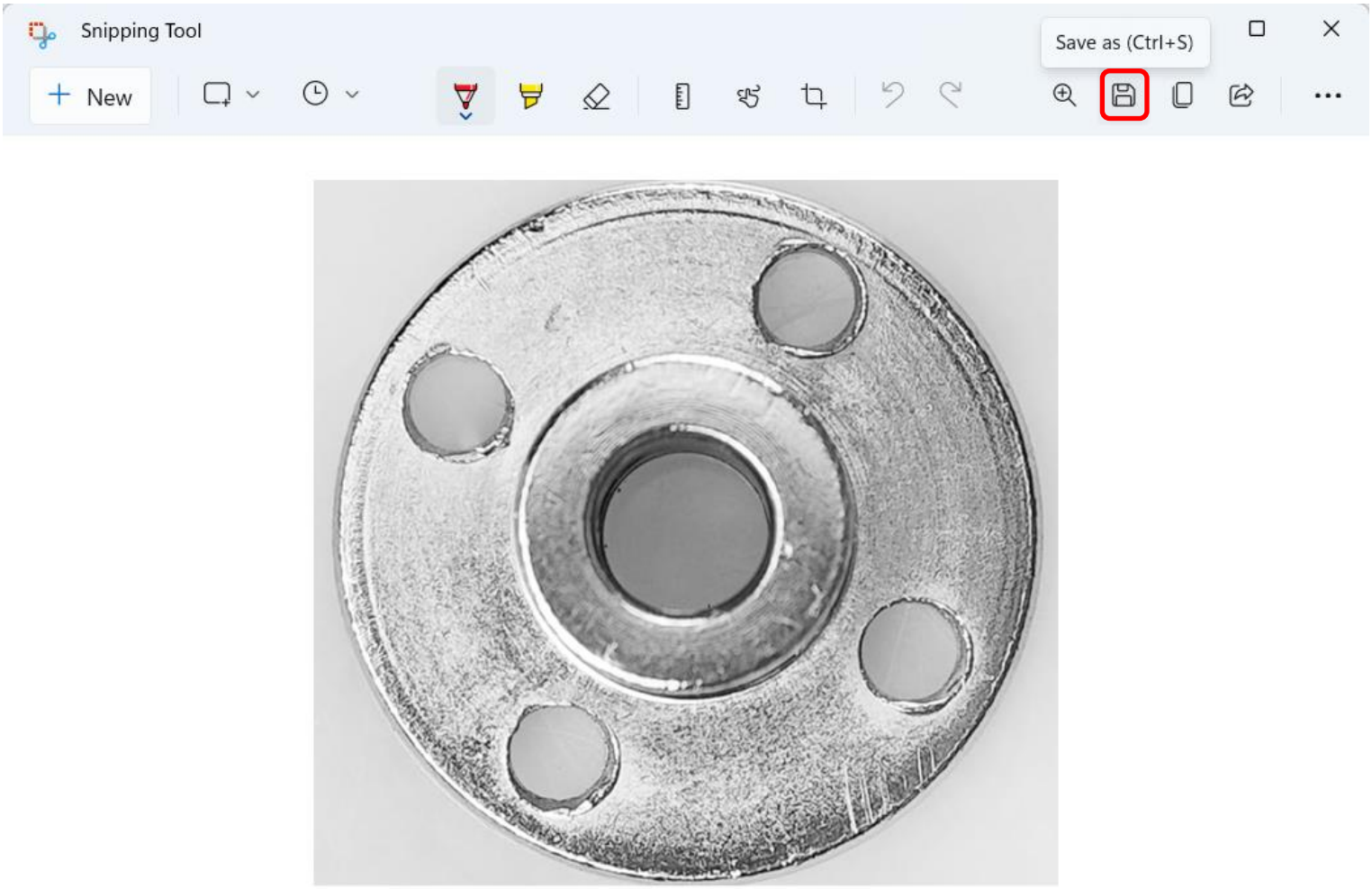
5mm	6mm	6.35mm	8mm
$5.59 FREE Delivery **Overnight 7 AM - 11 AM**	$6.59 FREE Delivery **Tomorrow**	$5.99 FREE Delivery **Tomorrow**	$7.59 FREE Delivery **Tomorrow**

Number of Items: **4**

In the listing there are images on the left side of the listing that can be expanded. One can take a screen shot of an image that best shows the layout of the holes on the part. Below is the best image that meets this need.

- press the **Windows, shift, and s keys together** and extend the selection rectangle over the **below image**. Do your best to exxtend the selection rectangle so the the image below is in the center of the rectangle. It is fine if white around the image is within the screen shot.

- at the bottem right of the Windows screen, a view of the screen shot should show. Click on it to show the Snipping Tool window show below.
- click on the **File icon** at the top right and save the .png file to the Downloads folder

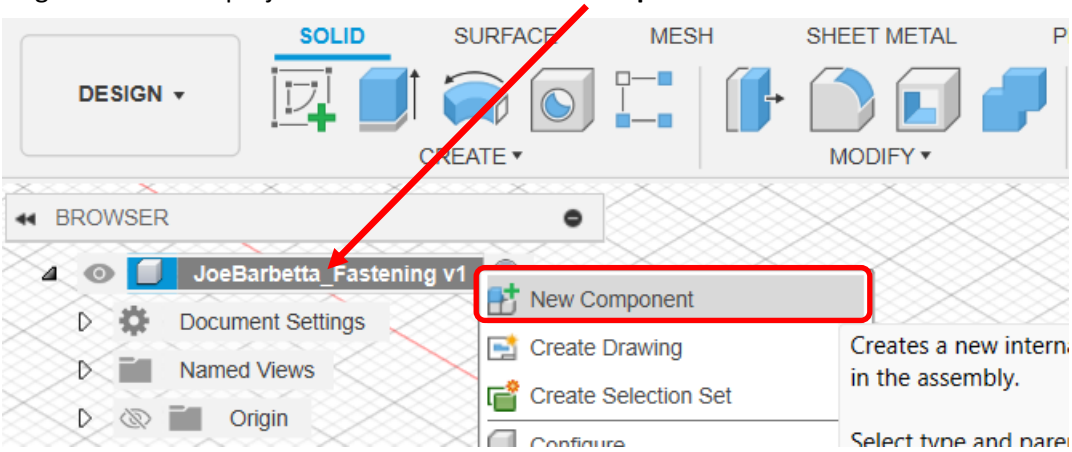


Continued on the next page.

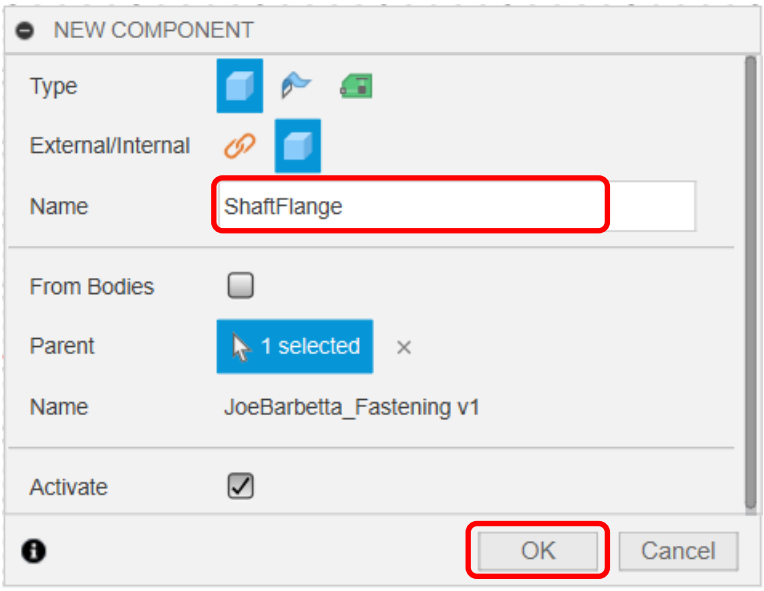
Creating the Off-the-Shelf Component

Sometimes a STEP file is made available for a purchased component, as is the case for many parts purchased from McMaster-Carr. In some cases one needs to design the part itself in Fusion. One could argue that the dimensions of the purchased component can simply be used to design the 3D printed parts. However, having a model of the component in Fusion provides a visualization of the entire assembly.

- right-click on the project name and select **New Component**

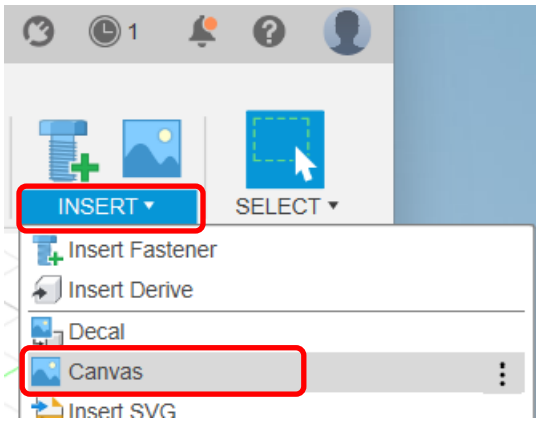


- enter **ShaftFlange** for the **Name** and click **OK**. For other scenarios, this would be a descriptive name for a different part.

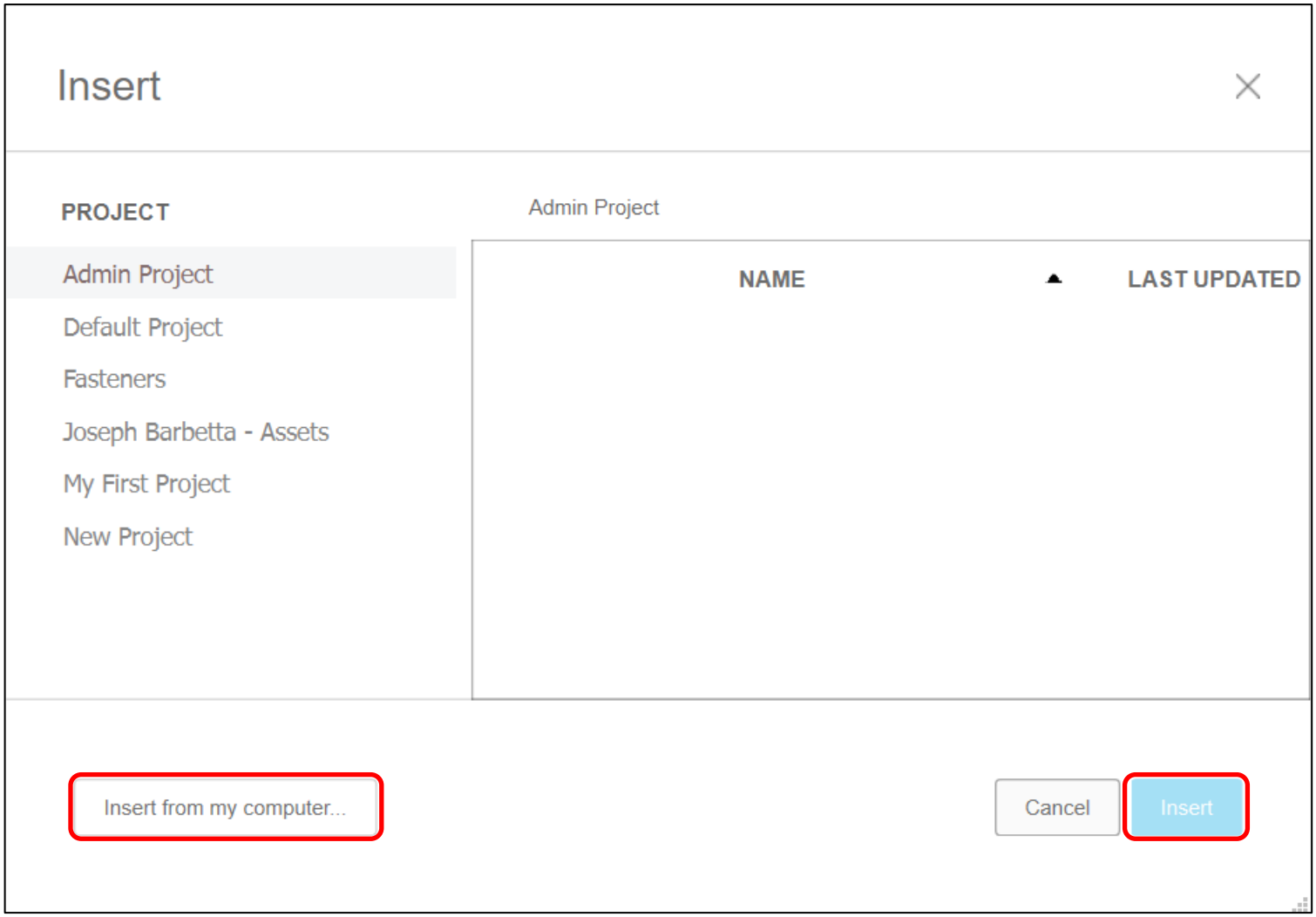


Using the Canvas Feature

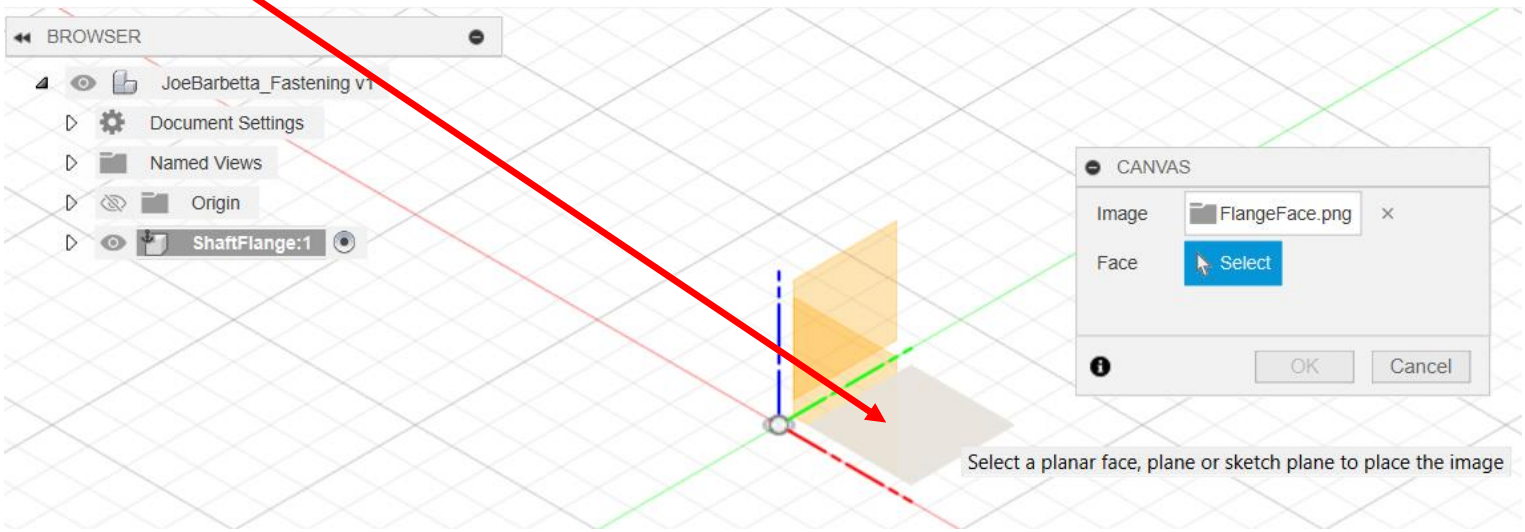
- from the **INSERT** menu at the top right of the Fusion screen, select **Canvas**



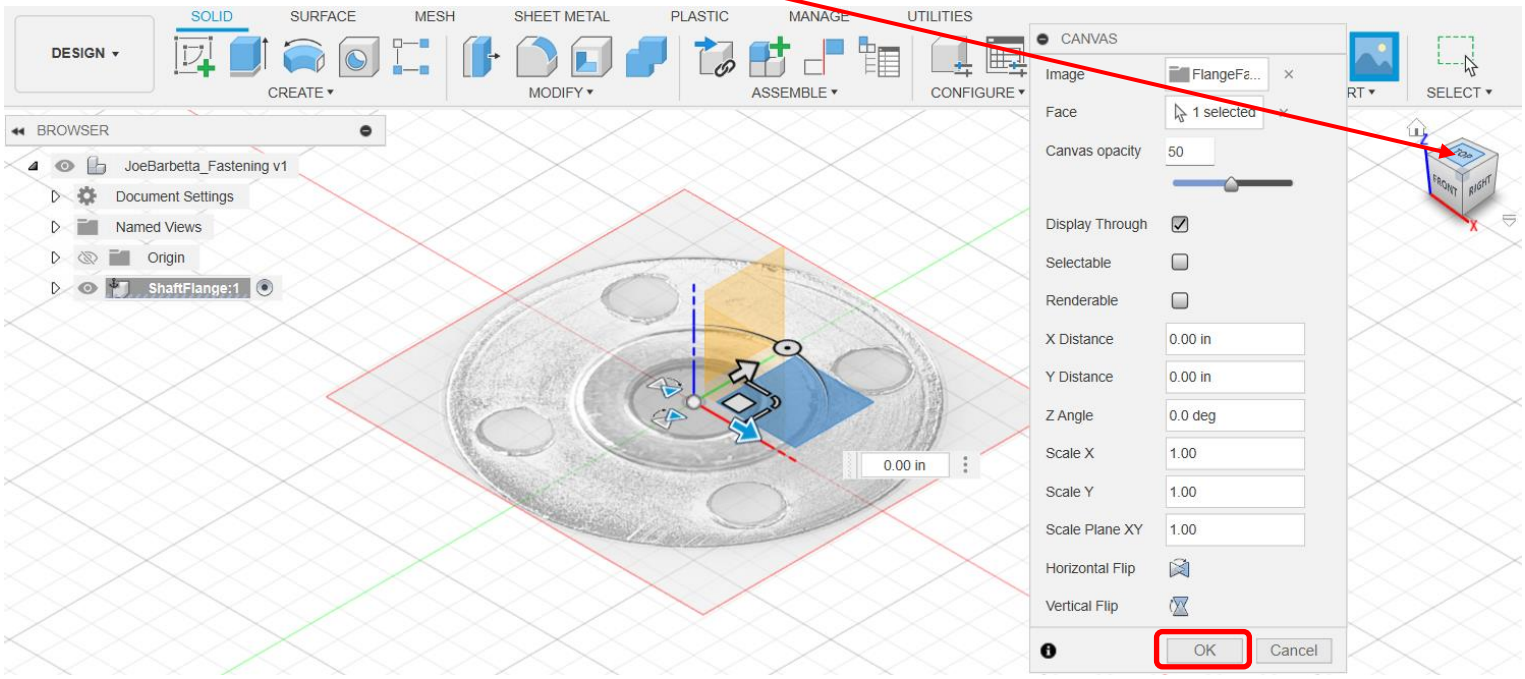
- click on Insert from my **compoter…** button and select the Downloads folder and then the .png file that was saved from the Windows Snipping Tool.
- click **Insert**



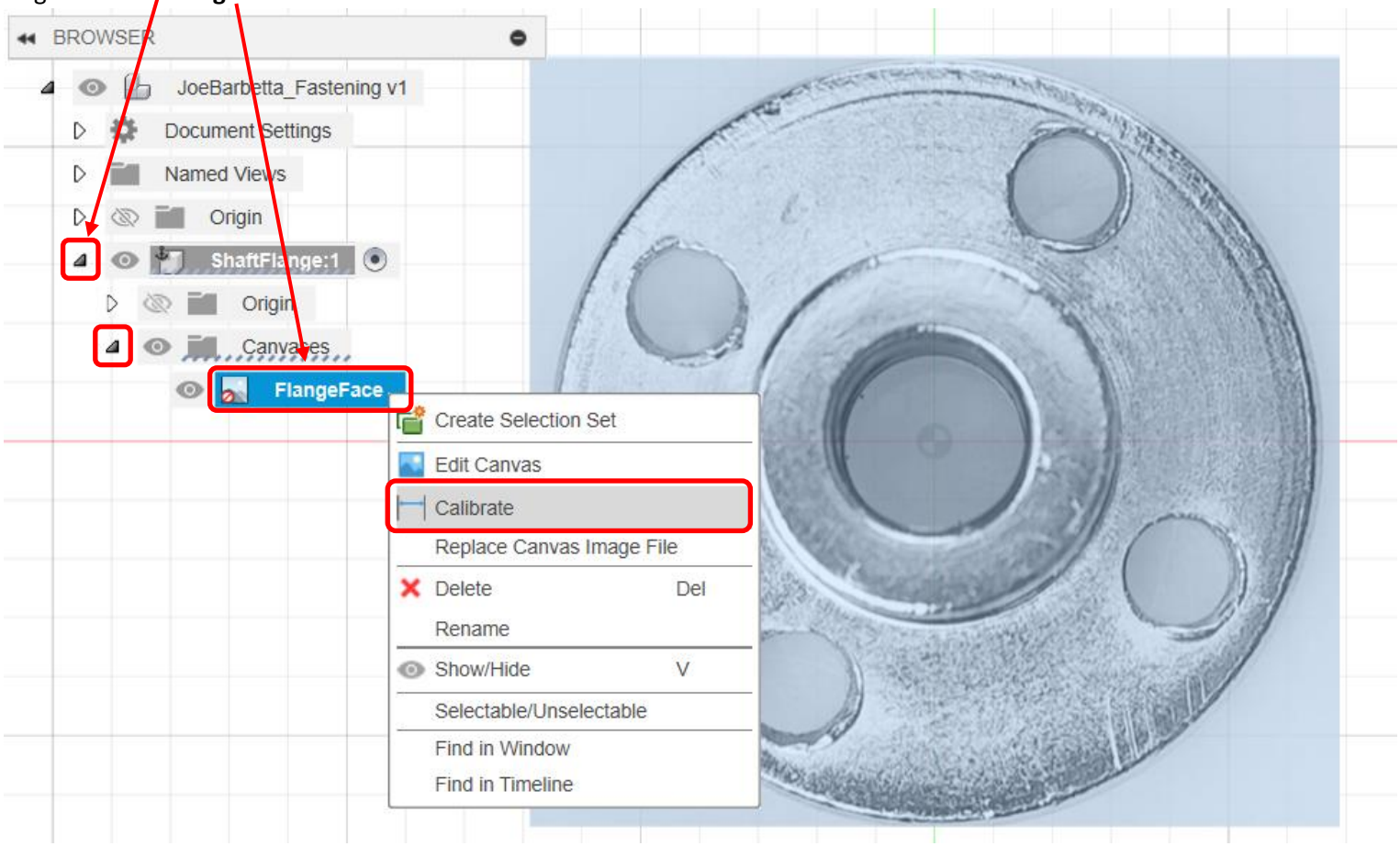
- click on the **bottom plane**



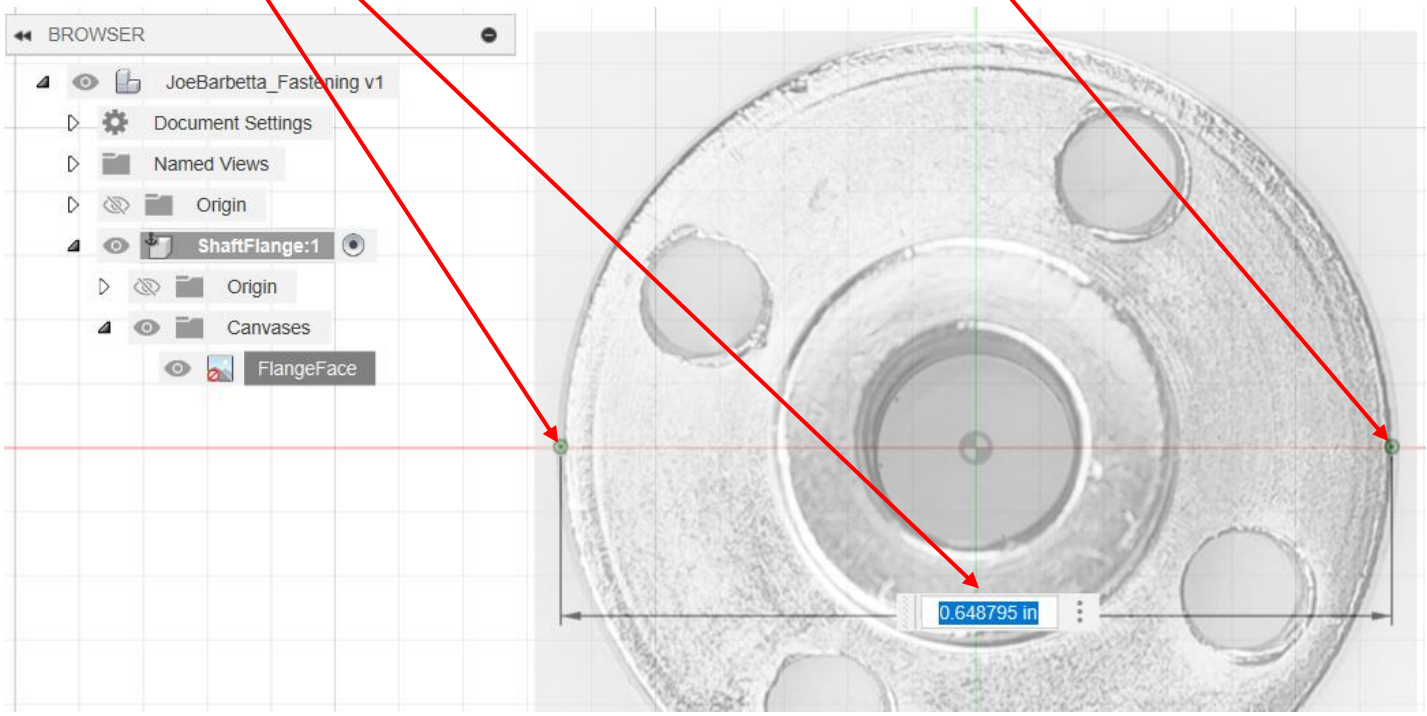
- click on the **TOP surface** of the **View Cube**
- click **OK**



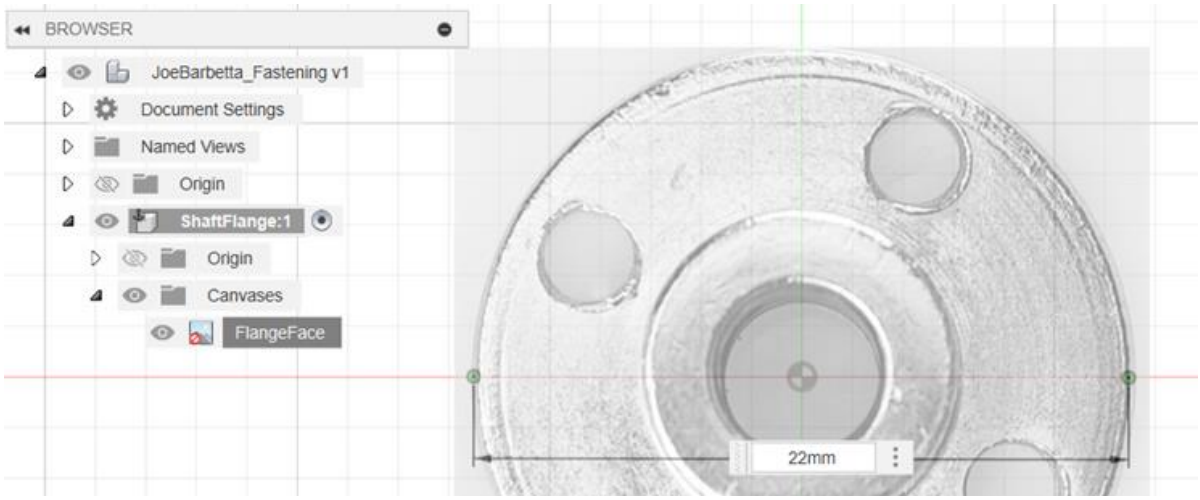
- zoom so the the flange image takes up most of the Fusion screen
- click on the **arrows** next to **ShaftFlange** and the **Canvases** folder to open them
- right-click on **FlangeFace** and select **Calibrate**



We know the flange diameter is 22 mm from the dimensions given for the part on Amazon.
- click on the **left edge** of the flange to set a point and then click on the **opposite edge** of the flange to set the 2nd point
- type **22mm** in the **value box** and press the Enter Key. Note that typing mm after the 22 will cause Fusion to convert the value to inches. Do not just type 22 because you will end up with a 22 inch flange!

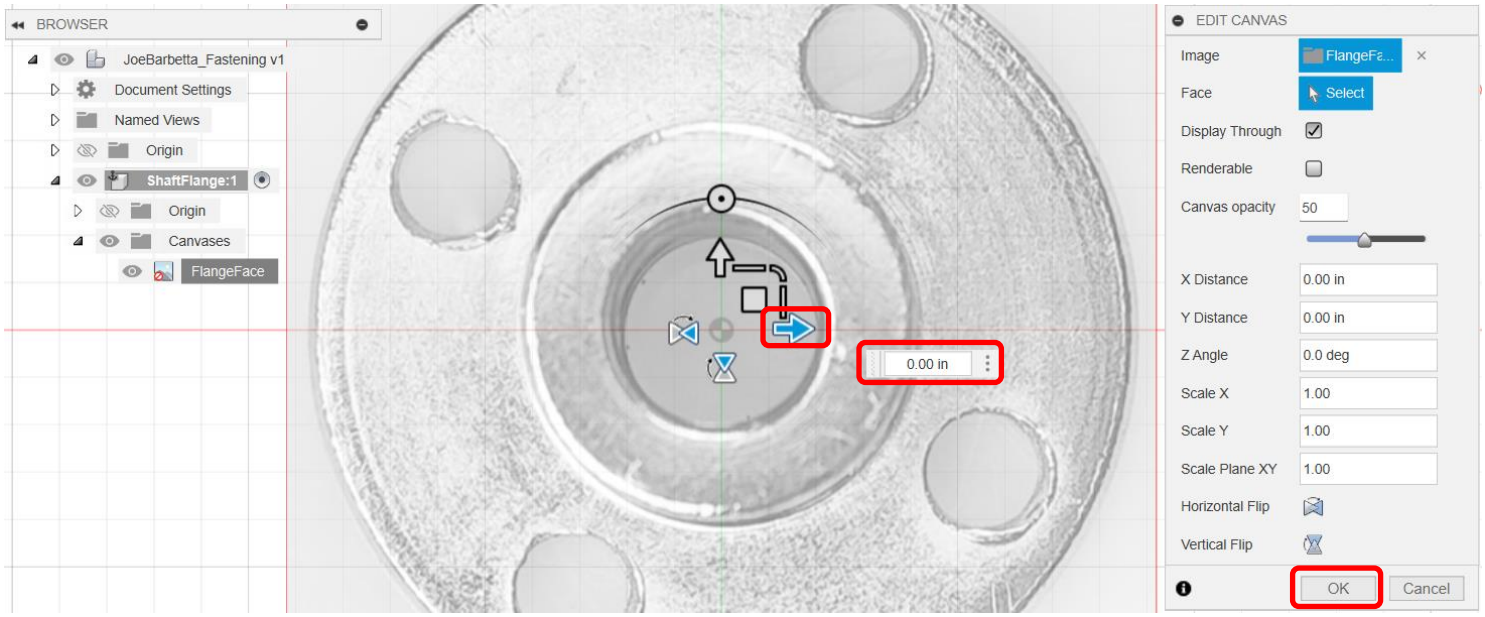


The Canvas image will be scaled so that the 2 point defined will be 22 mm apart and now the image is at a 1 to 1 scale. This will easily allow one to determine the size and/or positions of features.



- right-click on the **Canvas name** and select **Edit Canvas**

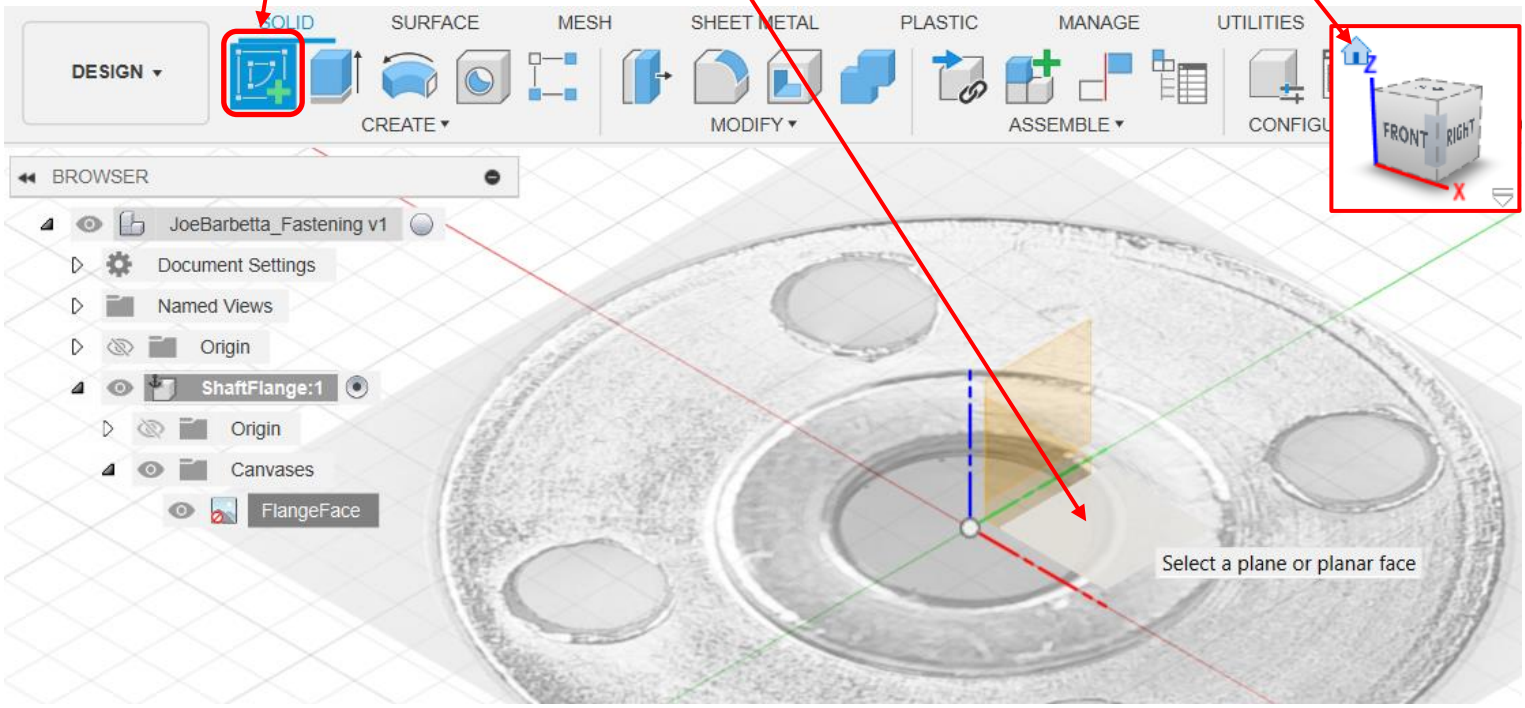
- if the Canvas image is **centered on the Origin**, as this is, one can click **OK**
- if the image needs to be shifted up/down or left/right, one can click on a **blue arrow to drag it or enter a value in the box**. For very small adjustments, entering a small value, such as 0.005, can make very small shifts. A canvas image need not be centered on the Origin, but if is circular, it can make it easier to create concentric features.



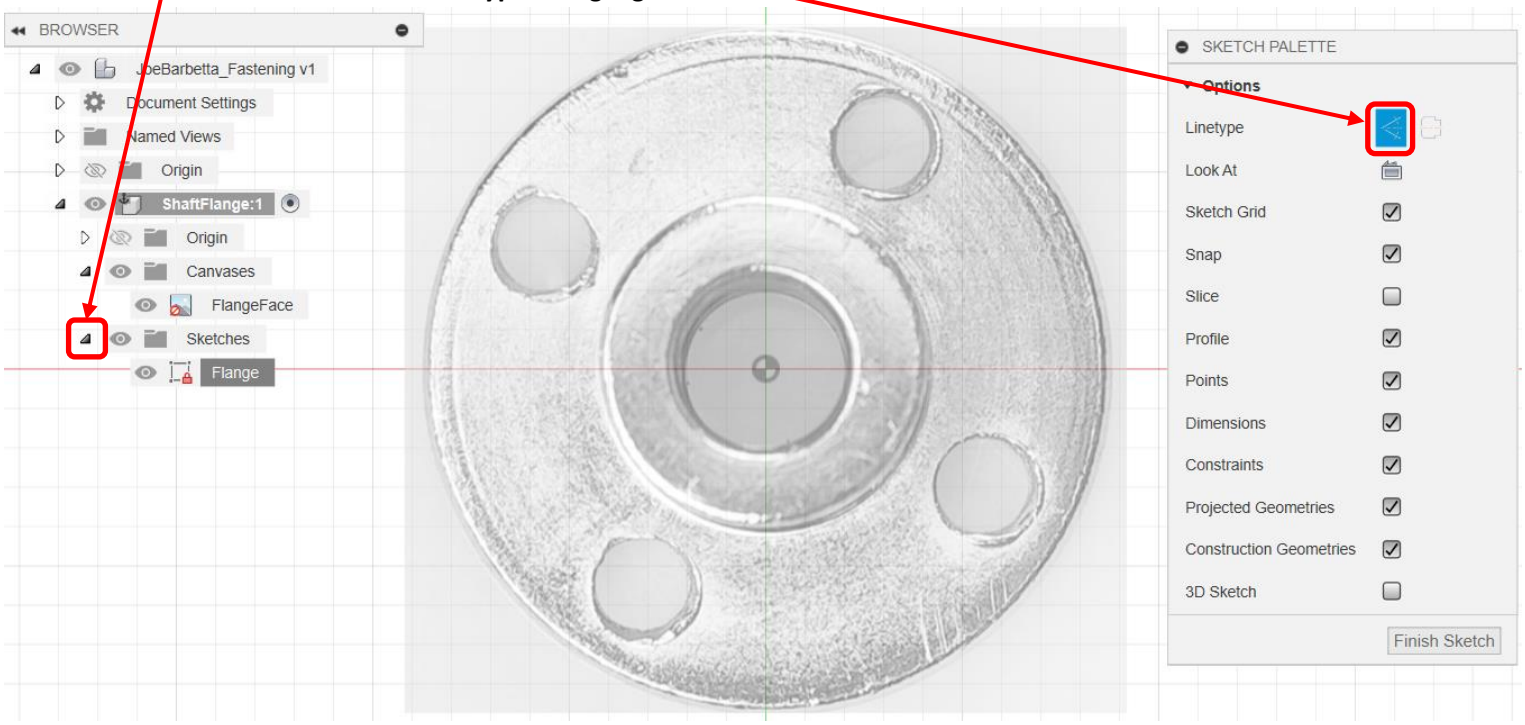
Creating the Flange Sketch

- click on the **Home icon** at the **View Cube**
- select the top **Create Sketch** tool and click on the **bottom 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.



- click on the **arrow** next to **Sketches** folder to open it
- right-click on the **Sketch** name, likely Sketch1 at first, and select **Rename**
- enter **Flange**
- click on the **Construction** icon for **Linetype** to highlight it blue

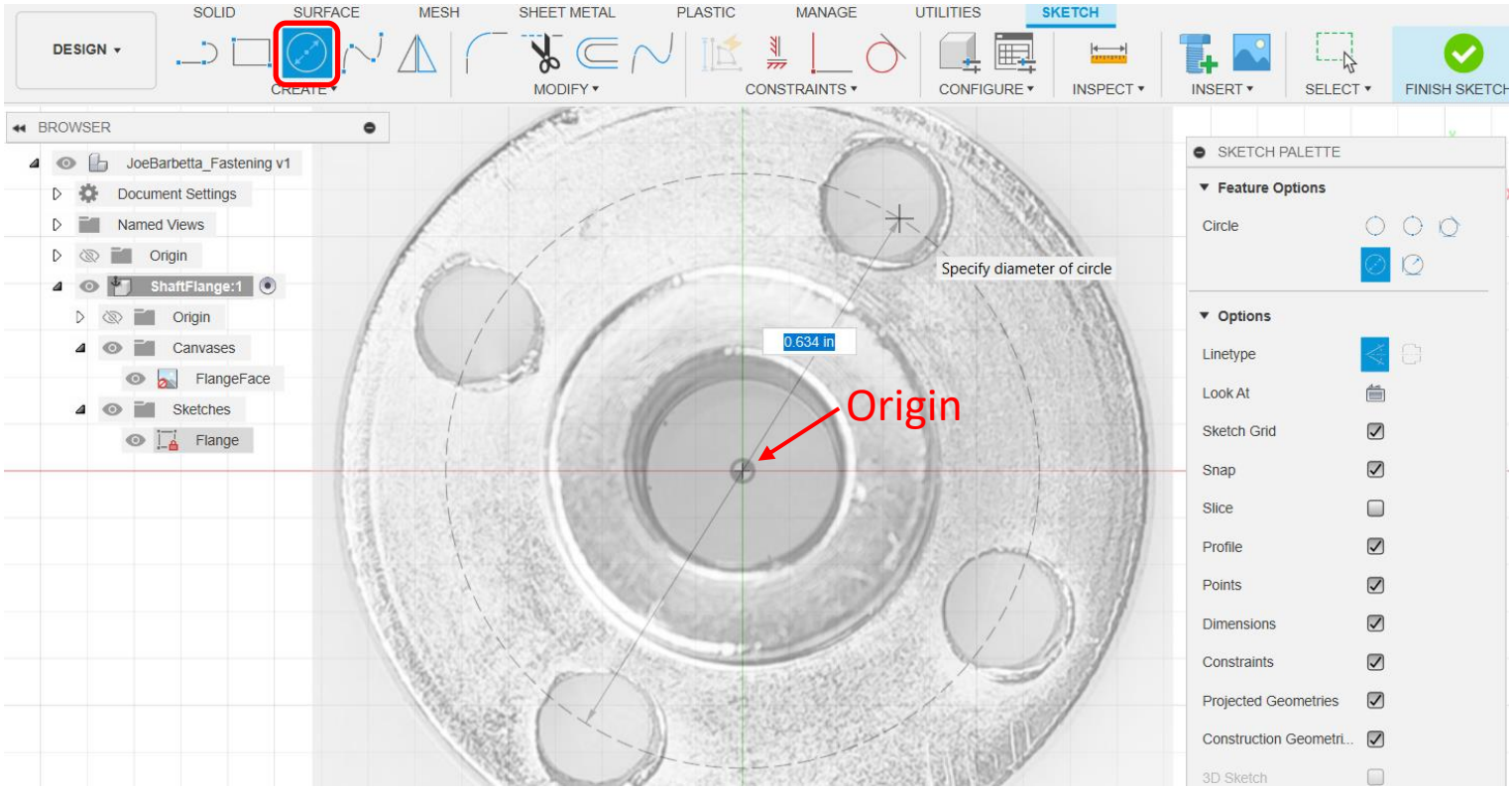


The reason for importing an image of the part is to determine the hole positions, which were missing from the dimensions on the Amazon page. Note that this method can be used to determine dimensions for other parts as well. Nothing beats getting an actual part and measuring sizes and positions with a caliper, but if one wishes to get the design started using an image can serve that purpose. Later on the dimension can be adjusted when the actual part is measured.

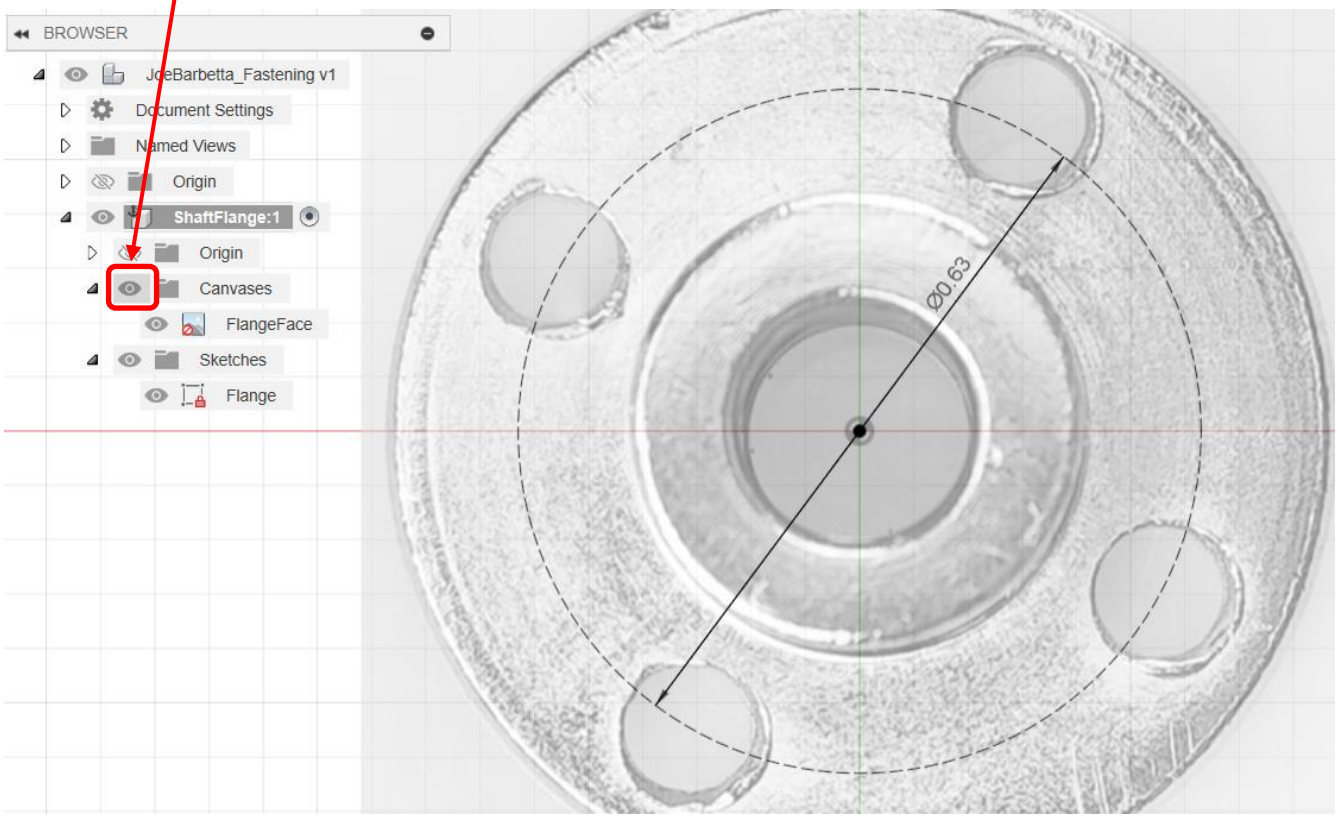
- select the **Center Diameter Circle** tool. If it is not visible, find it in the CREATE menu
- click on the **Origin** and **extend the circle out** until it looks like it is passing through the center of the holes and make note of the value, which here shows as 0.634 in, and press the Esc key to end the operation. If you end up creating the circle, one can use Undo.

Because the other dimensions given are “nice” integer values in millimeters, ie 22mm, 3mm, etc., this is a metric part. It is a good bet that other dimensions will also be “nice” metric values. Converting 0.634 inches to miliimeters by multiplying by 25.4 results in 16.1036. This dimension is likely 16mm.

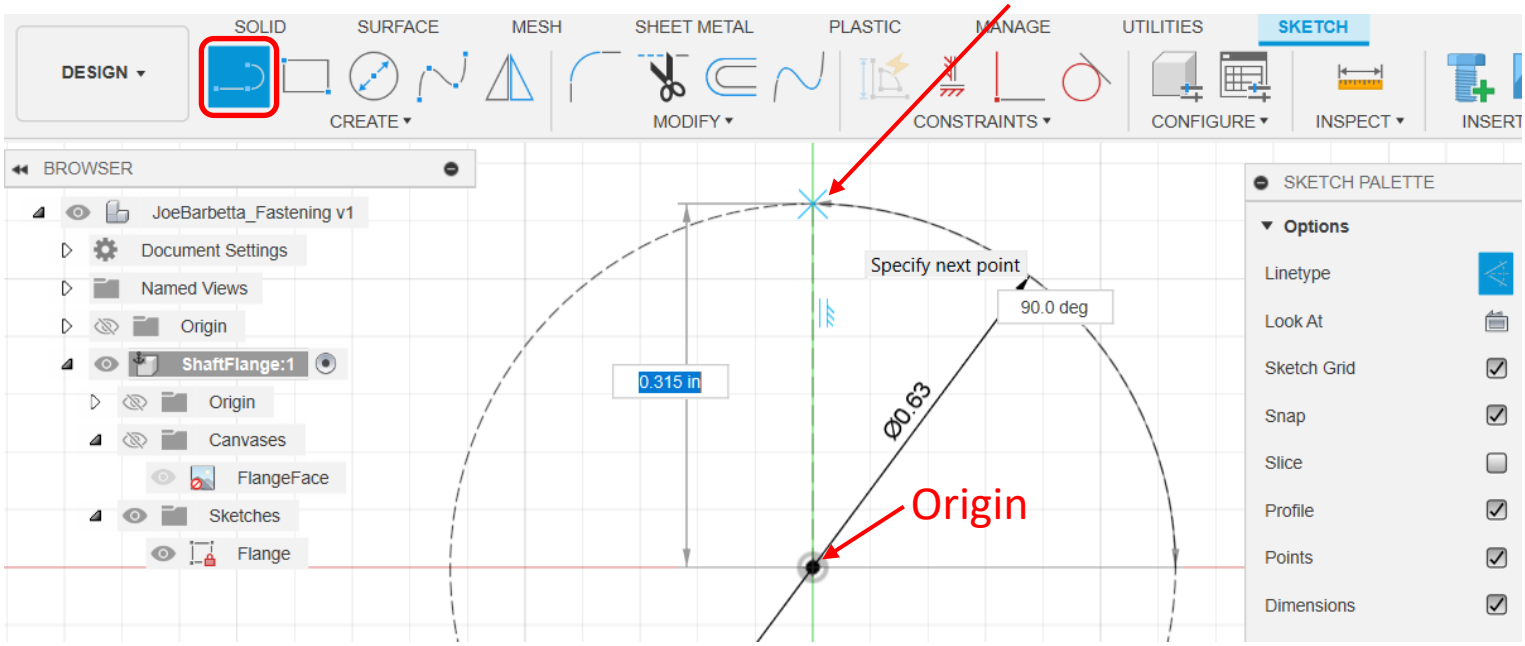
- select the **Center Diameter Circle** tool again and this time enter a value of **16mm** for the circle size



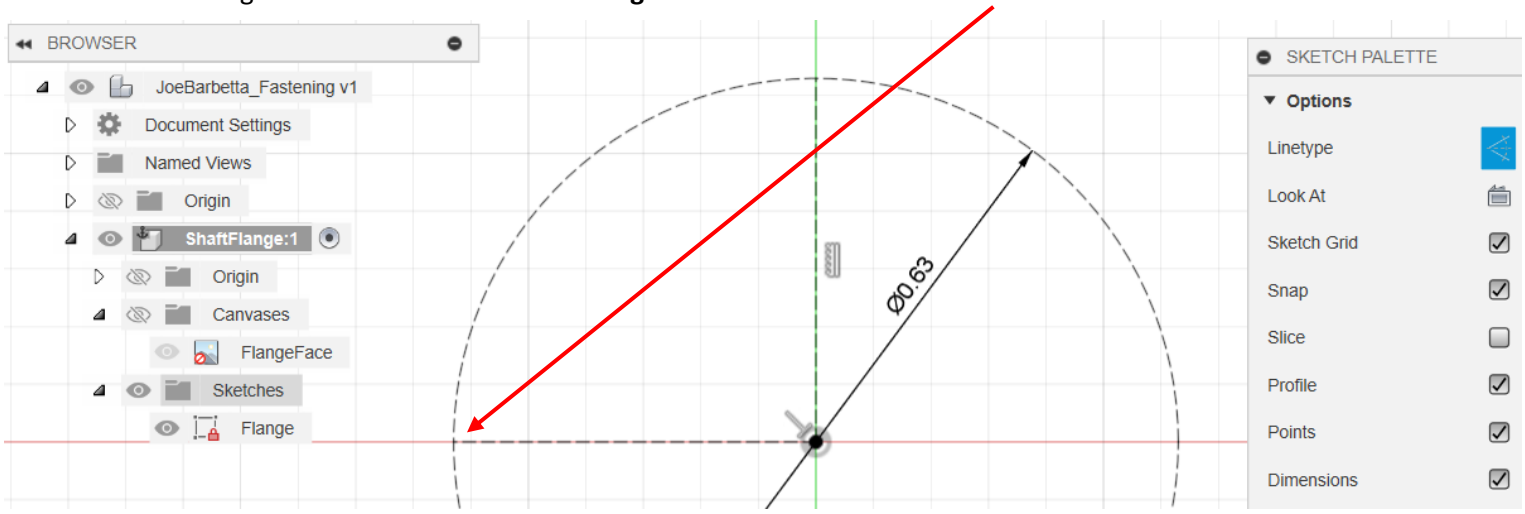
Note that the dimension was converted to 0.63, which is the rounded value of 0.6299.
- click on the **eye** icon next to the **Canvases** folder to hide the image. We only needed it to determine the dimension of the circle that the holes lie on.



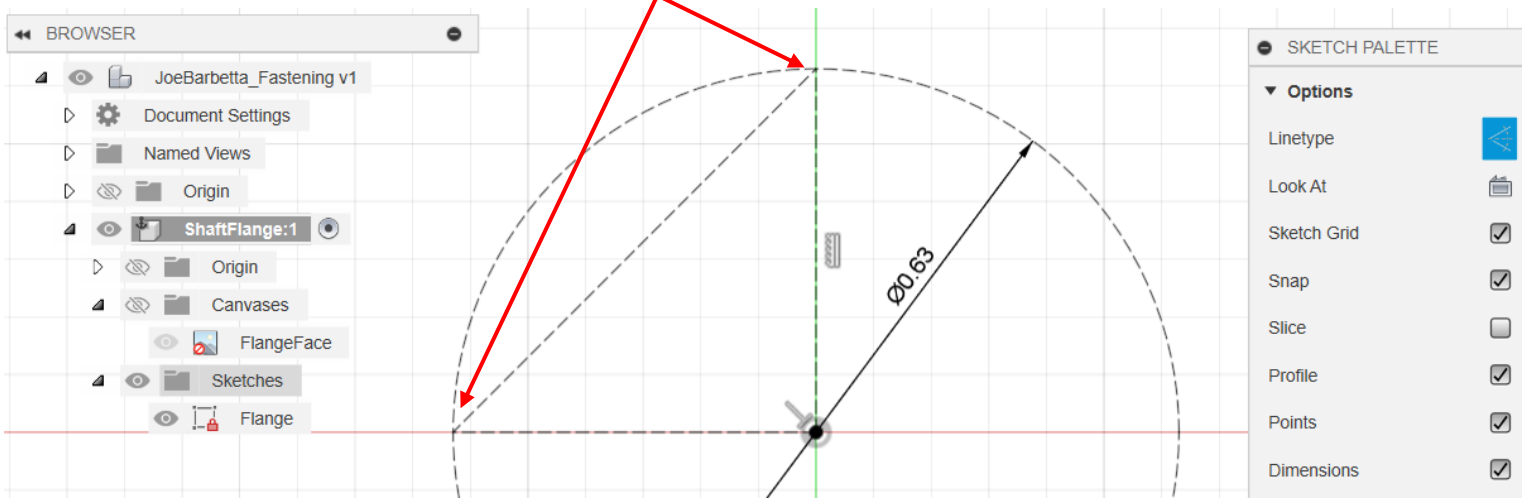
- select the **Line** tool
- click on the **Origin** and extend the line upward until it **meets the circle**, where a **blue x** will appear, and click on that point



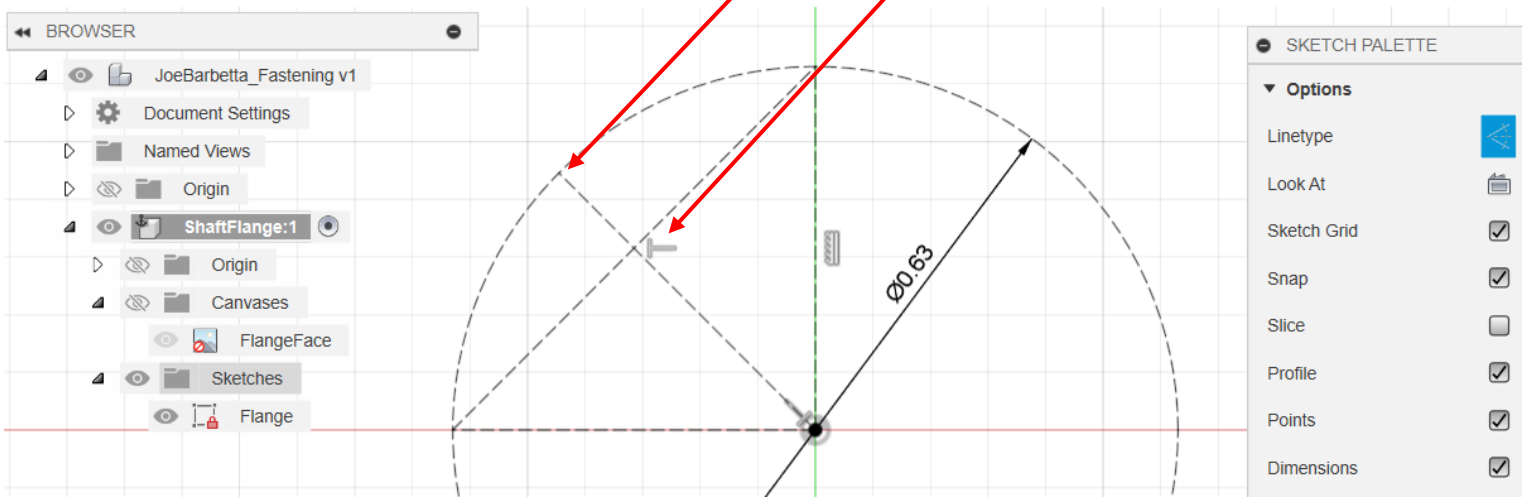
- use the **Line** tool again to create a line from the **Origin** and extend it to the **left to the circle**



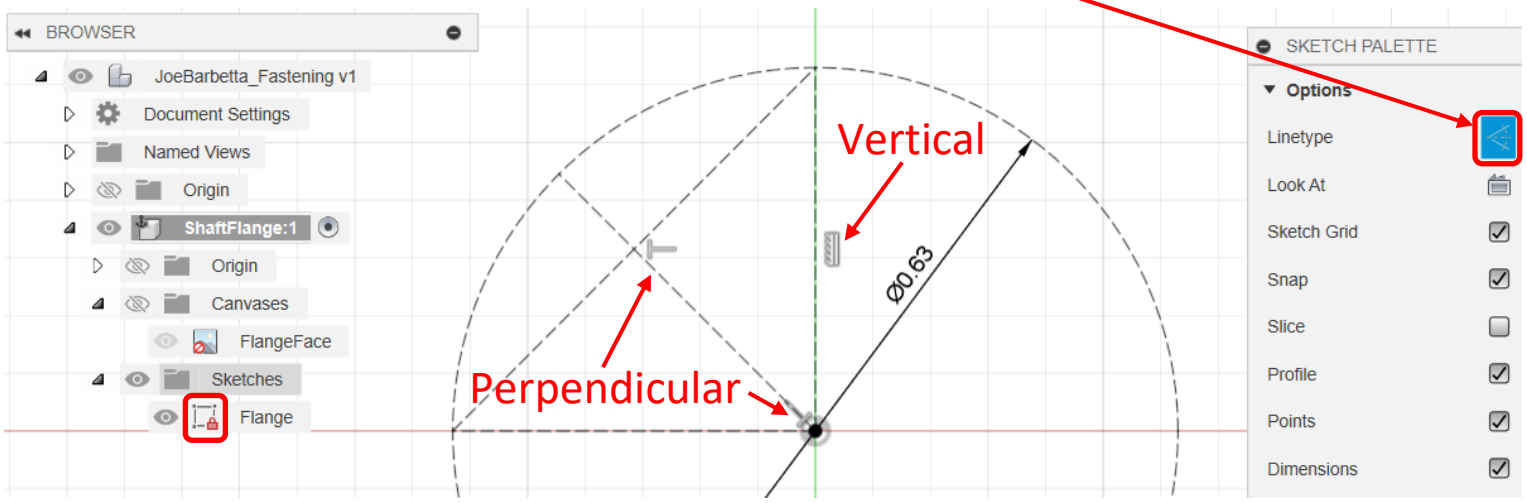
- use the **Line** tool again to create a line from the **end points** of the 2 lines just created

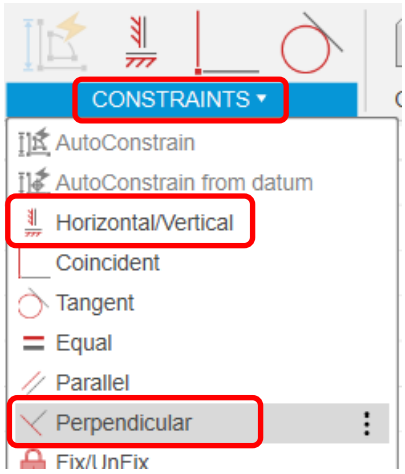


- select the **Line** tool again and create a line from the **Origin** to the **circle**. It should snap to a position where it is perpendicular to the diagonal line previously created, which should result in the **Perpendicular symbol** appearing.
- yell **"Wow this looks like a crazy geometry proof!"**



- click on the **Construction** icon for **Linetype** to remove the blue highlighting
- check if the **small red Lock icon** is shown on the Sketch symbol. This indicates that the Sketch is fully constrained.





- if the red Lock icon is not shown, try the following using options from the CONSTRAINTS menu at the top center of the Fusion screen

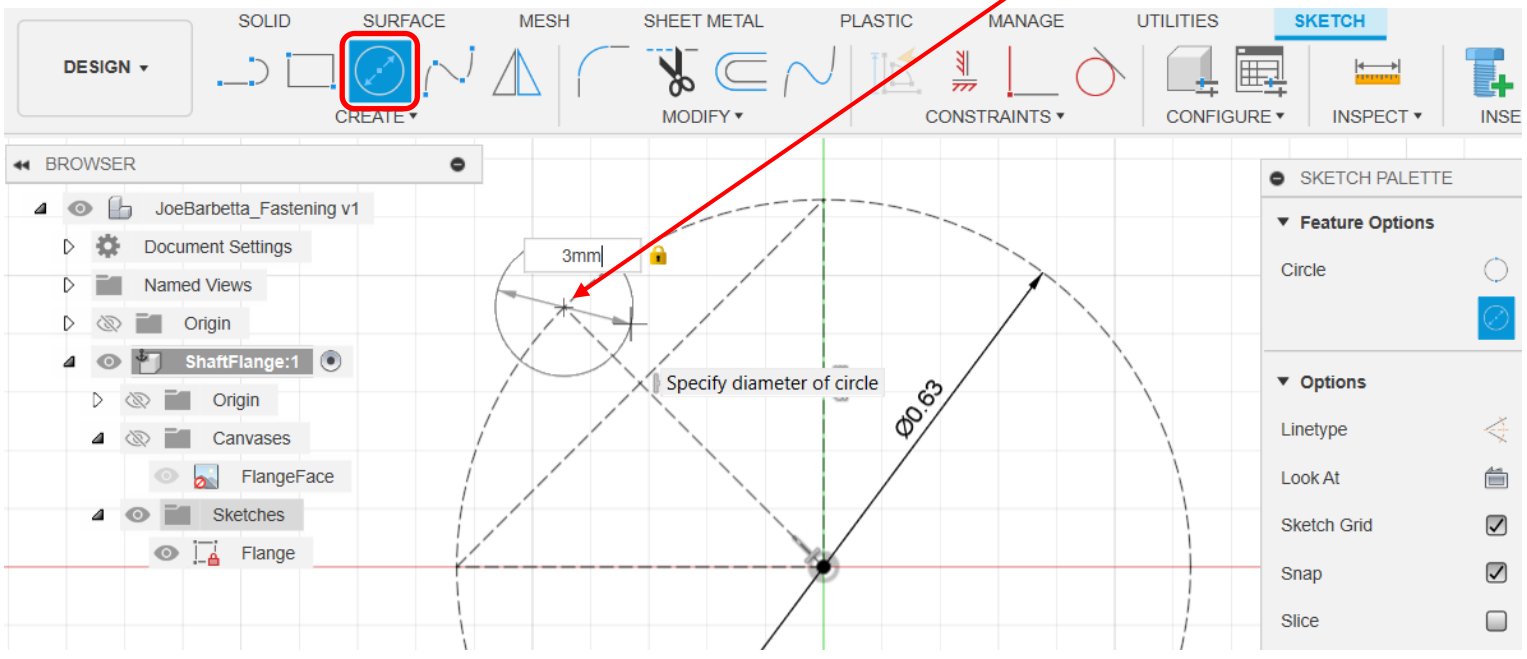
- check if the the Vertical symbol is visible. If it is not select Horizontal/Verical from the CONSTRAINTS menu and click on the vertical line.

- check if the Perpendicular symbol is visible at the Origin. It is hard to see. If not there, select Perpendicular from the CONSTRAINTS menu and click on the vertical line and the horizontal line

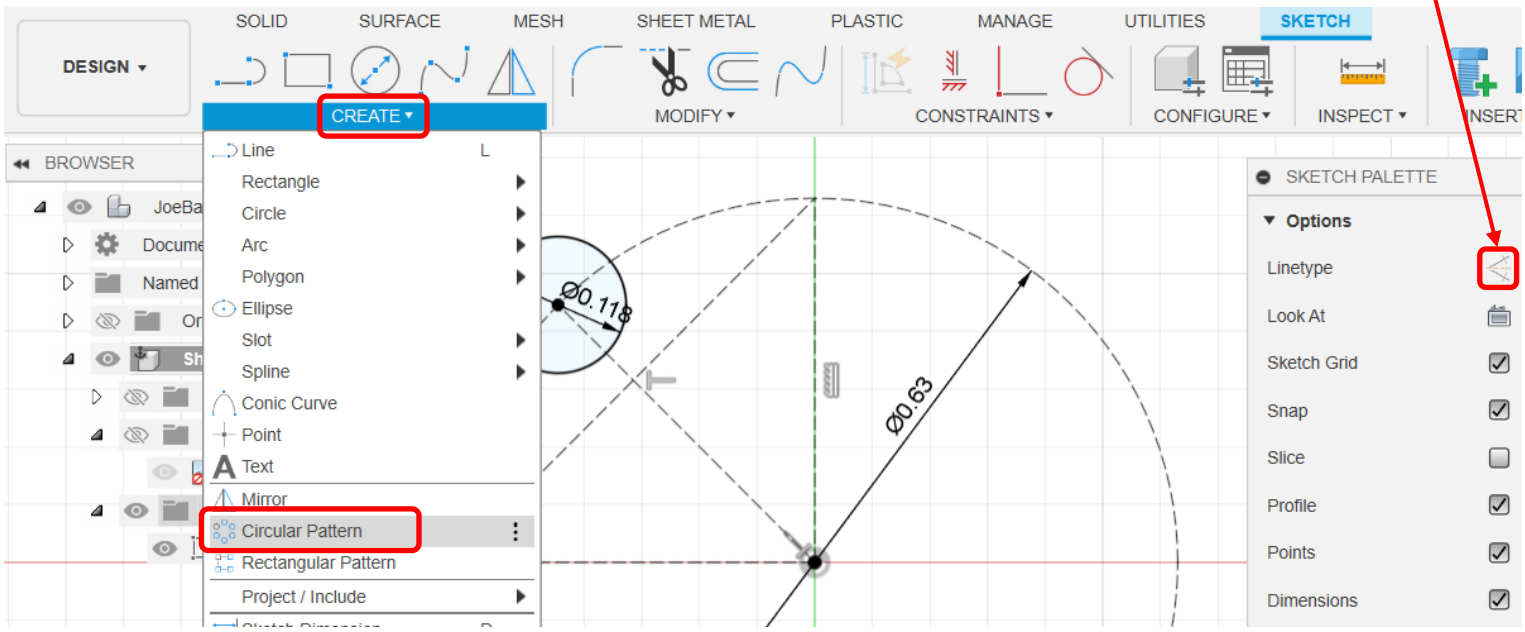
- check if the intersection of the 2 diagonal lines has a Perpendicular symbol . If not select Perpendicular again and click on the 2 diagonal lines.

If the red Lock icon still doesn't show, just continue with the assignment. Sketches don't have to always be constrained. It would be a convenience here to allow changing the circle diameter and maintaining proper hole positions.

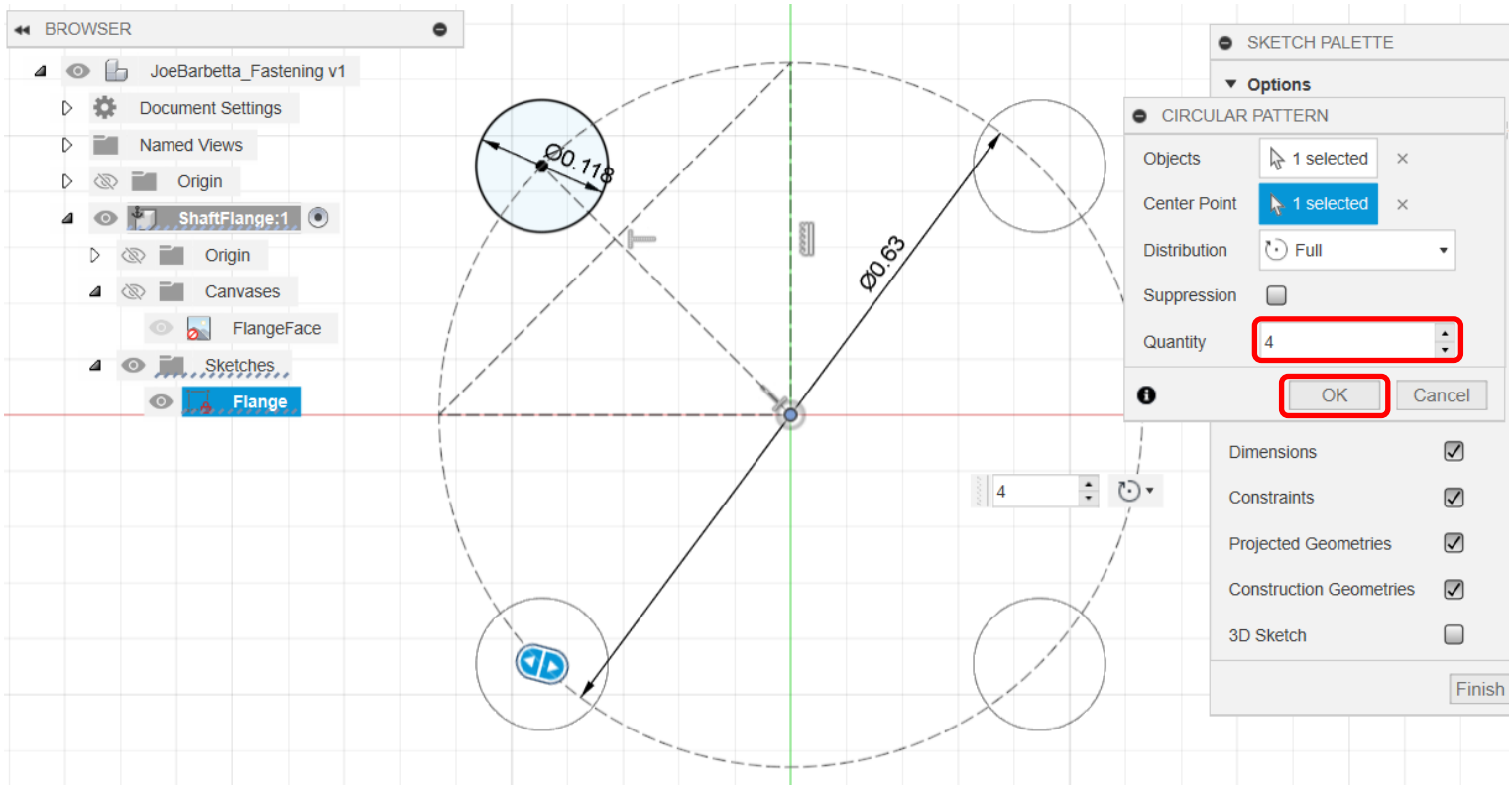
- select the **Center Diameter Circle** tool and click on the **intersection of the diagonal line and the circle**
- enter **3mm** and press the **Enter key**



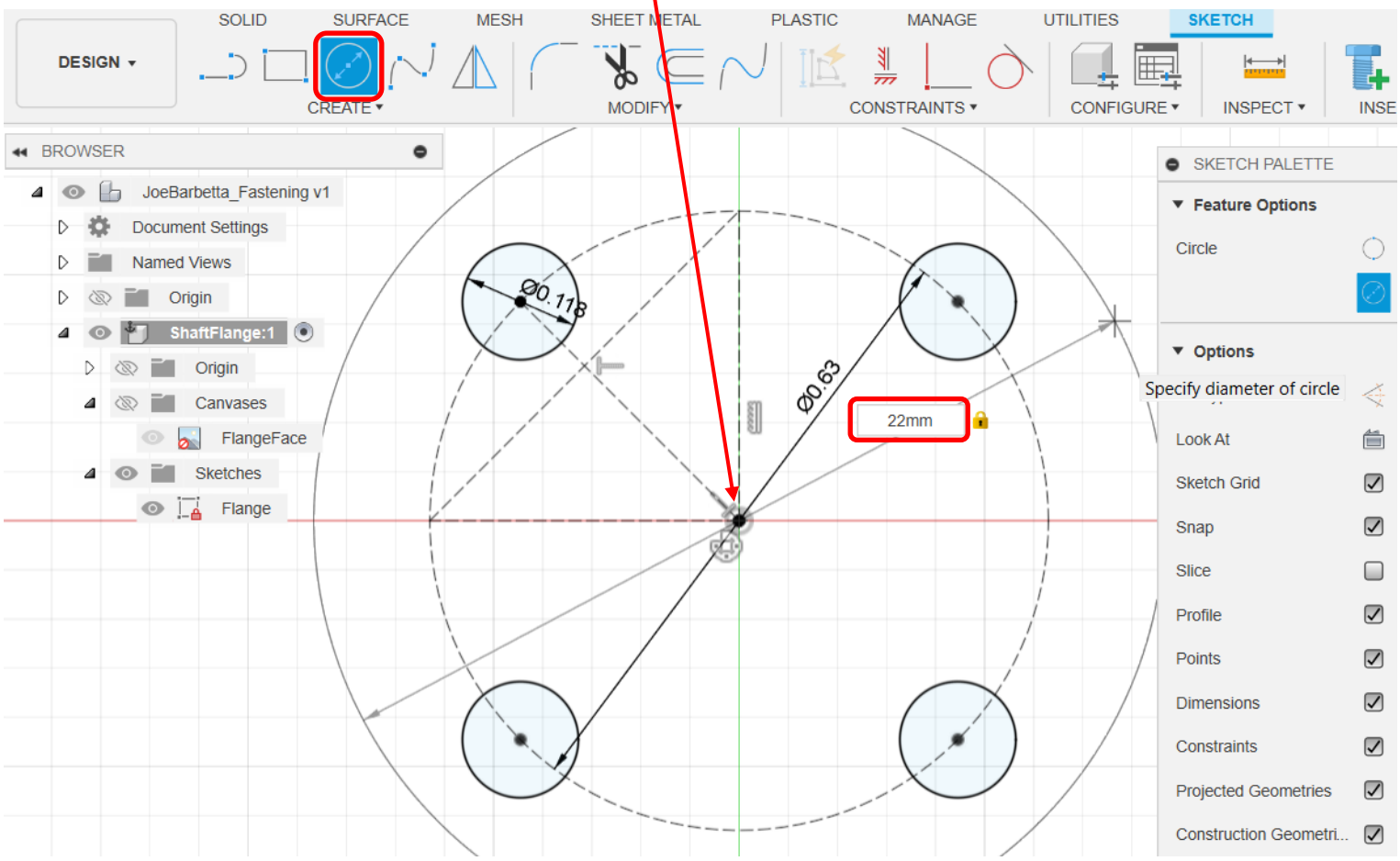
- the circle just created should have a **solid line** as opposed to the dashed line. If it is not solid, ensure that the **Construction** icon is not highlighted blue.
- note that the 3mm was converted to 0.118 as shown on the small circle that was just created
- from the **CREATE** menu select **Cicular Pattern**



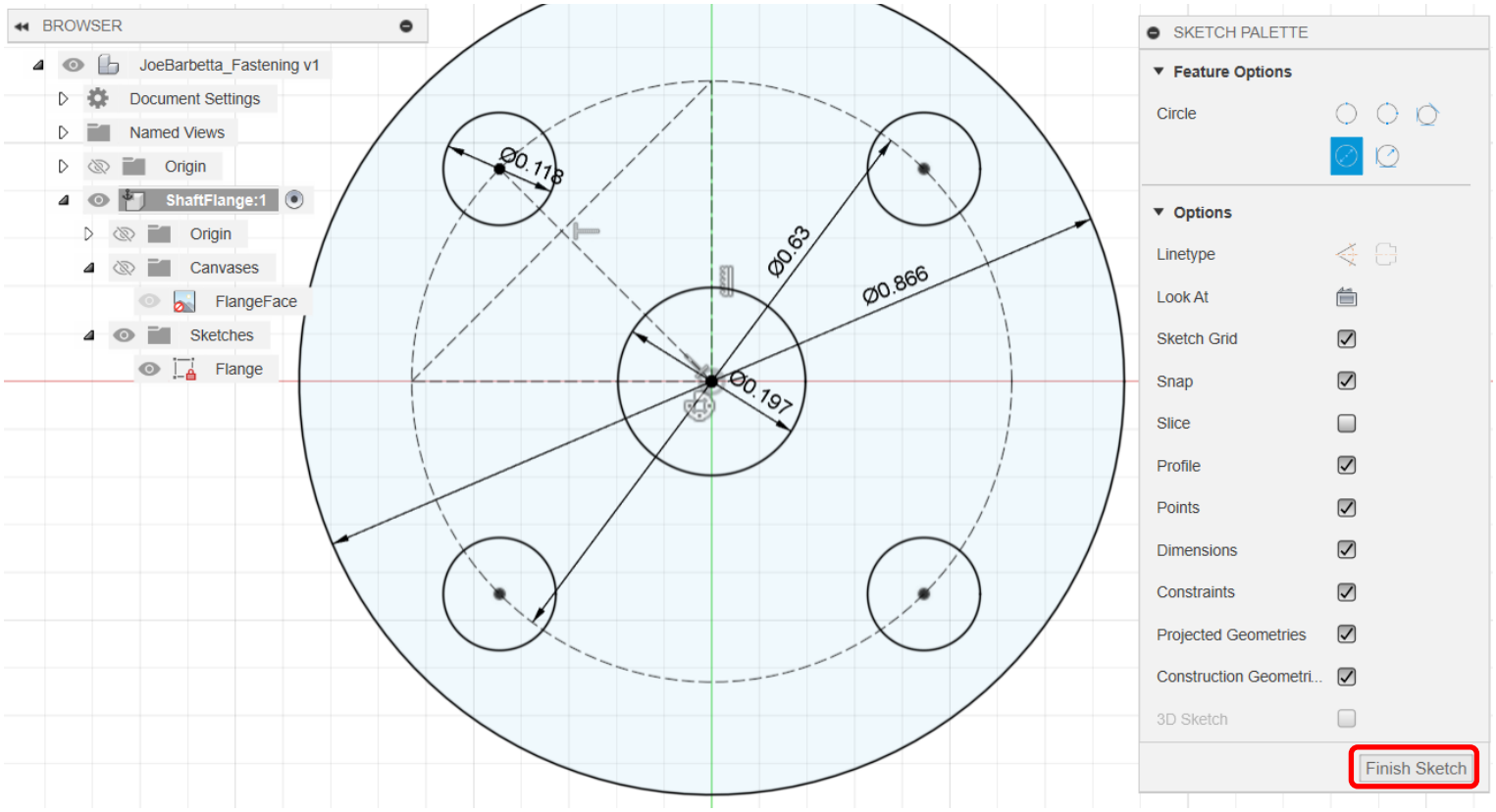
- click on the **small circle** just created and then the **Origin**
- enter **4** for **Quantity** and click **OK**



- select the **Center Diameter Circle** tool and click on the **Origin**
- extend the circle outward and enter a value of **22mm**

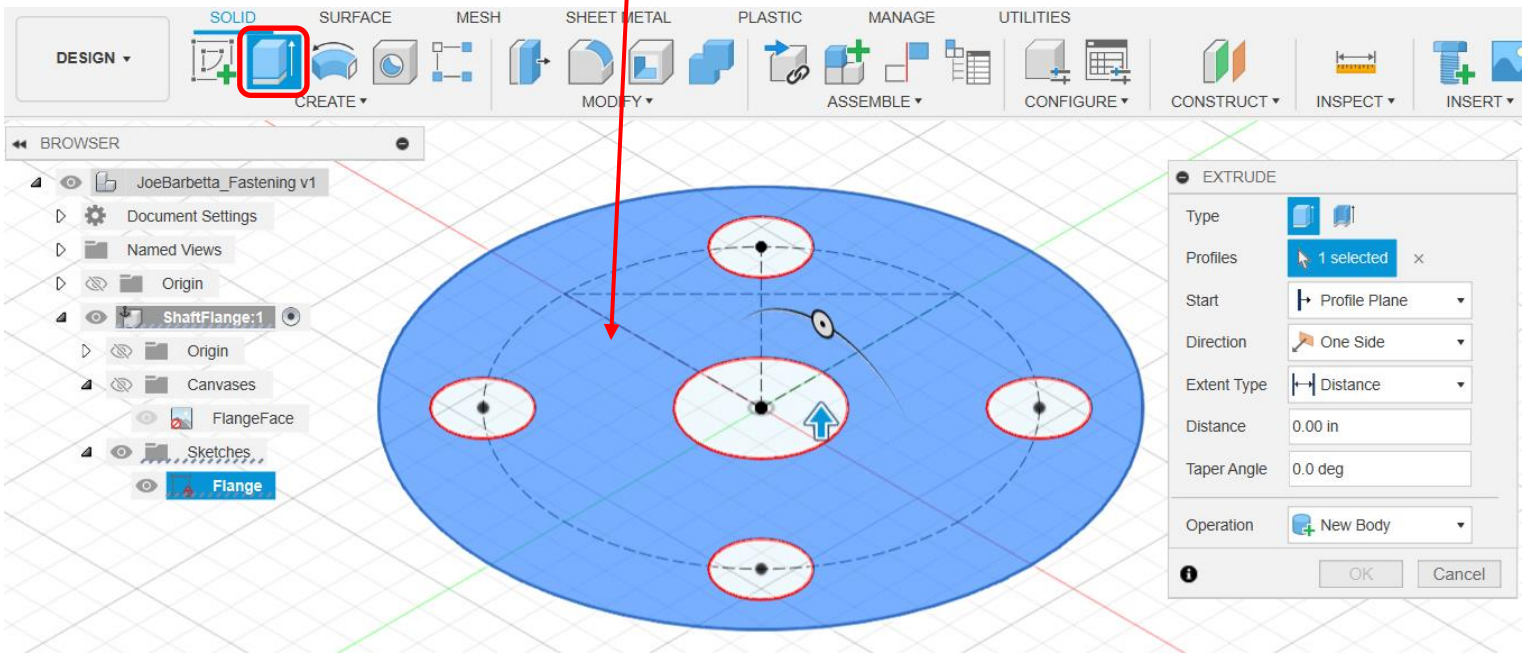


- yell **"This looks neat!"** and click **Finish Sketch**

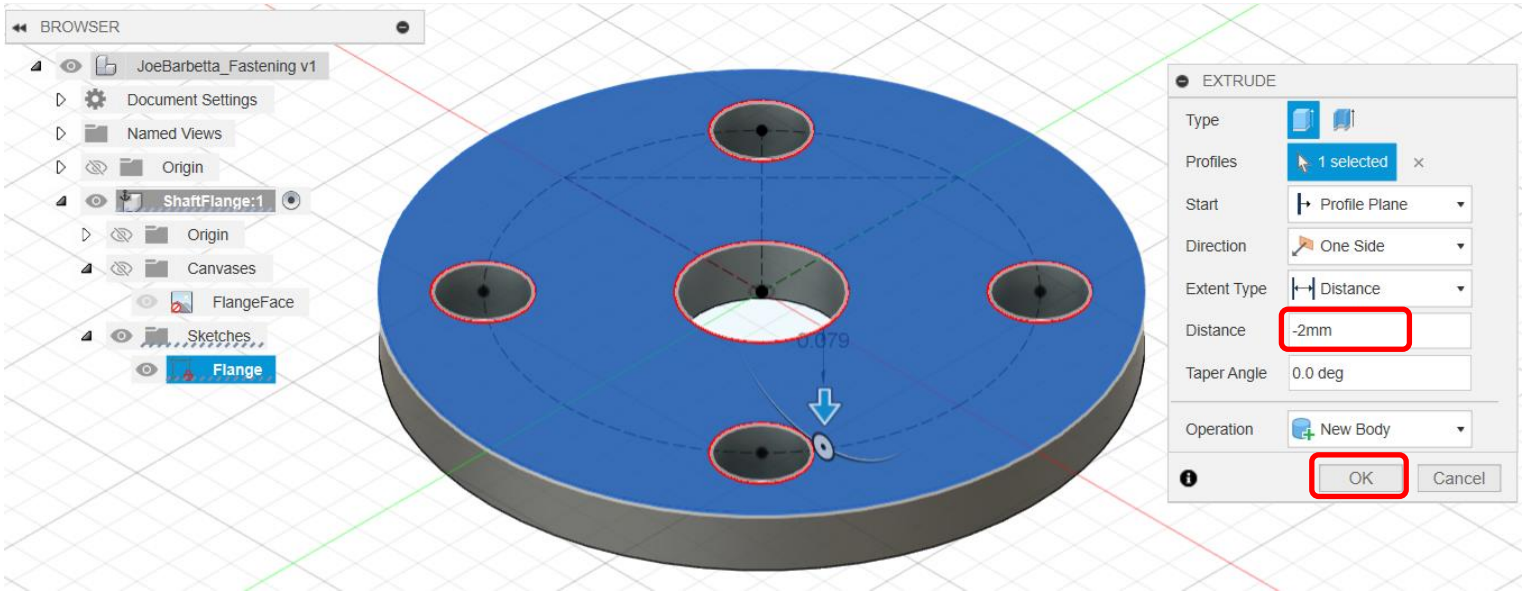


Extruding the Flange

- select the **Extrude** tool and click on the **region between the small circles** to turn the region blue

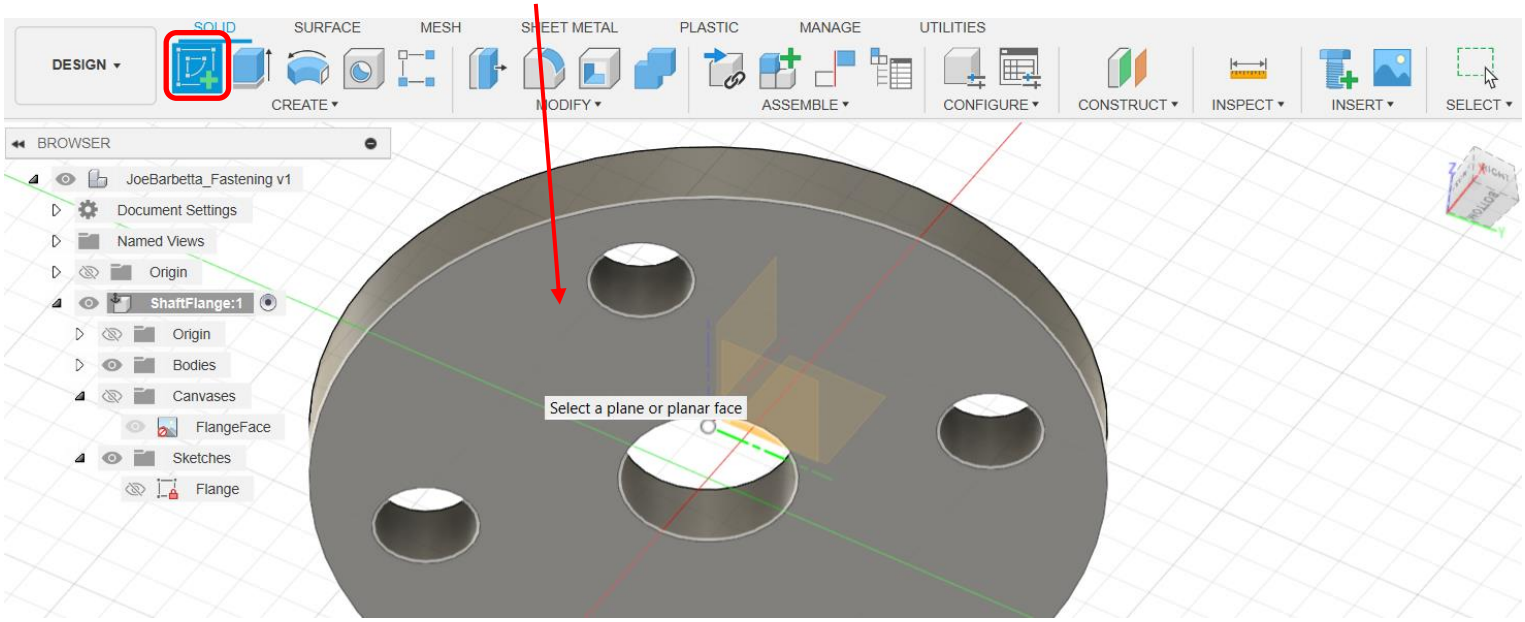


- enter -**2mm** (note the minus sign) for **Distance** and click **OK**

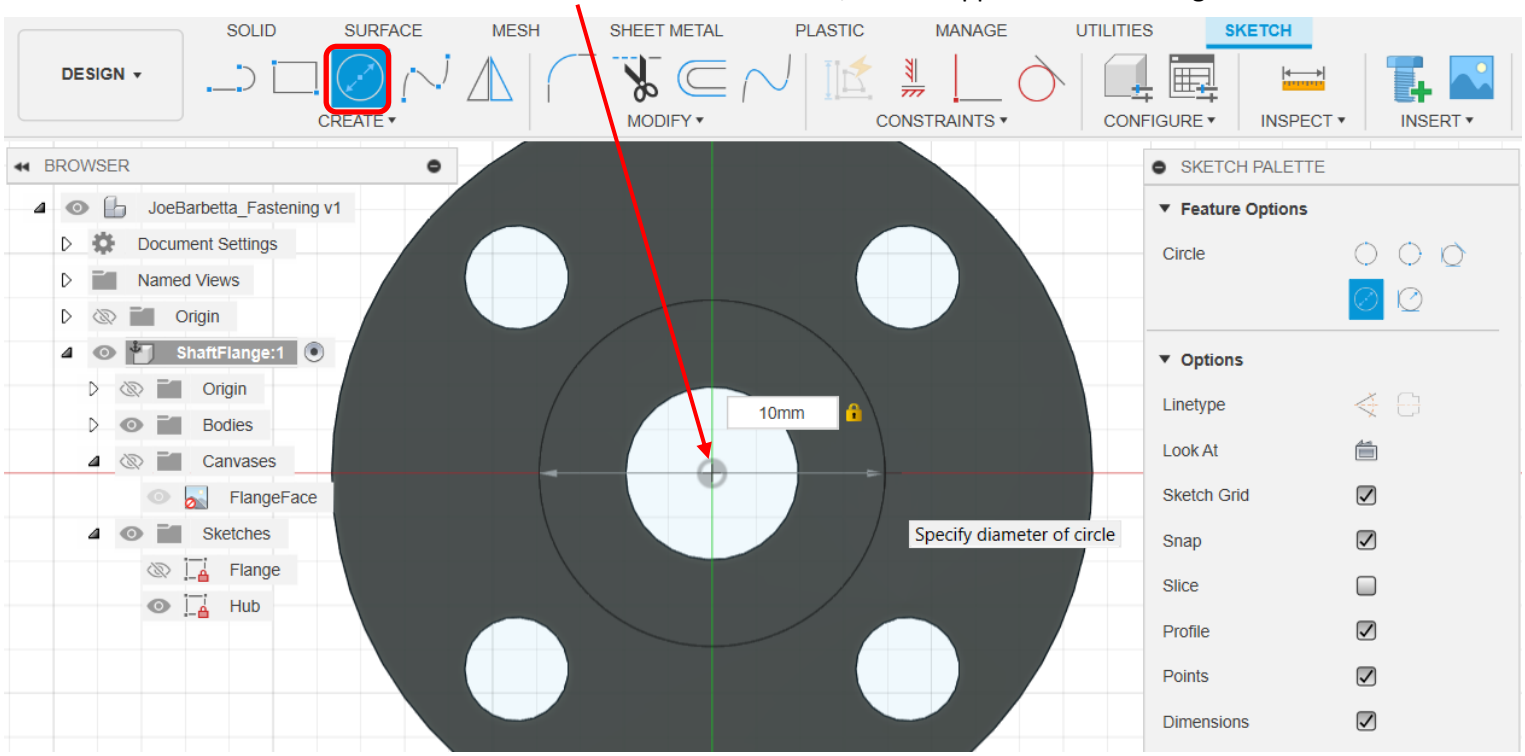


Creating the Hub

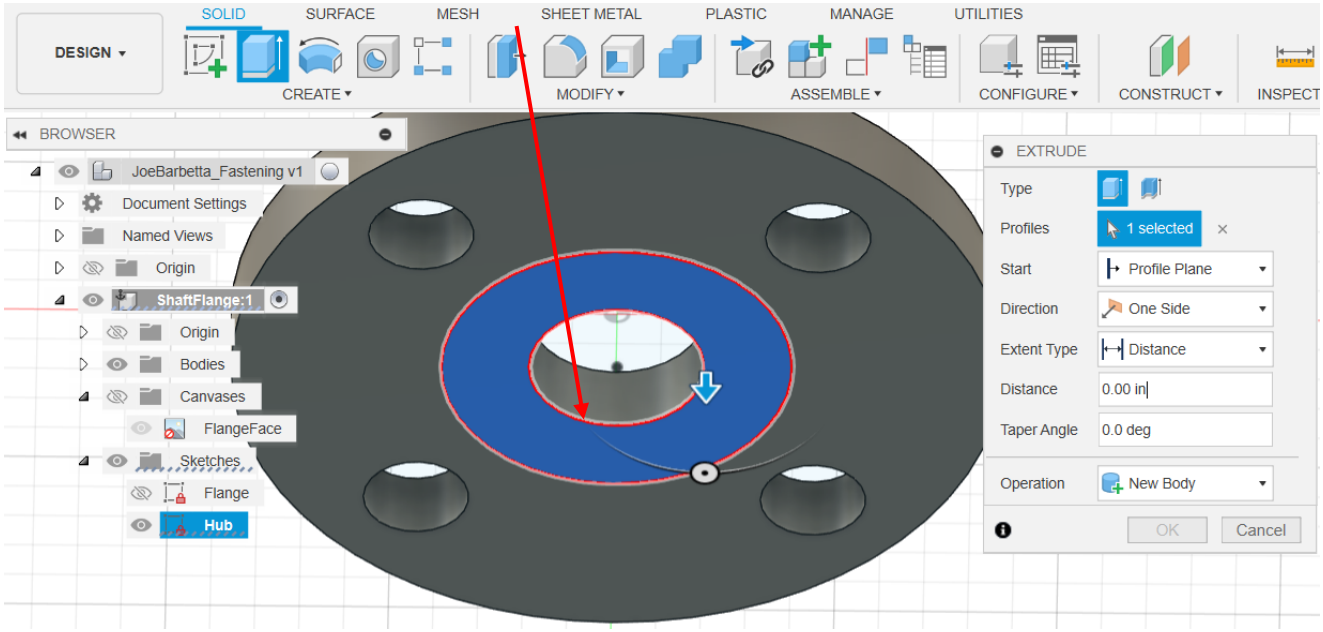
- turn the **View Cube** to access the **underside** of the flange
- select **Create Sketch** and click on the **underside face**



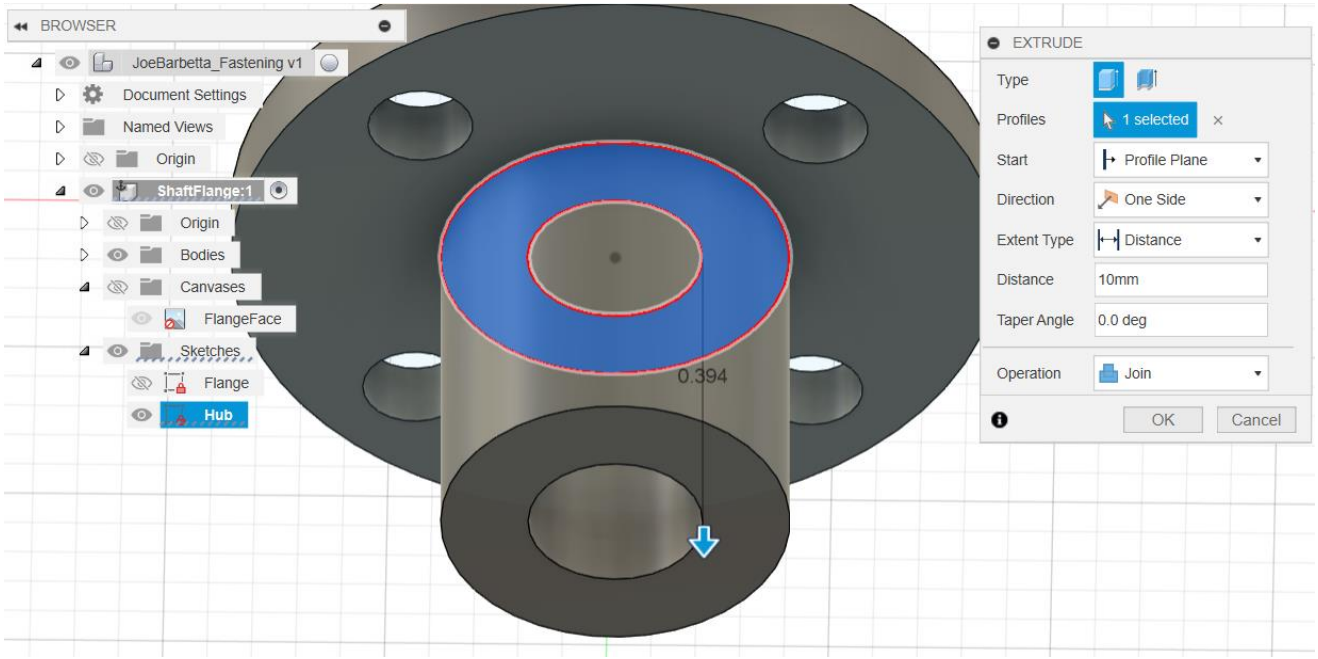
- create a **10mm** circle with its center at the **center of the middle hole**, which happens to be the Origin in this case

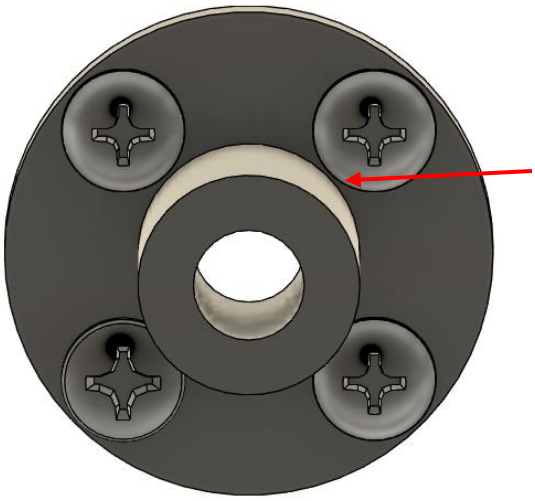


- select the **Extrude** tool and click on the region defined by the last circle



- enter **10mm** for **Distance** and click **OK**





- yell **“Did I really need to create this part?”**

This is how this part will look later on when the screw heads are inserted.

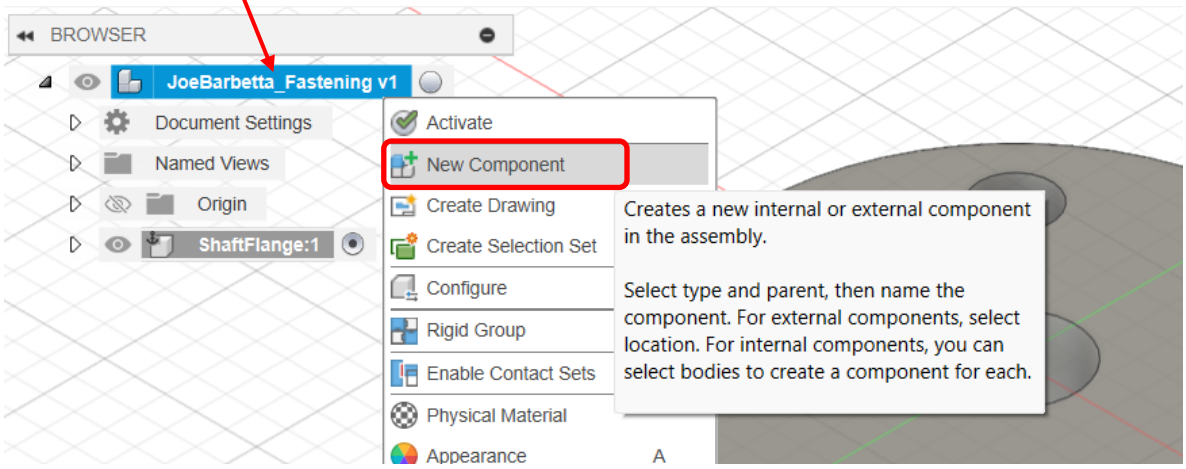
We can see that there is ***clearance*** between the hub and the screw heads. If we were to see ***interference***, where the screw heads would insersect the hub and thus not fit, we would need to chose a screw with a smaller head.

These are common ***pan head*** screws with a thread size of 4-40. This screw was chosen because it fits nicely in the 3mm (0.118”) holes. We could try to a 4-40 screw with a smaller head or choose a slightly longer flat head screw, which would have a smaller head diameter. Ideally, the hole bottoms would be countersunk, but it would work without countersinking the holes. If there is still no room, we could consider a #3 screw.

Creating the Mount Component

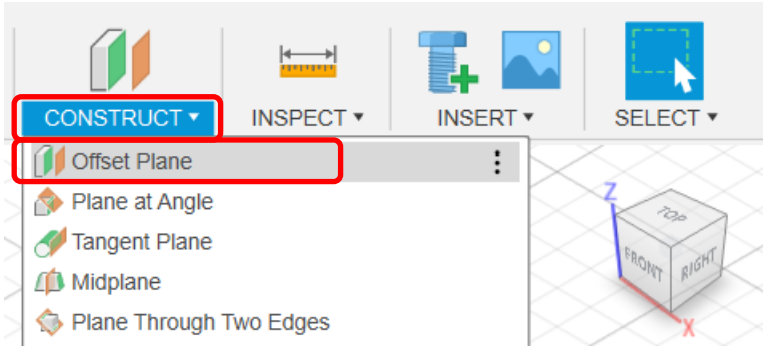
This will be our 3D printed part.

- click on the **project name** and select **New Component**
- name it using your **first name and last initial followed by "_Mount"**, e.g. **JoeB_Mount**

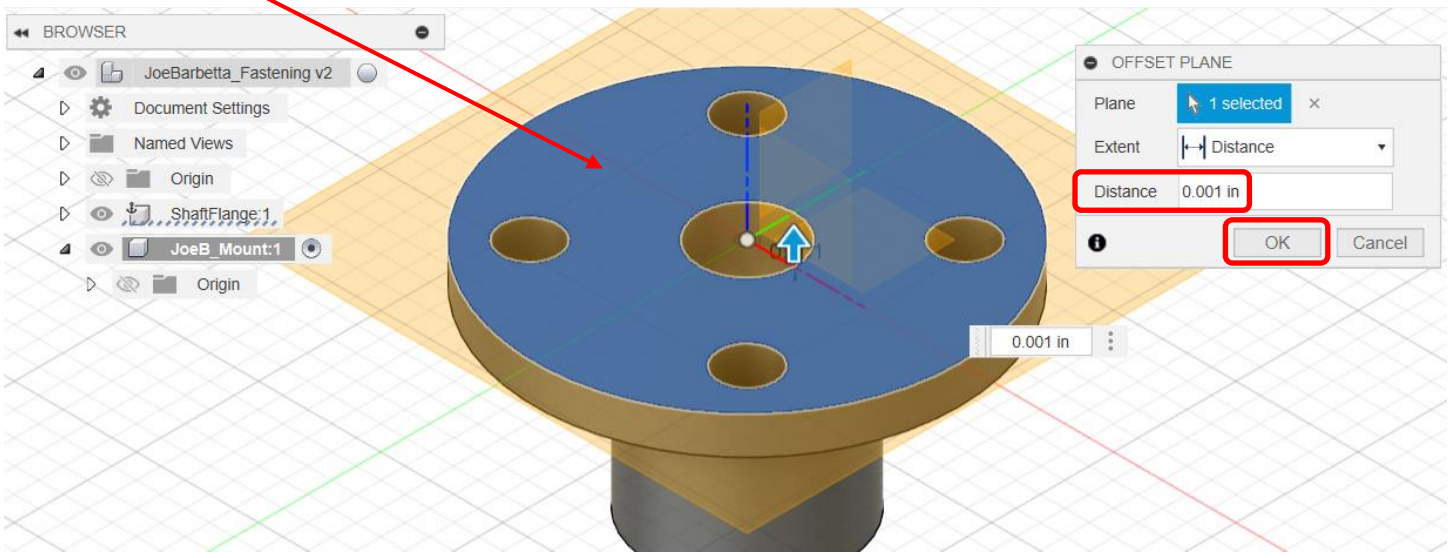


Creating an Offset Plane

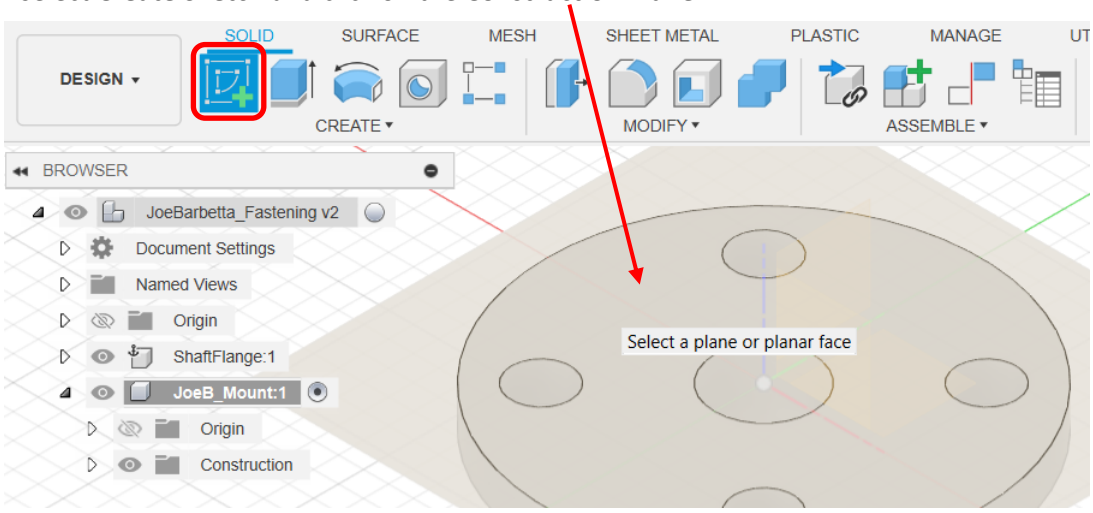
- from the **CONSTRUCT** menu select **Offset Plane**



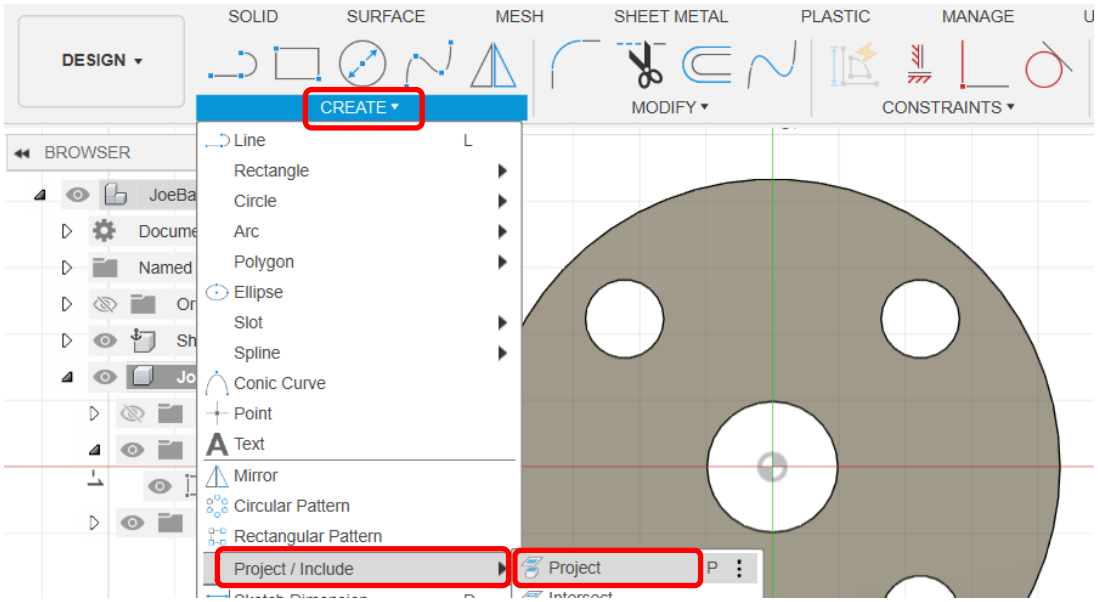
- click on the **face of the Flange**, enter **0.001** for **Distance**, and click **OK**



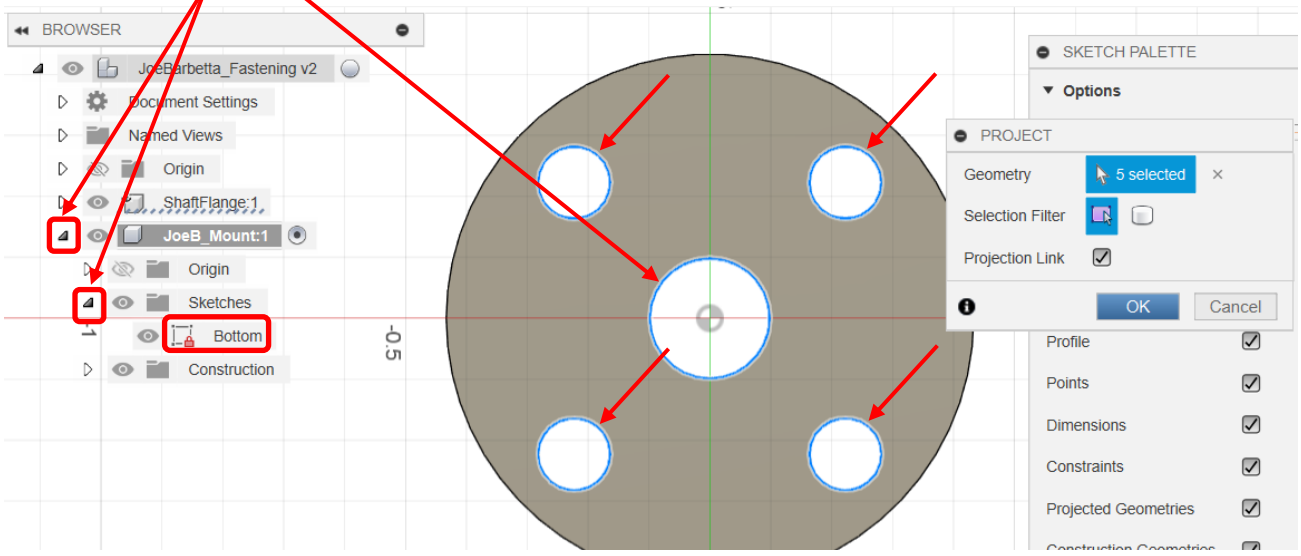
- select **Create Sketch** and click on the **Construction Plane**



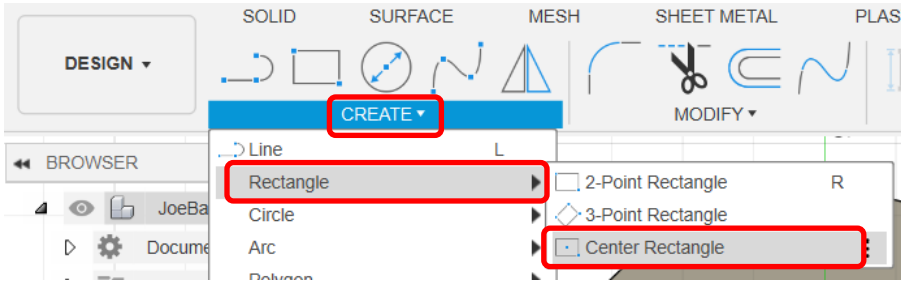
- from the **CREATE** menu select **Project / Include** and **Project**



- click on the **center circle** and the **4 smaller circles** and click **OK**
- click on the **arrows** for the new component and the arrows for Sketches and rename the Sketch to **Bottom**

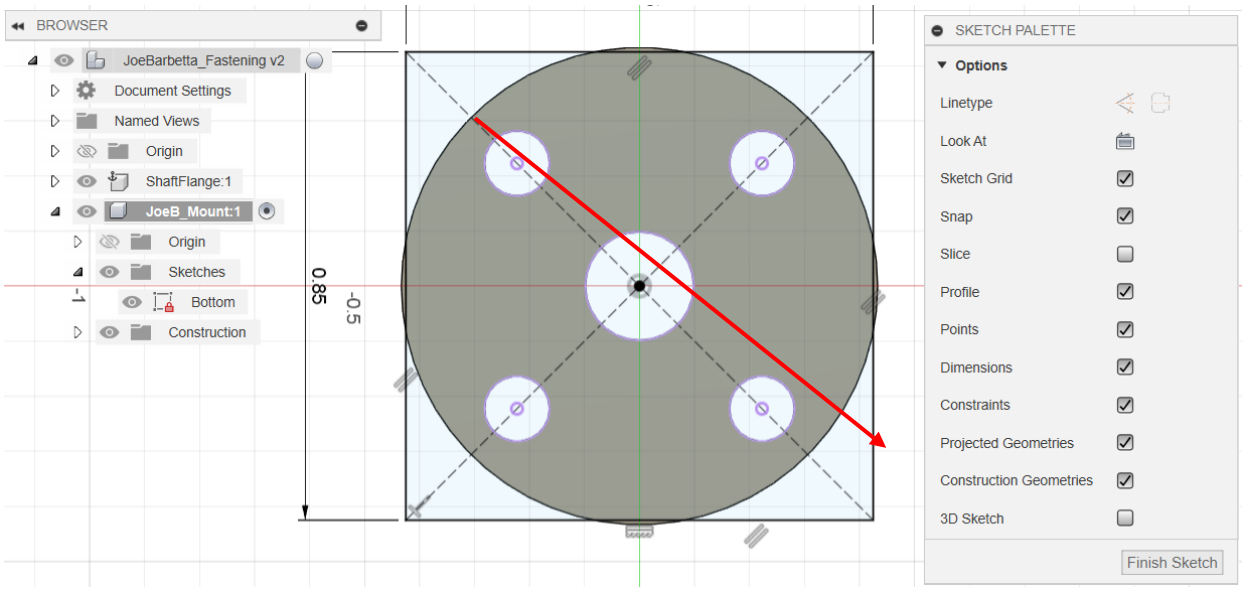


- from the **CREATE** menu select **Ractangle** and **Center Rectangle**

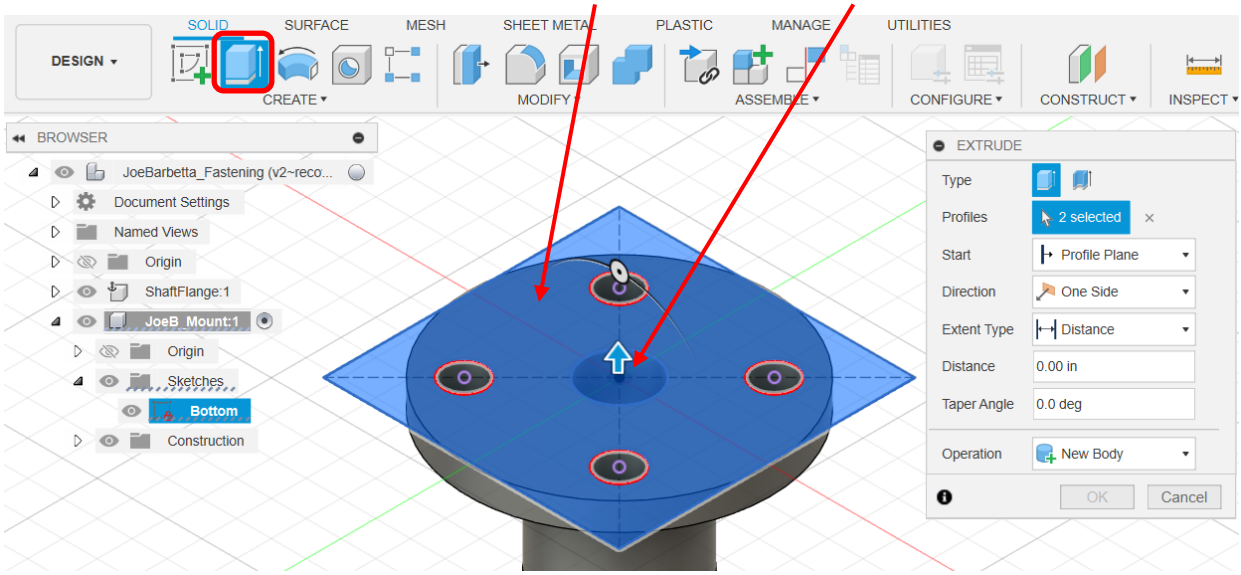


- click on the **center of the middle circle**, extend the rectangle outward, type **0.85**, press the **Tab key**, type **0.85**, and press the **Enter key**. The Tab key switches between the hieght and width values of the rectangle. This value is somewhat arbitrary and chosen because it just about covers the flange.
- **click Finish Sketch**

Note that in this case the Shaft Flange was created by centering it on the Origin. For other designs, the part being attached to may not be centered on the Origin. In that case the projecttion created for the center hole whould still allow centering the mount on the part.



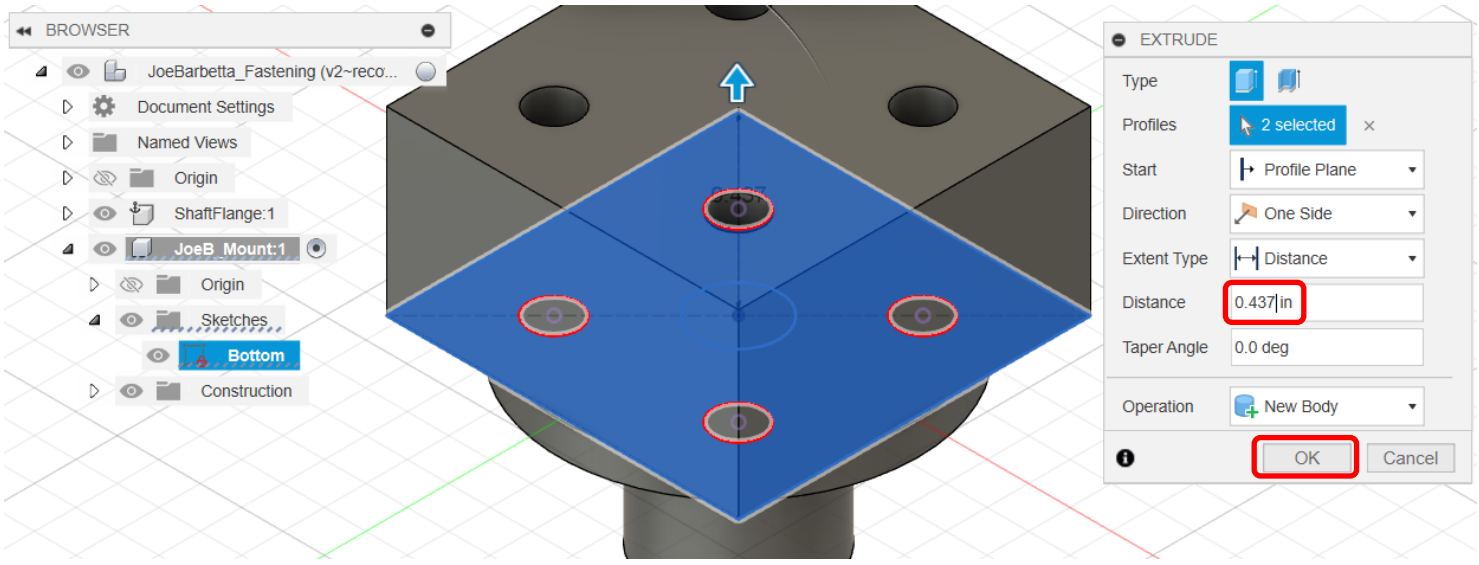
- click on the **Home icon** at the **View Cube** and zoom to achieve a view similar to that below
- select the **Extrude** tool and click on the **rectangular region and the center circle** to cause them to turn blue



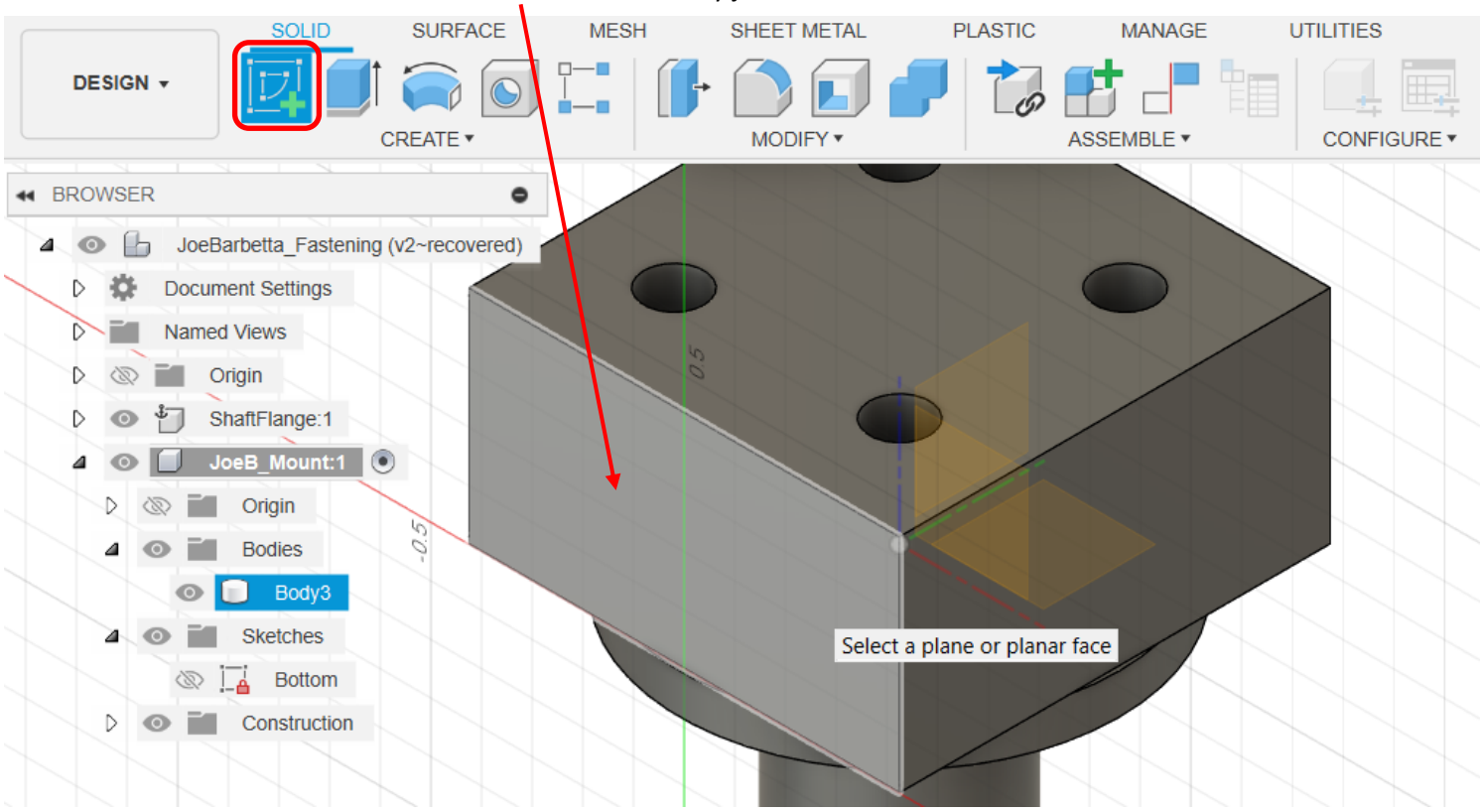
A value of 0.437 will be used for the height of this block. The determination of this value was started by considering that about an 1/8” of material above the 1/4” rod would be adequate to secure the rod and that the material under the rod could be about 1/16” and thus 0.437” (0.125 + 0.25 + 0.062) could be considered for the height of the block.

A round head screw that would pass through the 2mm (0.079) flange and the 0.437 block would require a length of 0.516. Good luck finding that. How about a 1/2” screw?

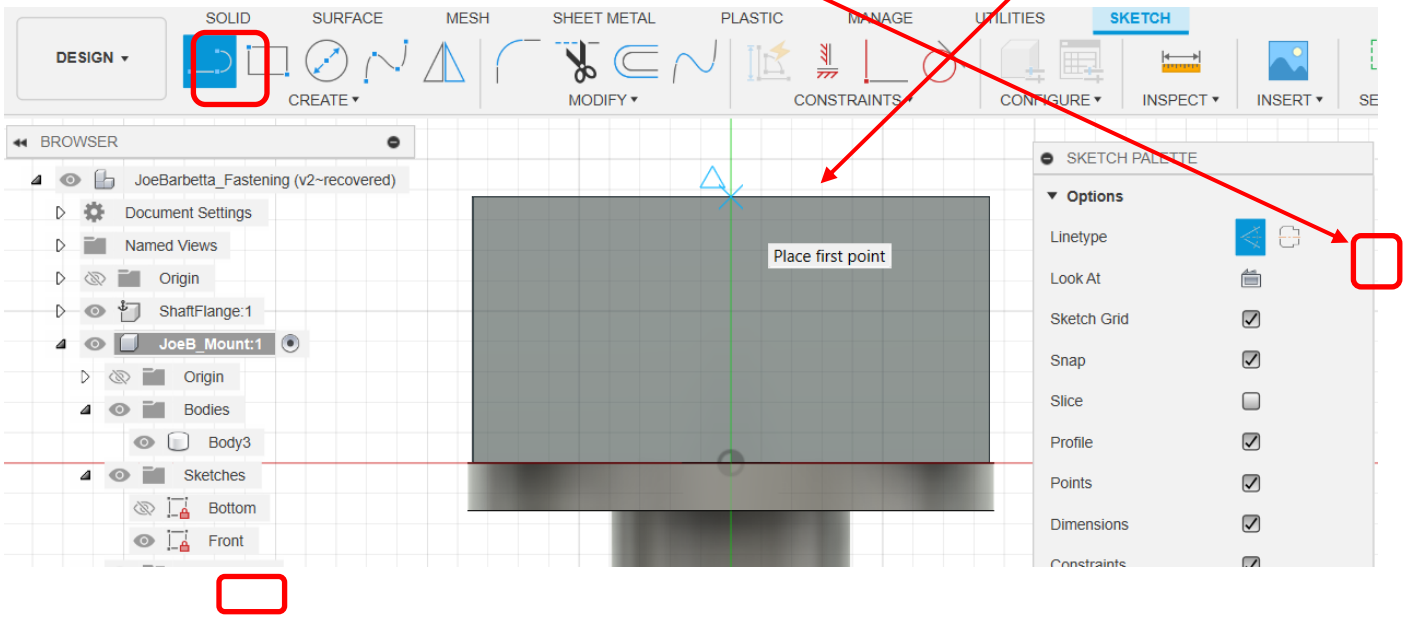
- enter **0.437** for **Distance** and click **OK**



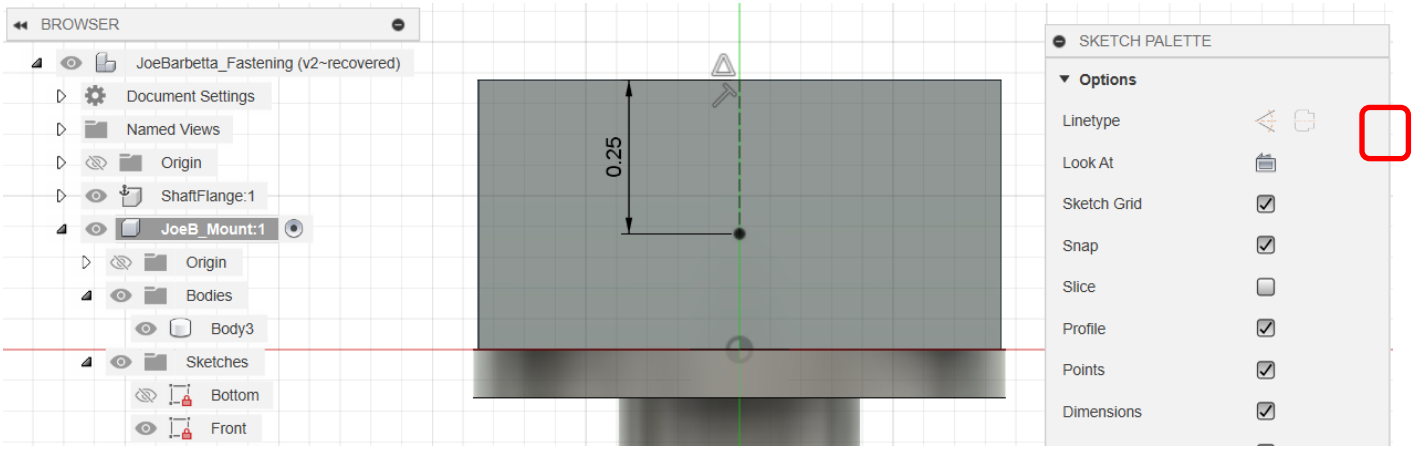
- select **Create Sketch** and click on the **Front face** of the body just created



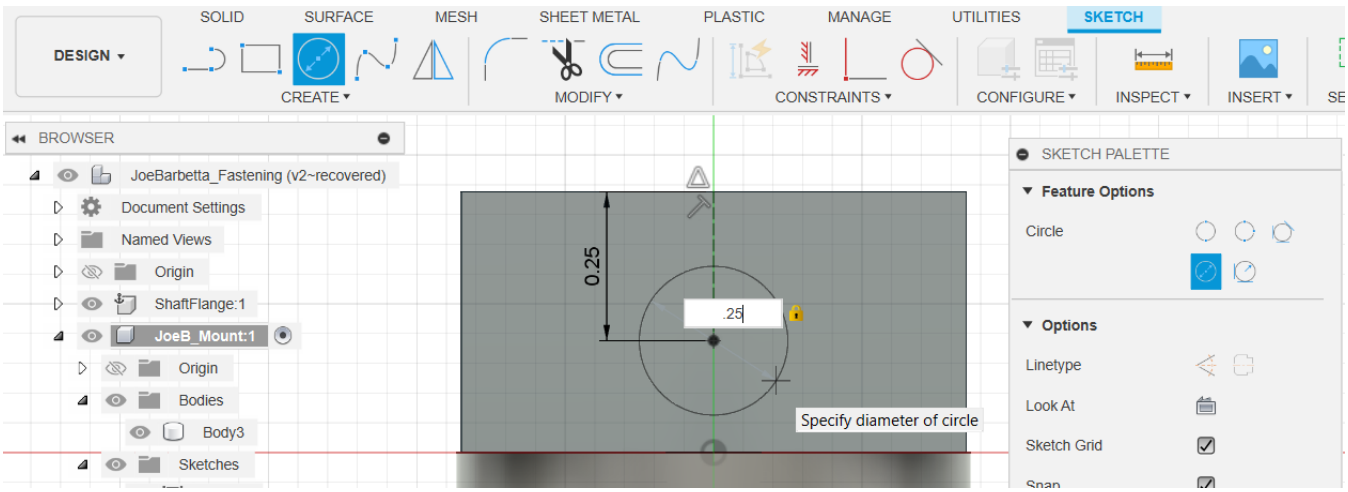
- rename the **Sketch** to **Front**
- click on the **Construction icon** to highlight it blue
- select the **Line** tool and move the mouse over the **center of the top edge** until the **blue triangle** shows
- click at this **point** and **extend the line downward**



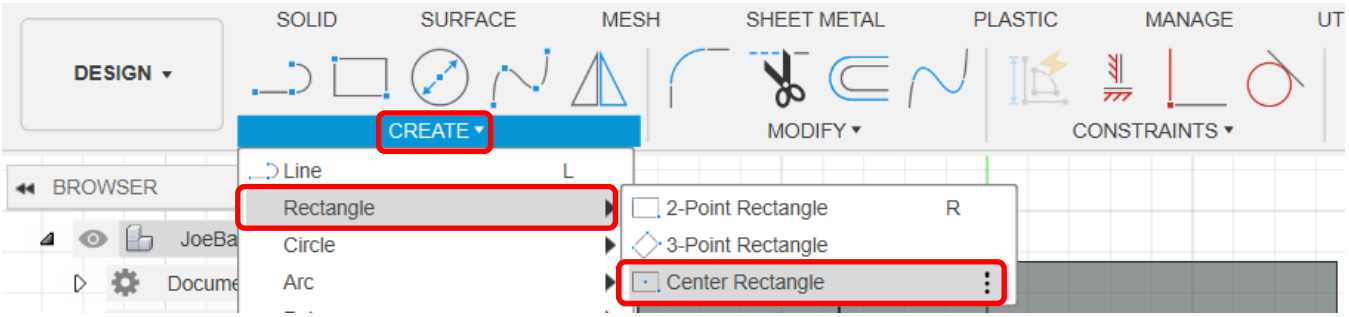
- type **0.25** and press the **Enter key**, which should result in the line shown below
- click on the **Construction icon** again to remove the highlighting



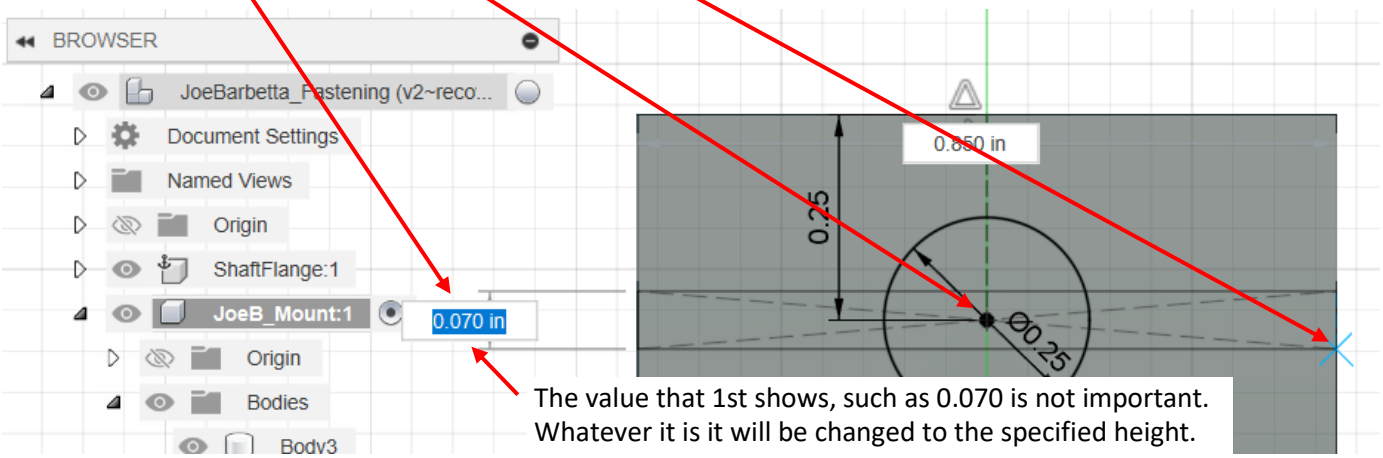
- select the Center Diameter Circle tool and click on the bottom end of the line
- extend the circle outward, type 0.25, and press the Enter key



- from the **CREATE** menu select **Rectangle** and **Center Rectangle**

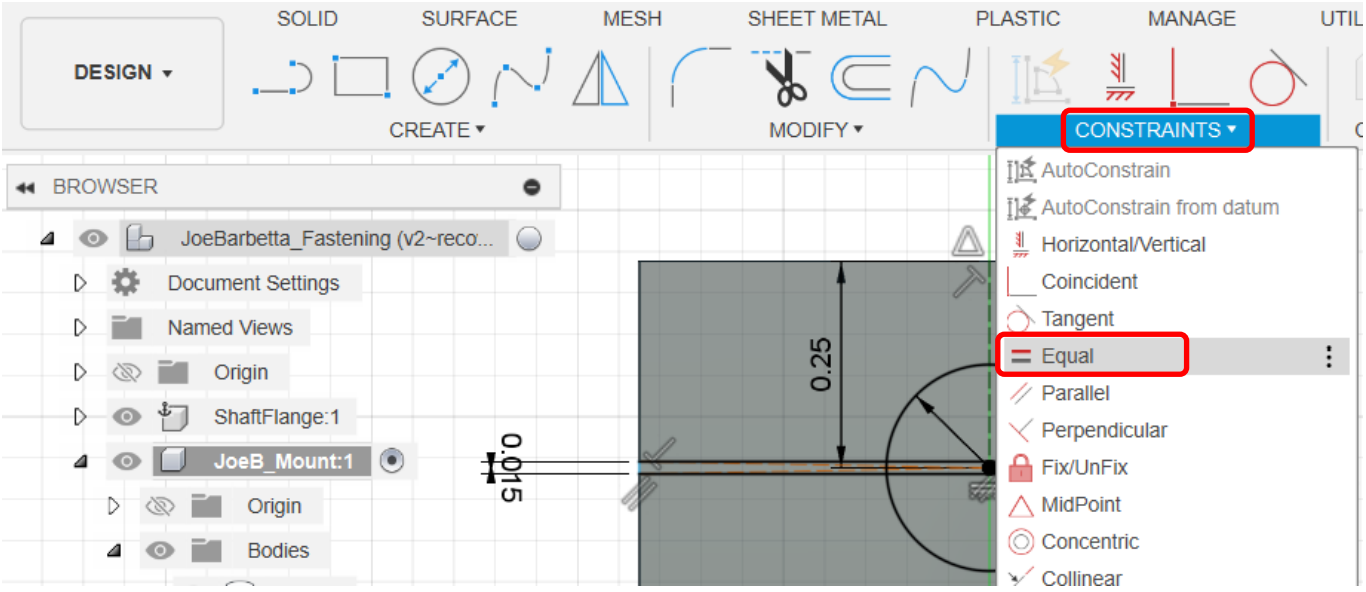


- click on **the center of the circle** and extend the rectangle **to the right and downward** until the right edge of the rectangle is even with the **right edge of the block**. Note the **blue x** that will show.
- enter a value of **0.015** for the rectangle height

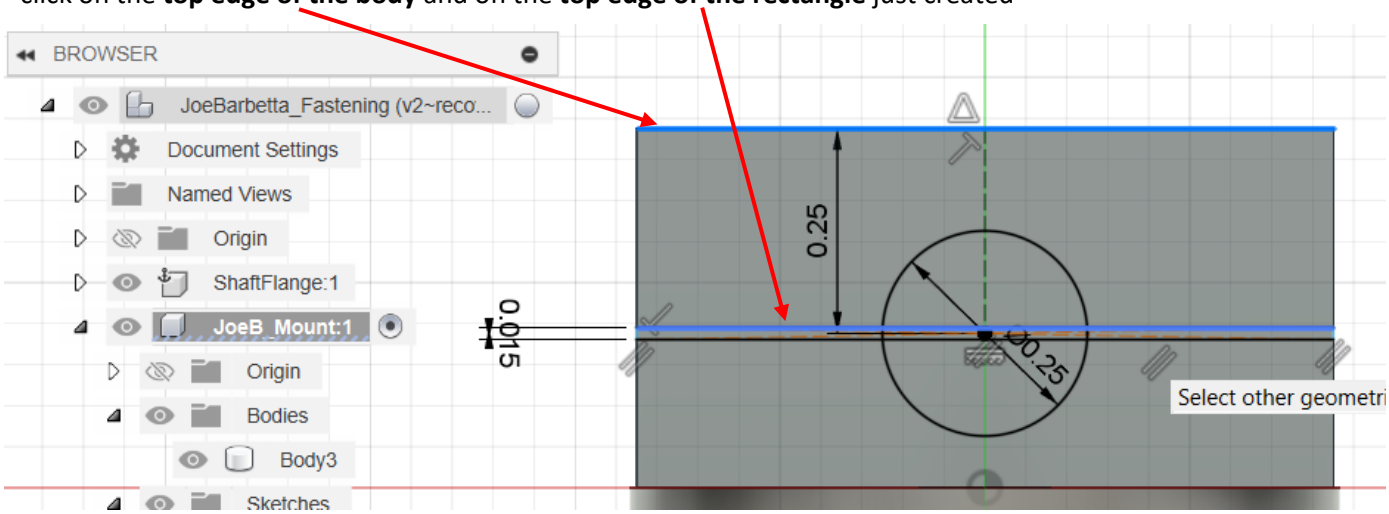


The value that 1st shows, such as 0.070 is not important. Whatever it is it will be changed to the specified height.

- from the **CONSTRAINTS** menu, select **Equal**

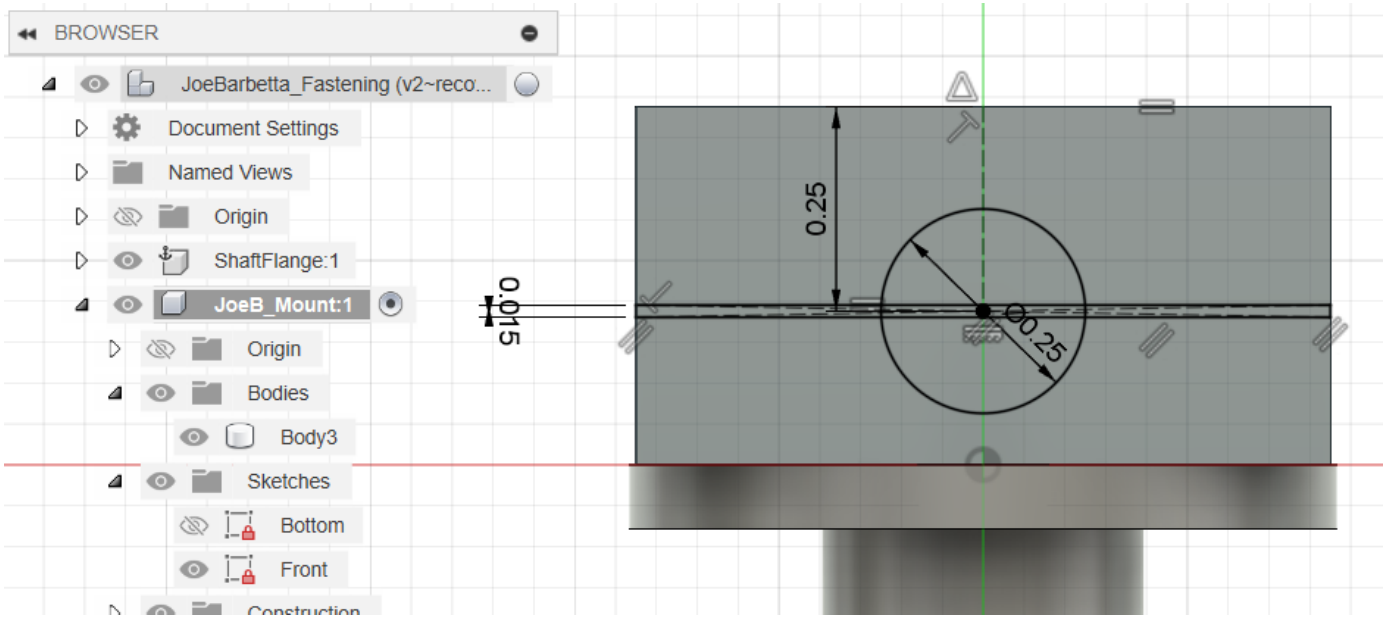


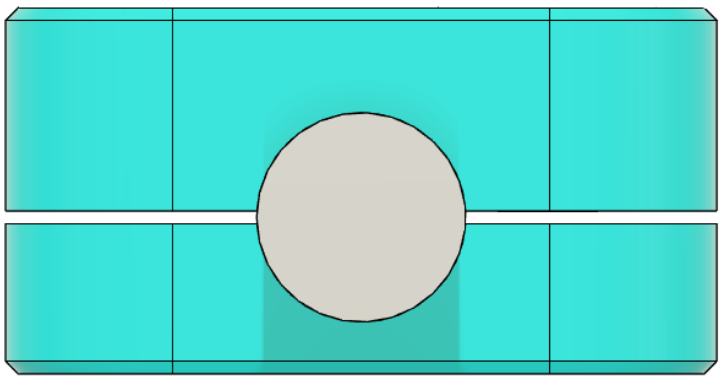
- click on the **top edge of the body** and on the **top edge of the rectangle** just created



The sketch should look like that below. Note that it is OK if the dimension lines and values are in different locations. Note that the **Front Sketch** shows a **small red lock icon**, which indicates that the sktech is fully constrained.

- click **Finish Sketch**

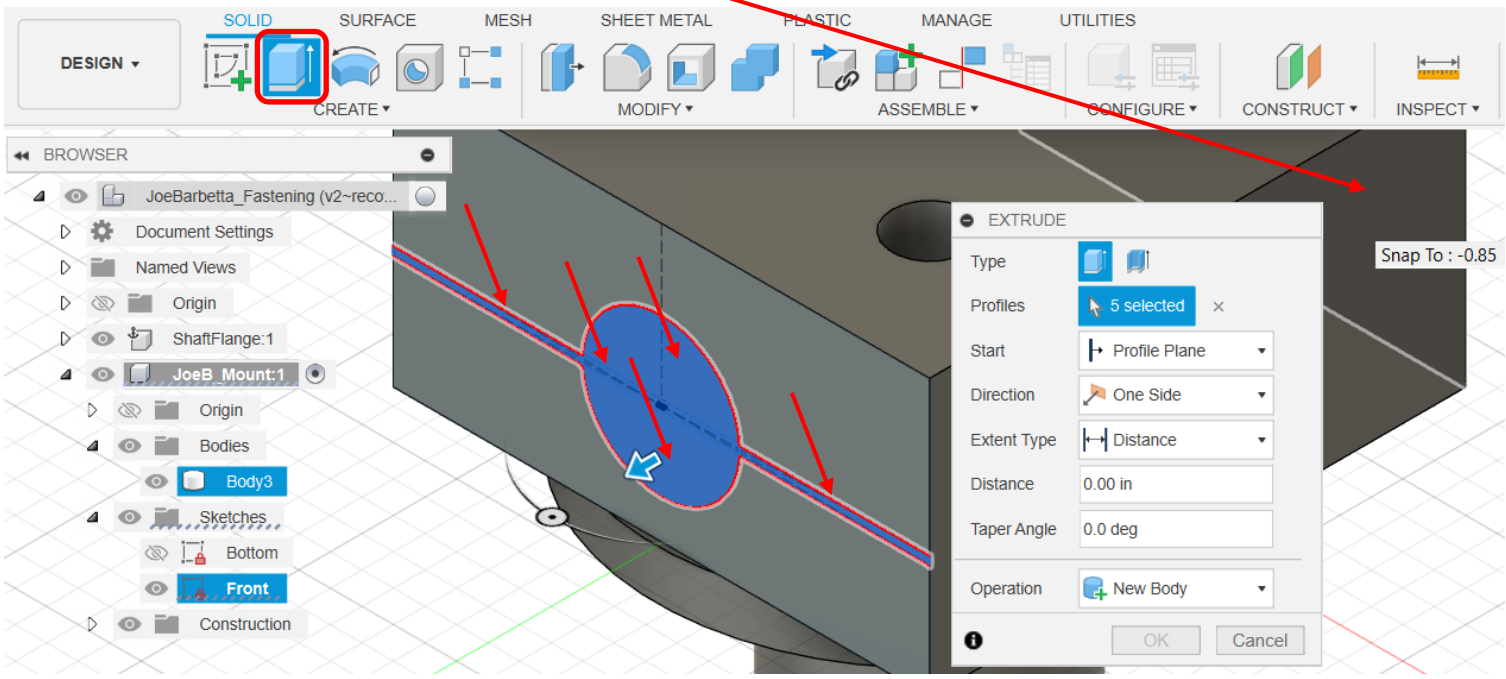




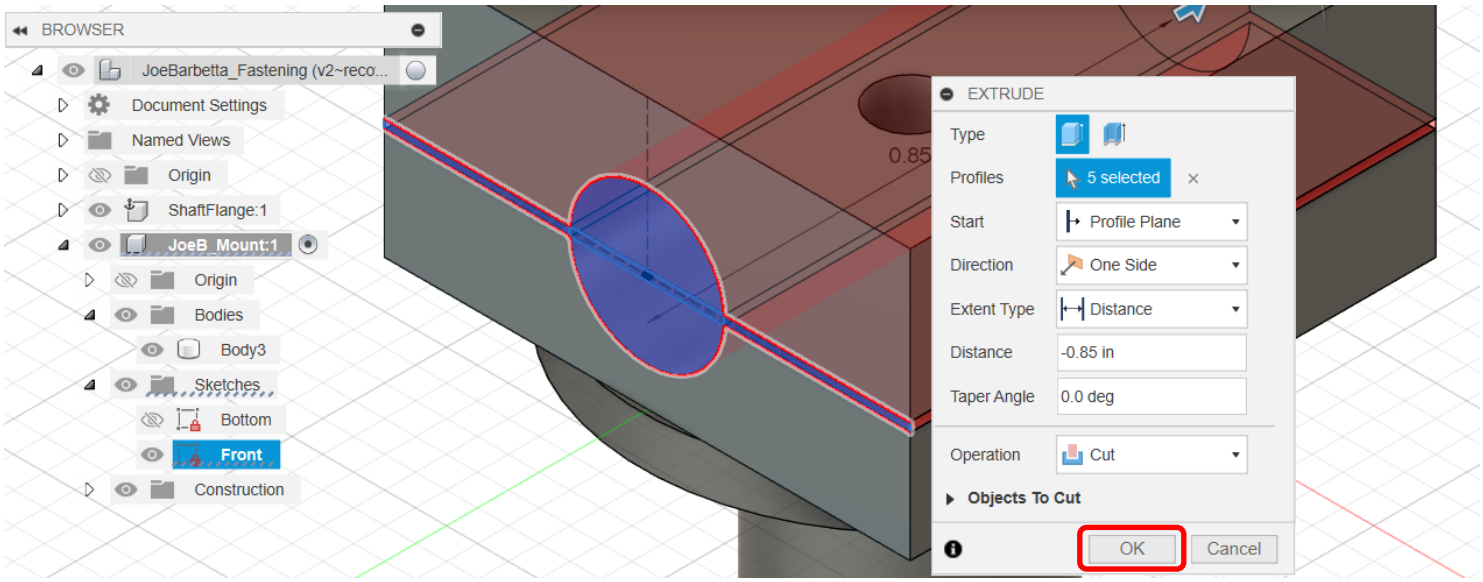
The value of 0.015 defines the space between the top and bottom of the mount.

If there were no space, the rod may be loose when the mount is fully tightend.

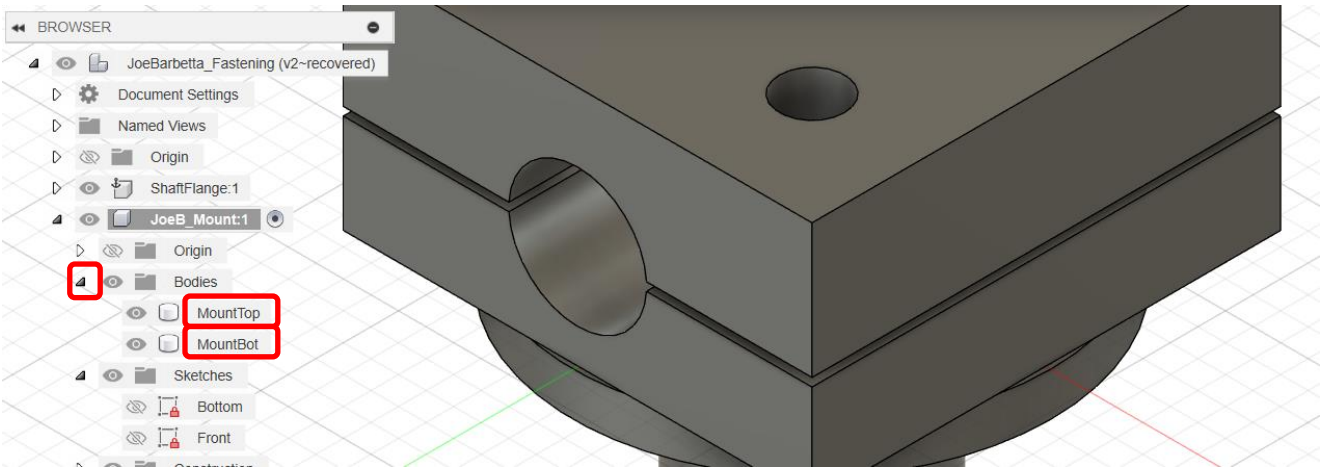
- zoom into the sketch that was just created as shown below
- select the **Extrude** tool
- click on the **thin rectangular area to the left and right of the center circle**
- click on the **thin rectangular area in the circle**
- click on the **top and bottom half of the circle** so that the region as shown below is all highlighted blue
- click on the area shown, which is selecting the **rear face**. Note that is selected through the part.



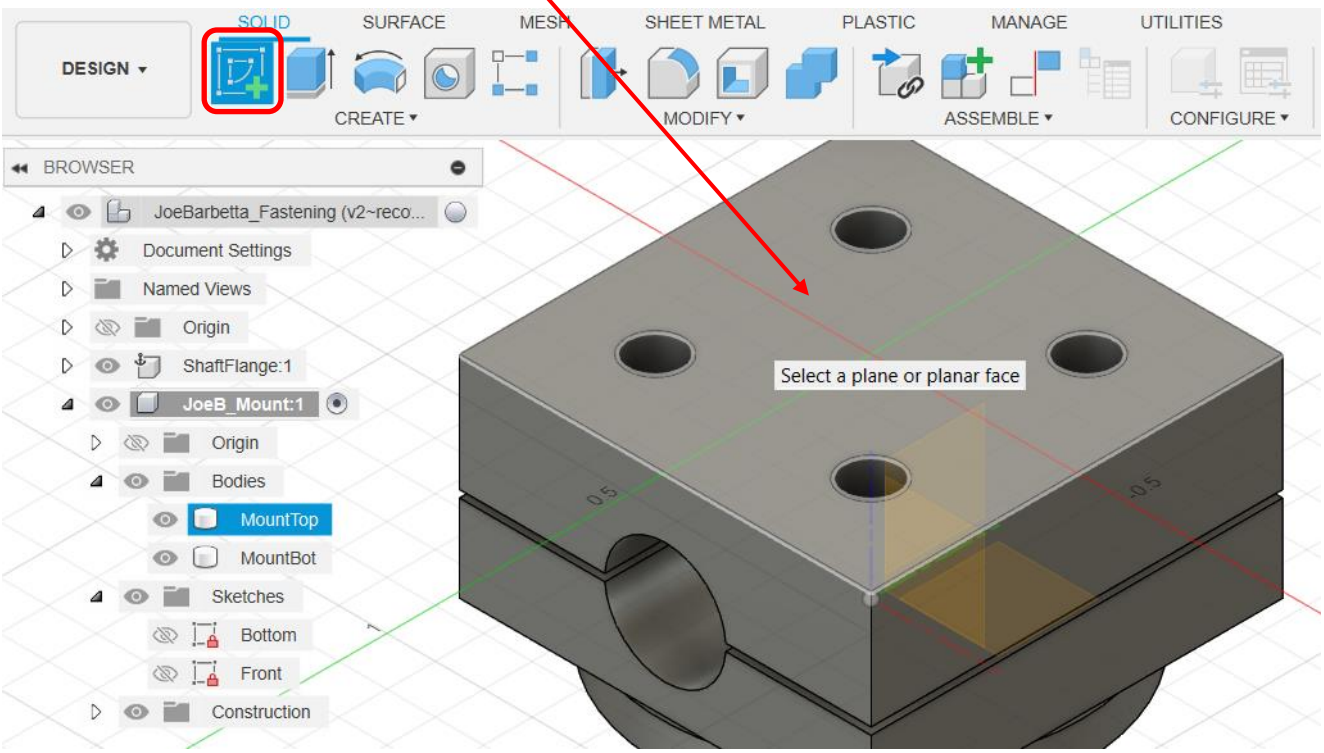
- the extrude operation will indicate the mass to be cut away in red
- click **OK**



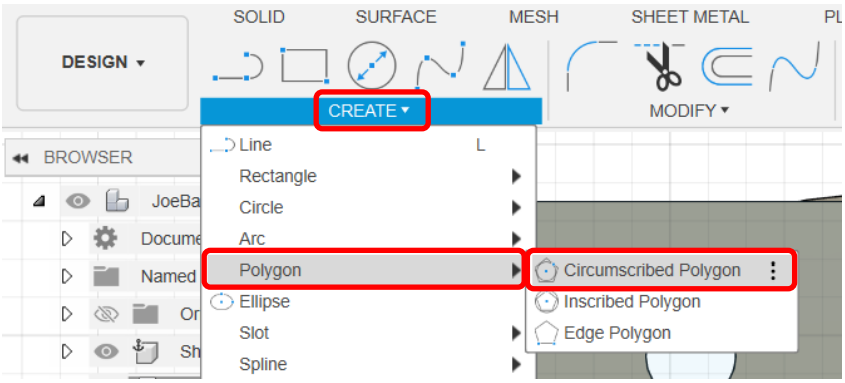
- click on the **arrow** for the **Bodies** folder to open it
- move the mouse over the 2 Sketch names, Body1 and Body2 at first. Rename the Body for the top as **MountTop** and that for the bottom to **MountBot**



- select **Create Sketch** and click on the **top face** of the mount
- rename the sketch to **Top**

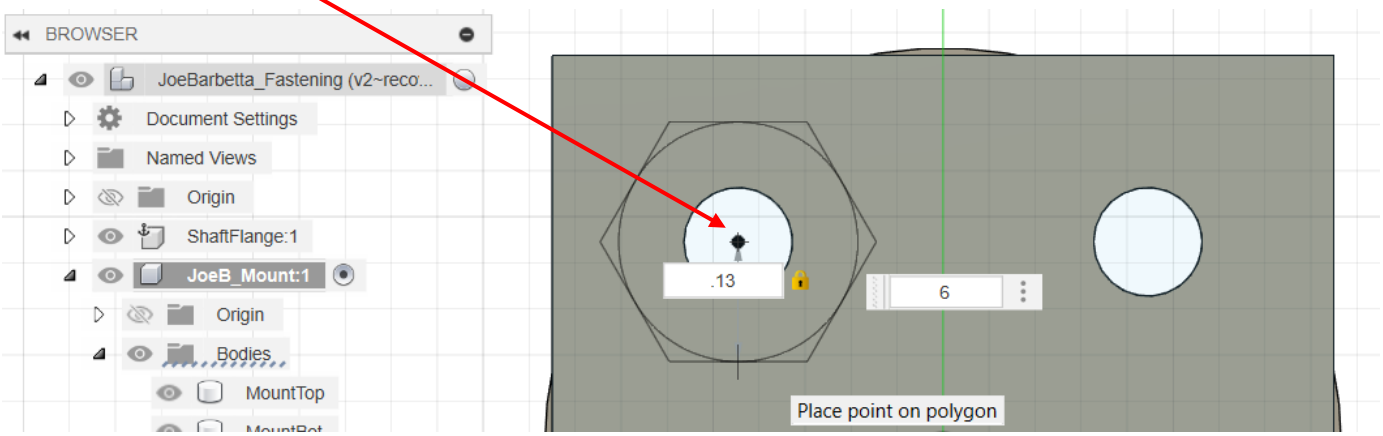


- from the **CREATE** menu select **Polygon** and **Circumscribed Polygon**

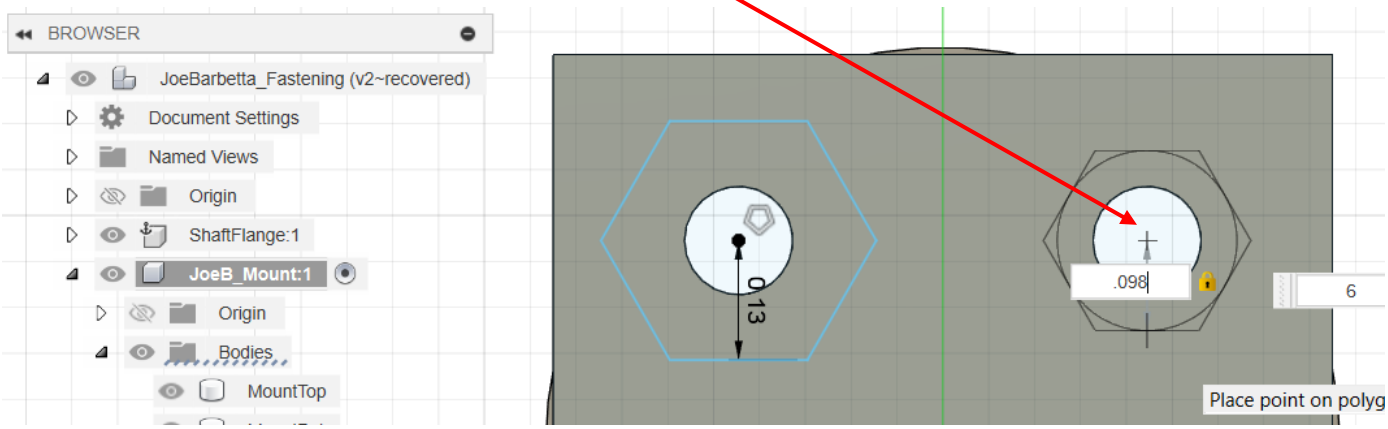


Note that we are implementing four fastening options.
1 – using a machine screw with a regular size hex nut
2 – using a machine screw with a small pattern hex nut
3 – using a machine screw with a threaded insert
4 – using a self threading screw

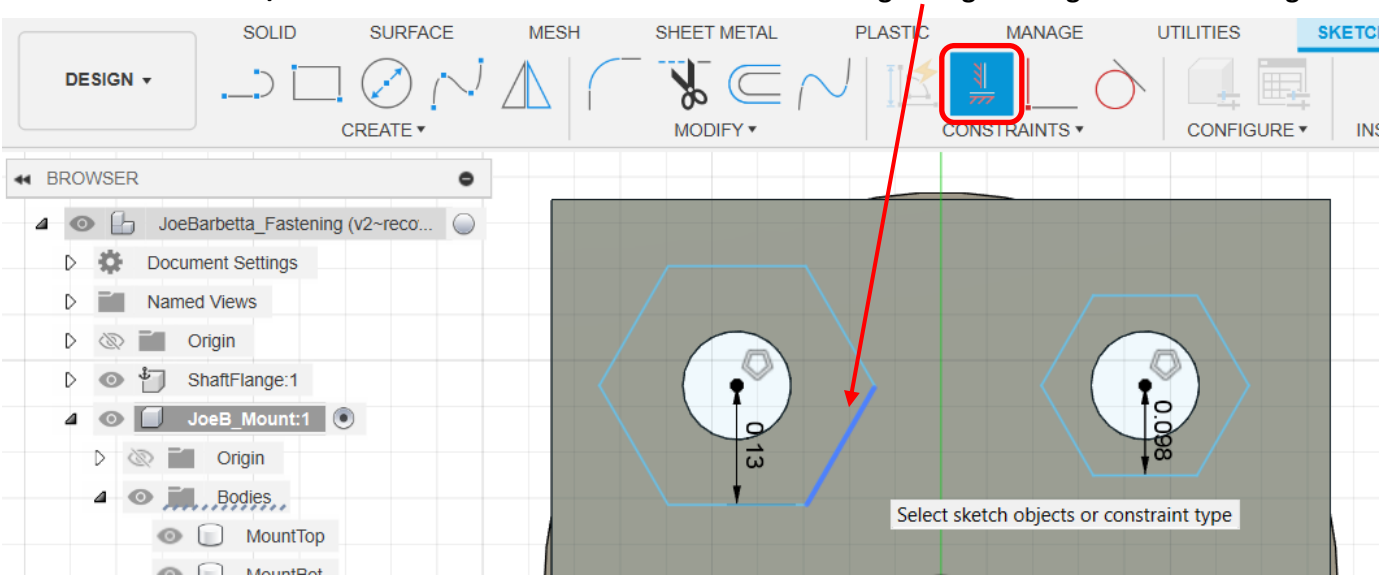
- click on the **center of the top left hole**, extend the polygon edge down, and enter **0.13**



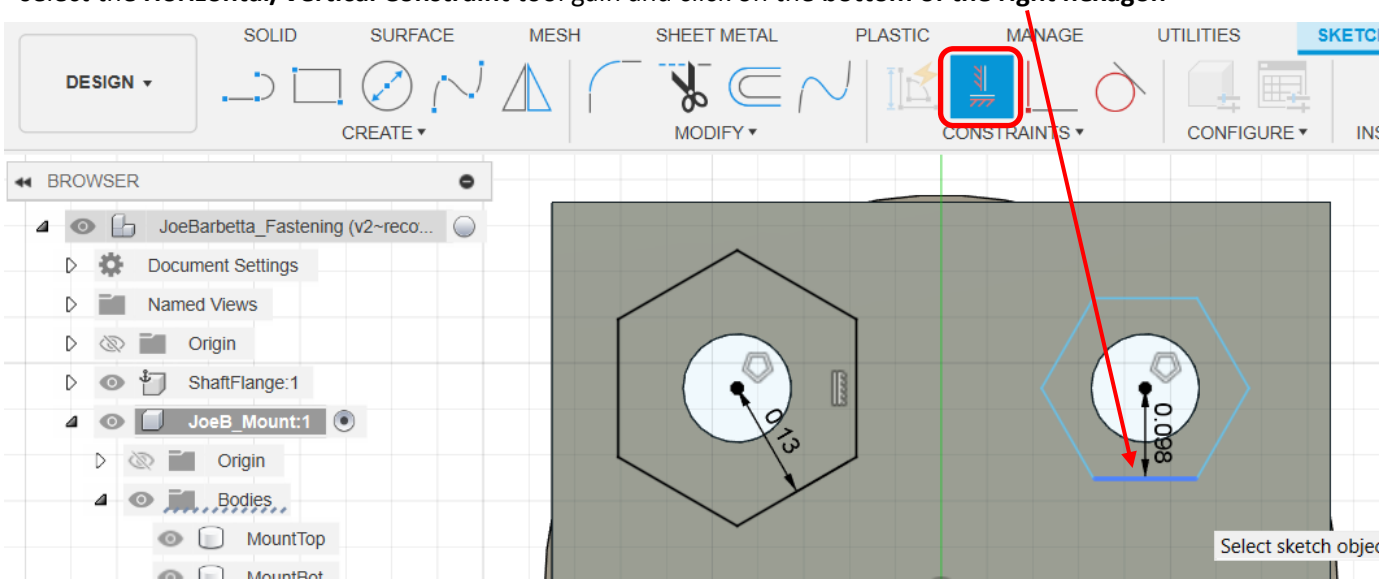
- select the **Polygon** tool again and click on the **center of the top right hole**, extend the polygon edge down, and enter **0.098**



- select the **Horizontal/Vertical Constraint** tool and click on the **bottom right diagonal edge** of the **left hexagon**

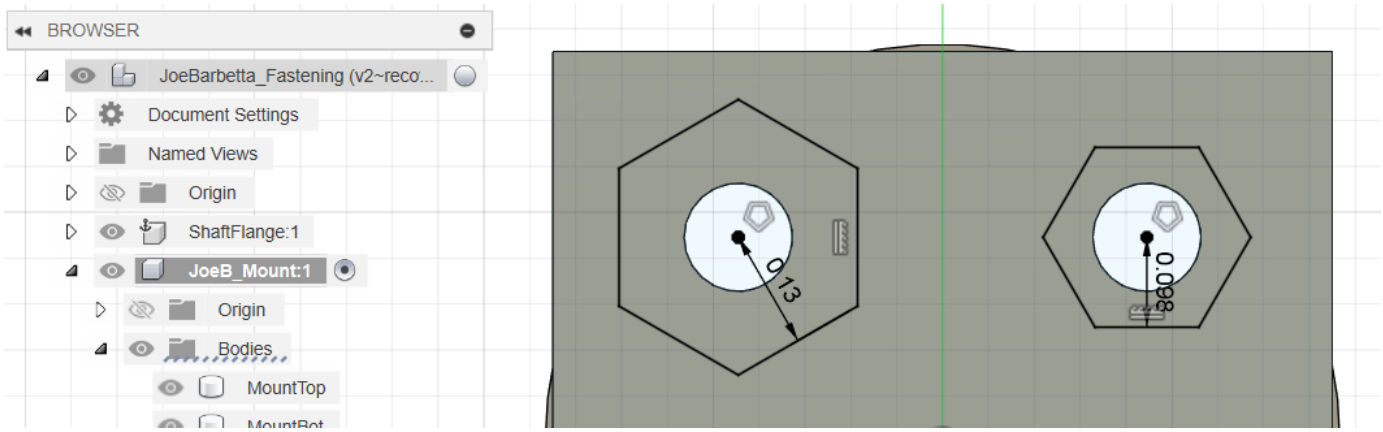


- select the **Horizontal/Vertical Constraint** tool gain and click on the **bottom of the right hexagon**

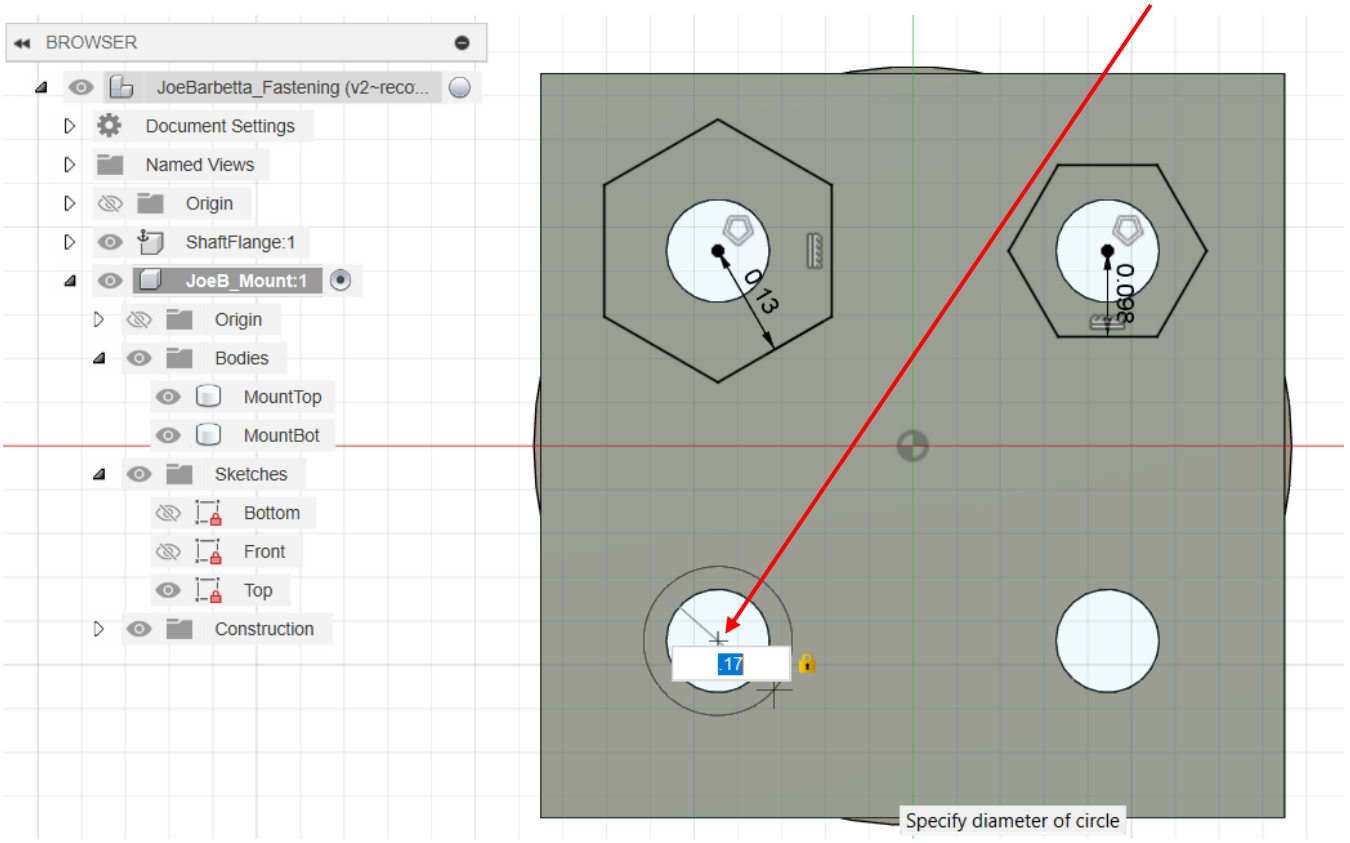


The values of 0.130 and 0.098 were used to create polygons with a width of 0.260” and 0.196”. The full size nut for a 4-40 screw has a width of 1/4” (0.250”) and the small pattern nut has a width of 3/16” (0.187”). The polygon values will provide some clearance between the walls and the nut sides in the pockets that the nuts will sit in.

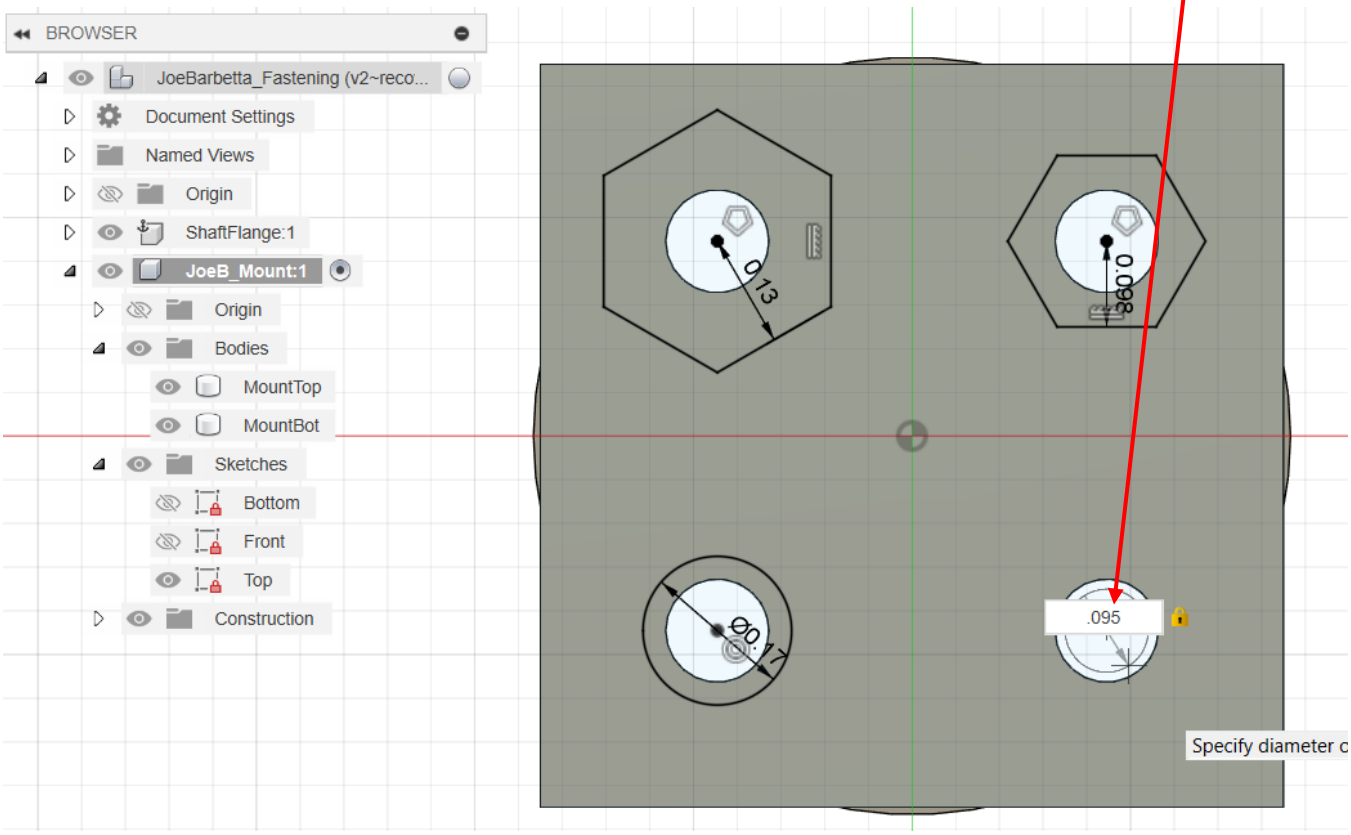
This is the result. Note how using this constraint is a method to control the angle of the hex nut. Sometimes a particular angle will be better for a design depending on the amount of material above or below a nut compared to that on the left or right side of a nut.



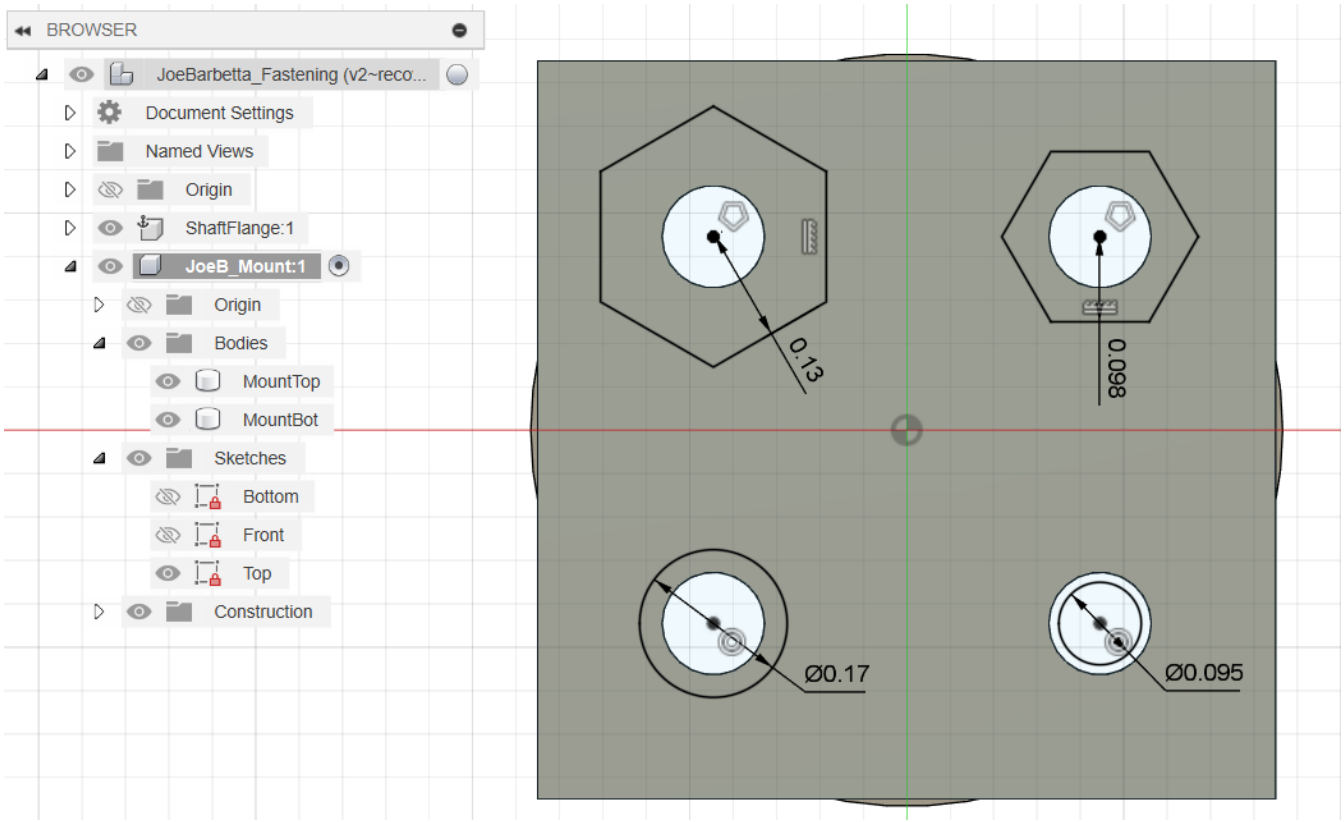
- select the **Center Diameter Circle** tool and create a **0.17** diameter circle centered on the **bottom left hole**



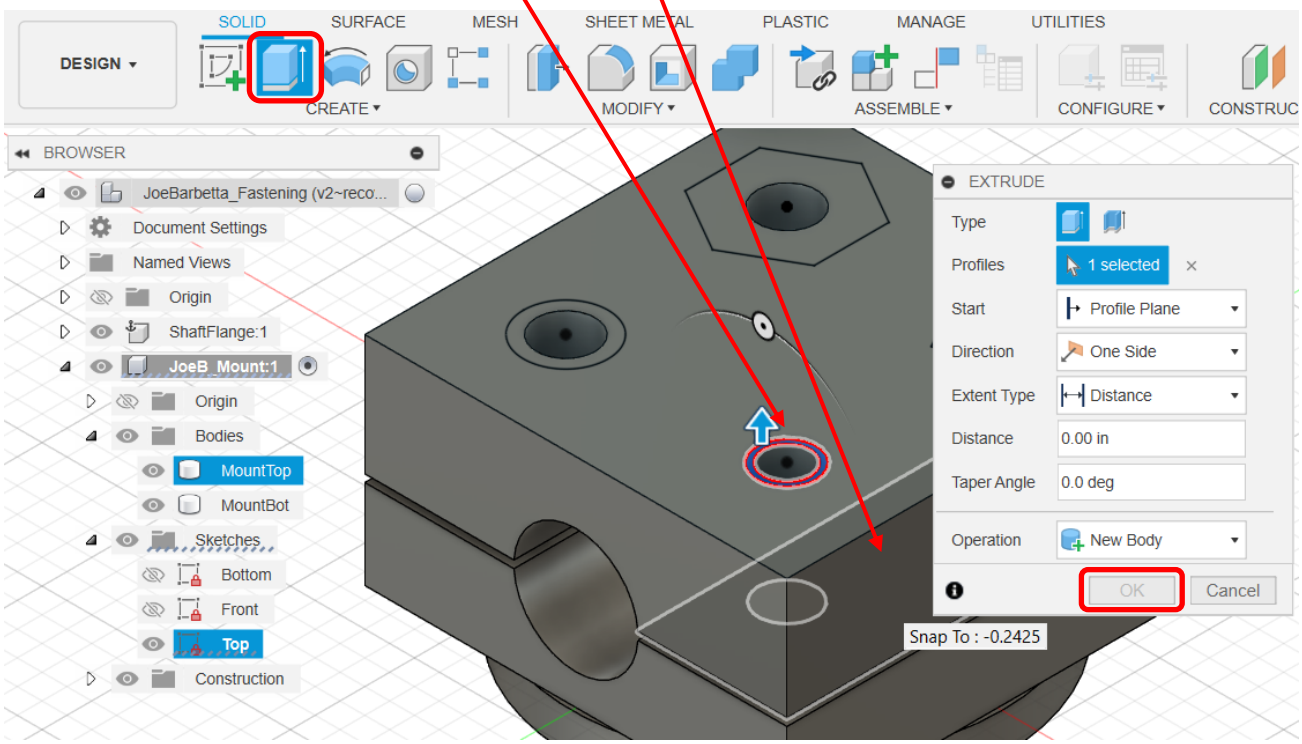
- select the **Center Diameter Circle** tool and create a **0.095** diameter circle centered on the **bottom right hole**
Note that this circle will be smaller than the hole.



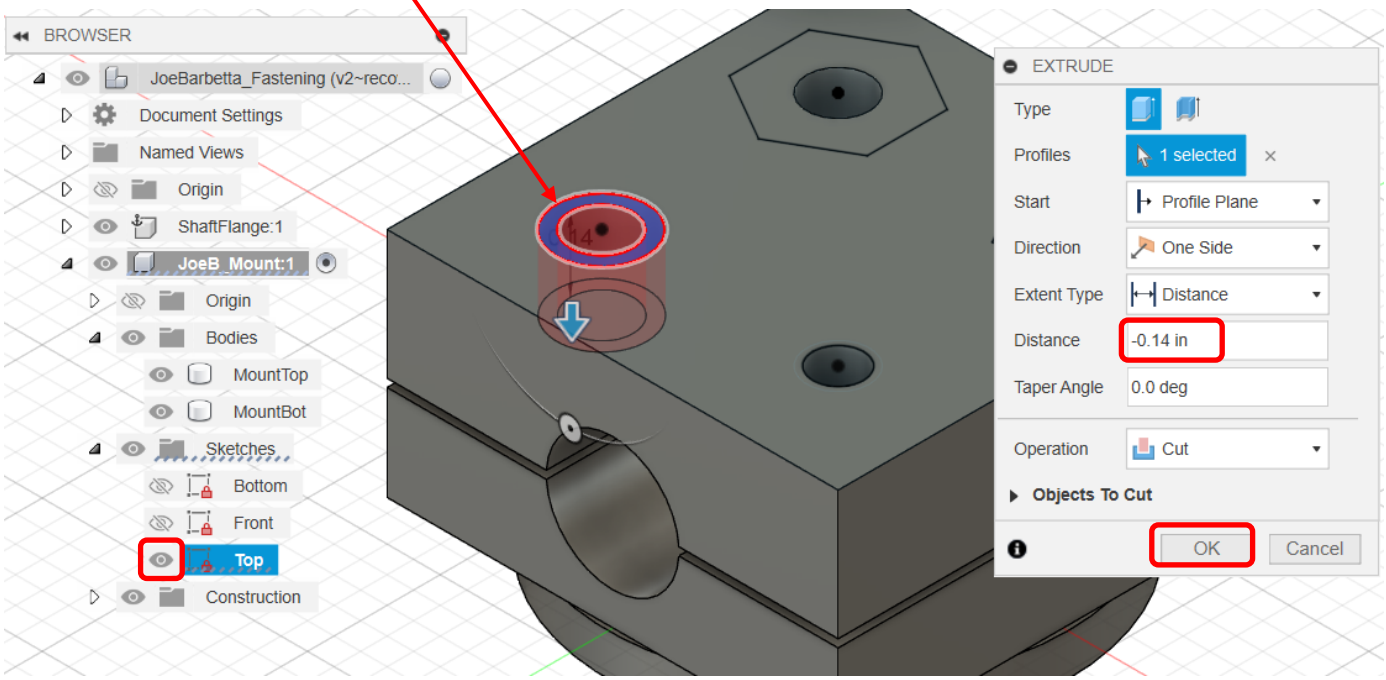
- drag the Dimension values to make them easier to see
- click **Finish Sketch** and click on the **Home icon** at the **View Cube**



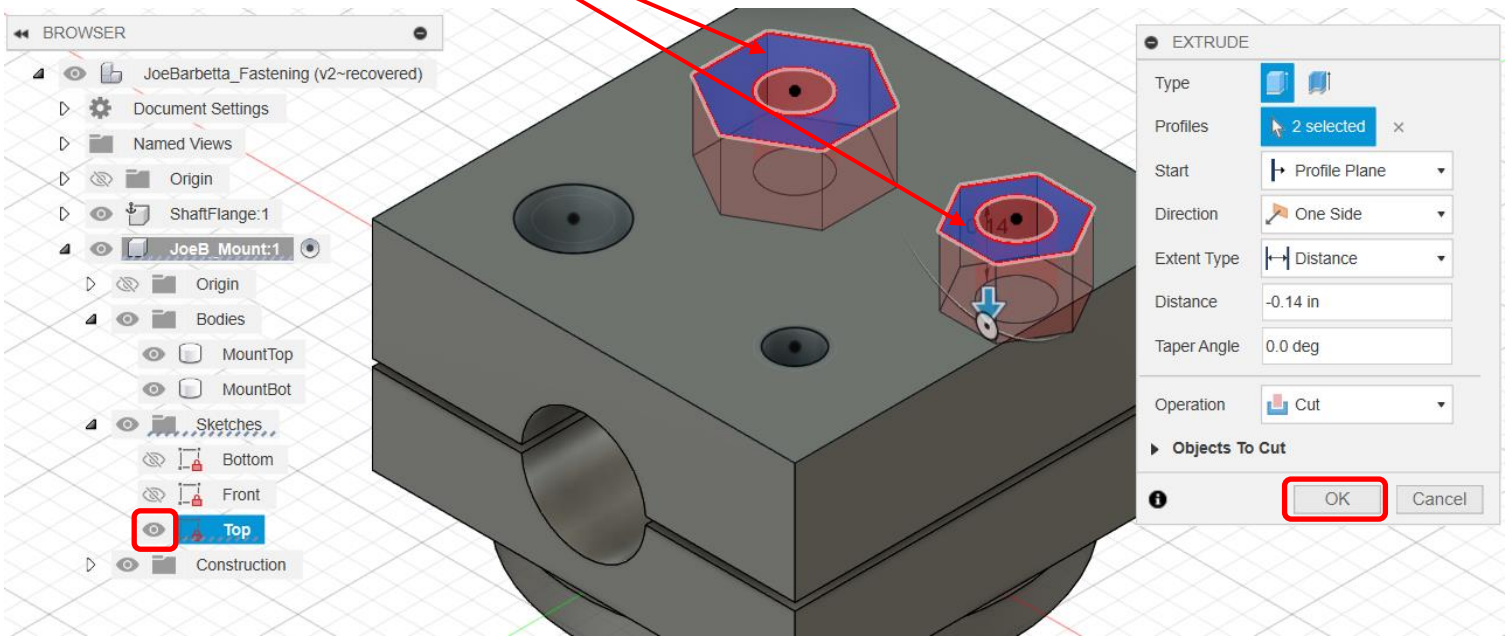
- select the **Extrude** tool
- click on the small **region between the hole and circle** to highlight it blue
- move the **mouse over the region indicated by the arrow**, which should highlight the bottom surface and click
- click **OK**



- click on the **eye** icon for the **Top** Sketch to make it visible again
- select the **Extrude** tool again
- click on the **region between the left hole and the outer circle**
- enter **-0.14** (note the minus sign) for **Distance** and click **OK**

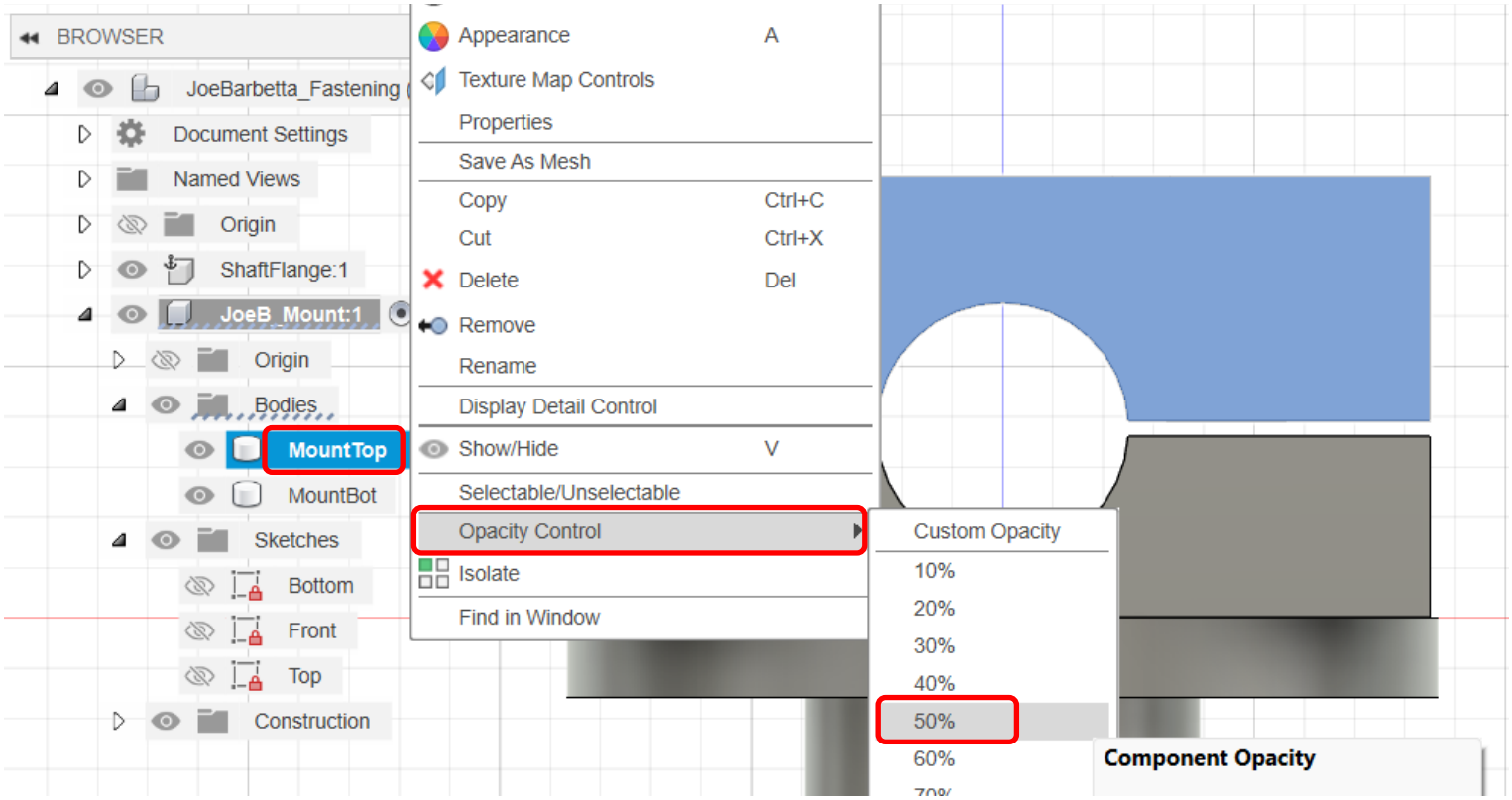


- select the **Extrude** tool again
- click on the **2 hexagonal regions**
- enter **-0.14** (note the minus sign) for **Distance** and click **OK**
- click on the **eye** icon for the **Top** Sketch to hide it



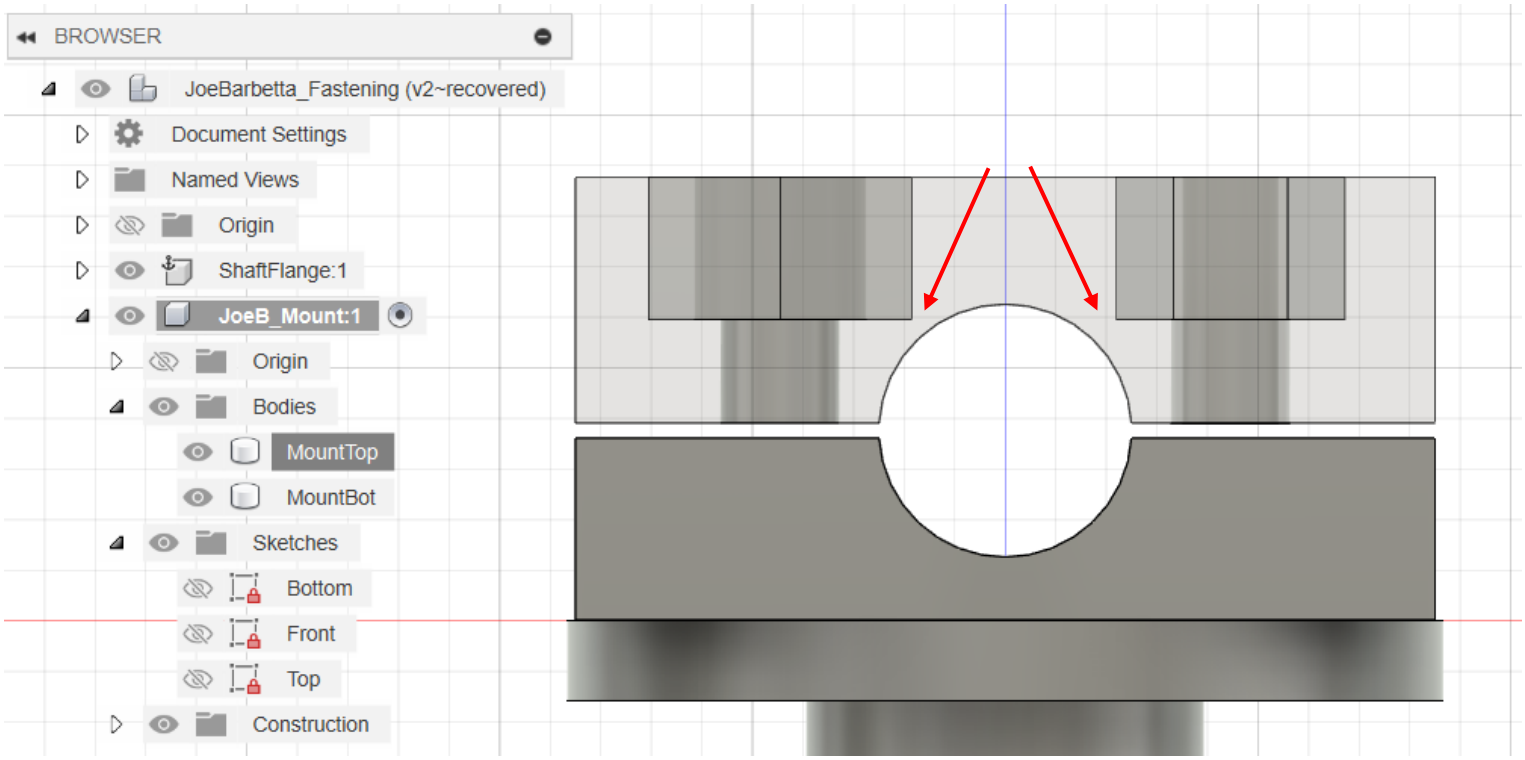
Adjusting Opacity

- right-click on **MountTop** and select **Opacity Control** and **50%**

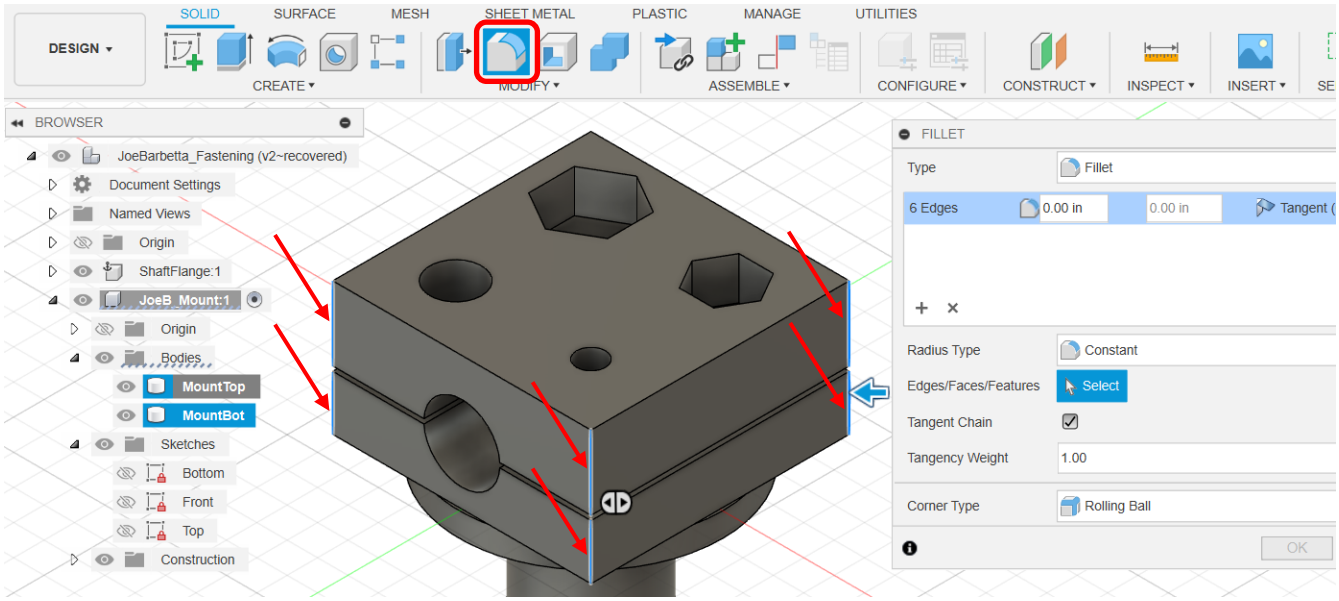


Note how this allows one to see the distance between to bottom edges of the pockets and the horizontal hole for the rod. The pocket depth was chosen so that the 1/2" screws will easily reach the nuts.

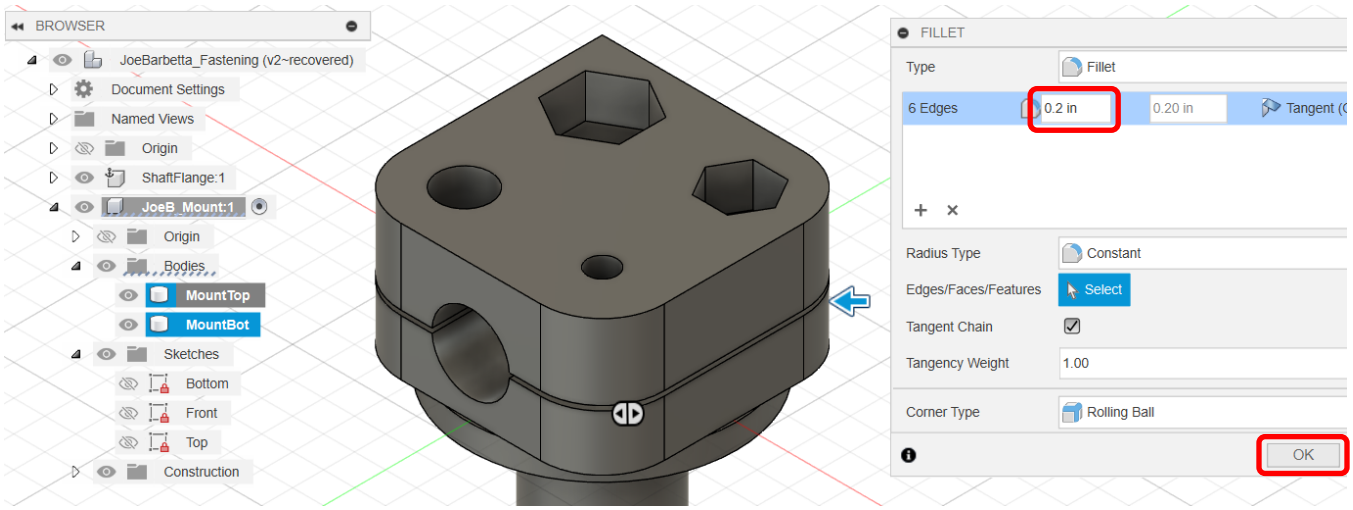
- change the **Opacity Control** back to **100%**



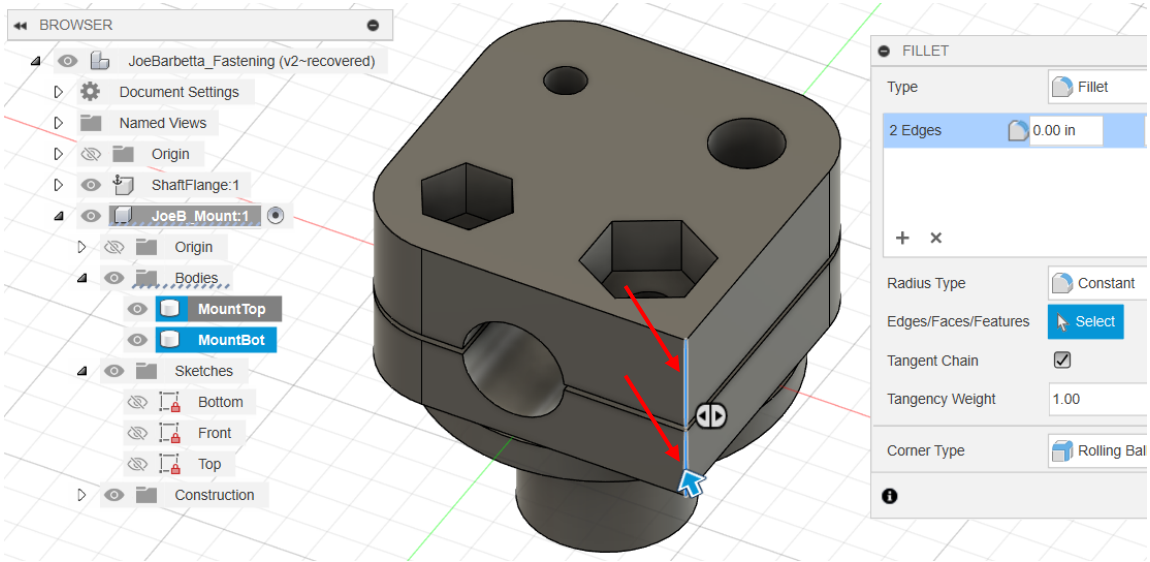
- select the **Fillet** tool. If it is not visible, find it in the **MODIFY** menu.
- click on the **edges** of the **MountTop** and **MountBot** bodies, which will turn the edges blue



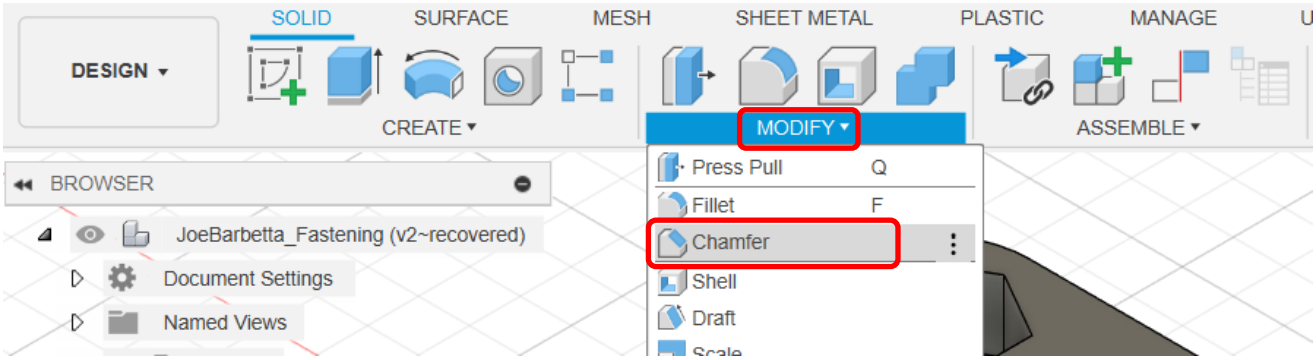
- enter a value of **0.2** and click **OK**. The 0.2” value is not critical. It reduces some plastic use and enhances aethetics.



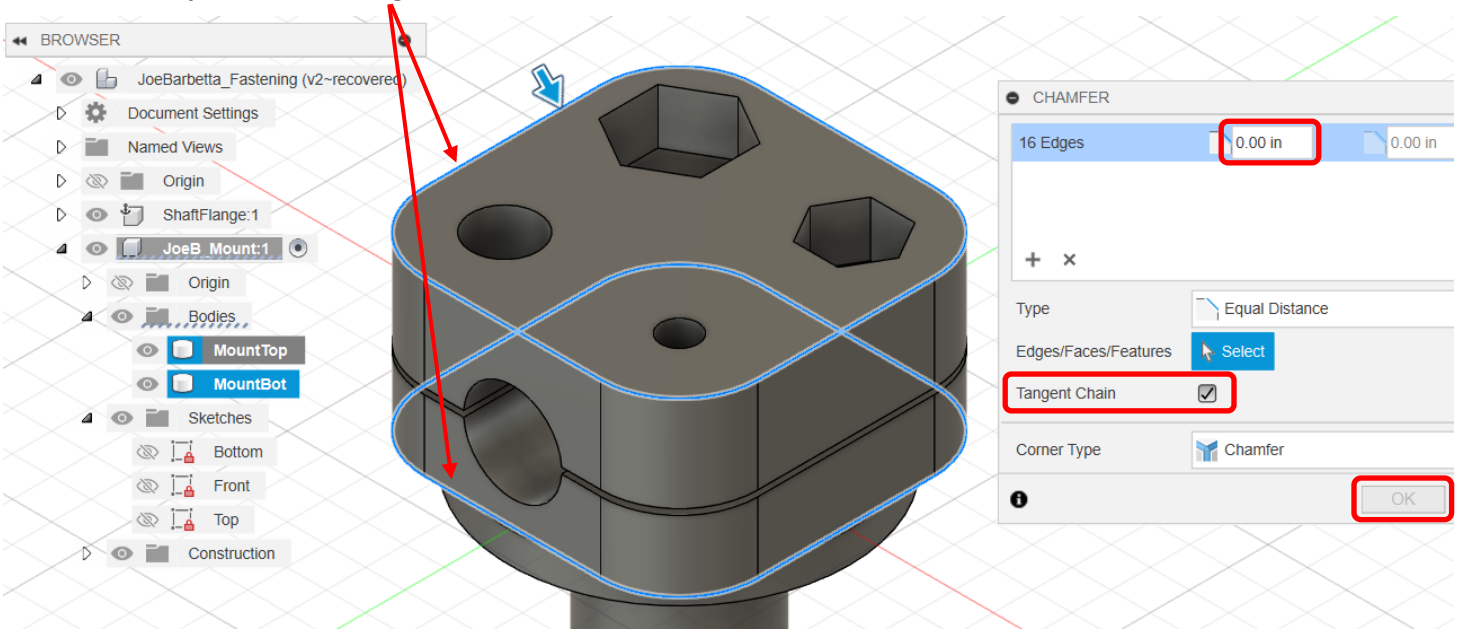
- use the **View Cube** to access the rear edge and perform the same **0.2 Fillet** operation



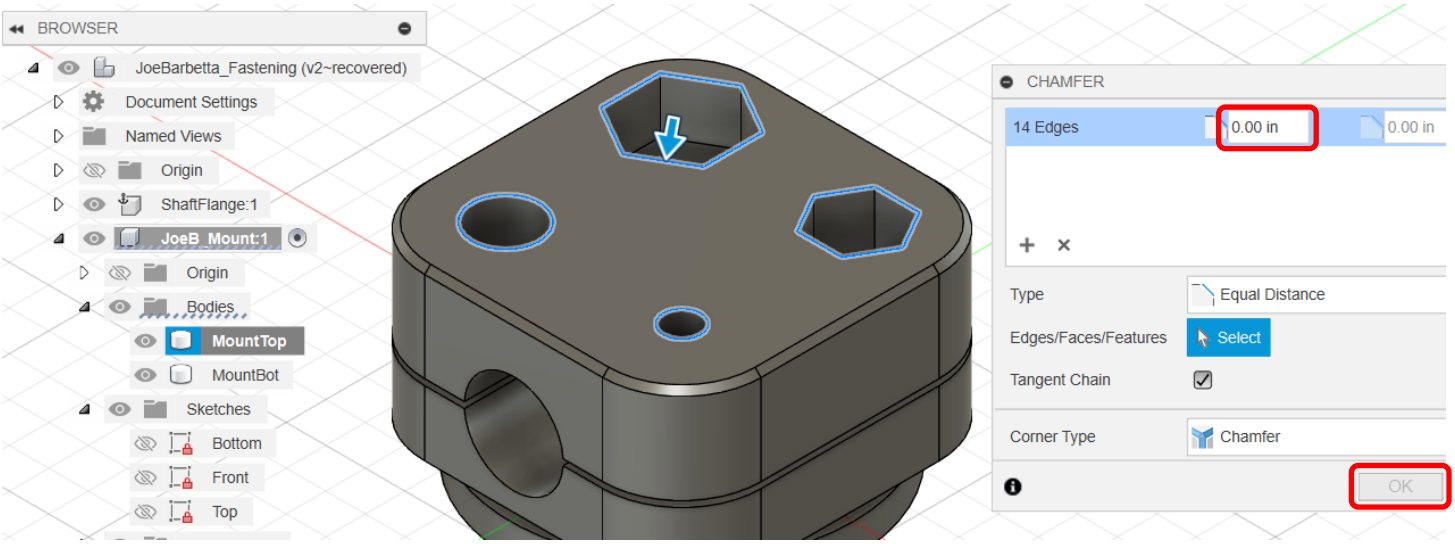
- from the **MODIFY** menu select **Chamfer**



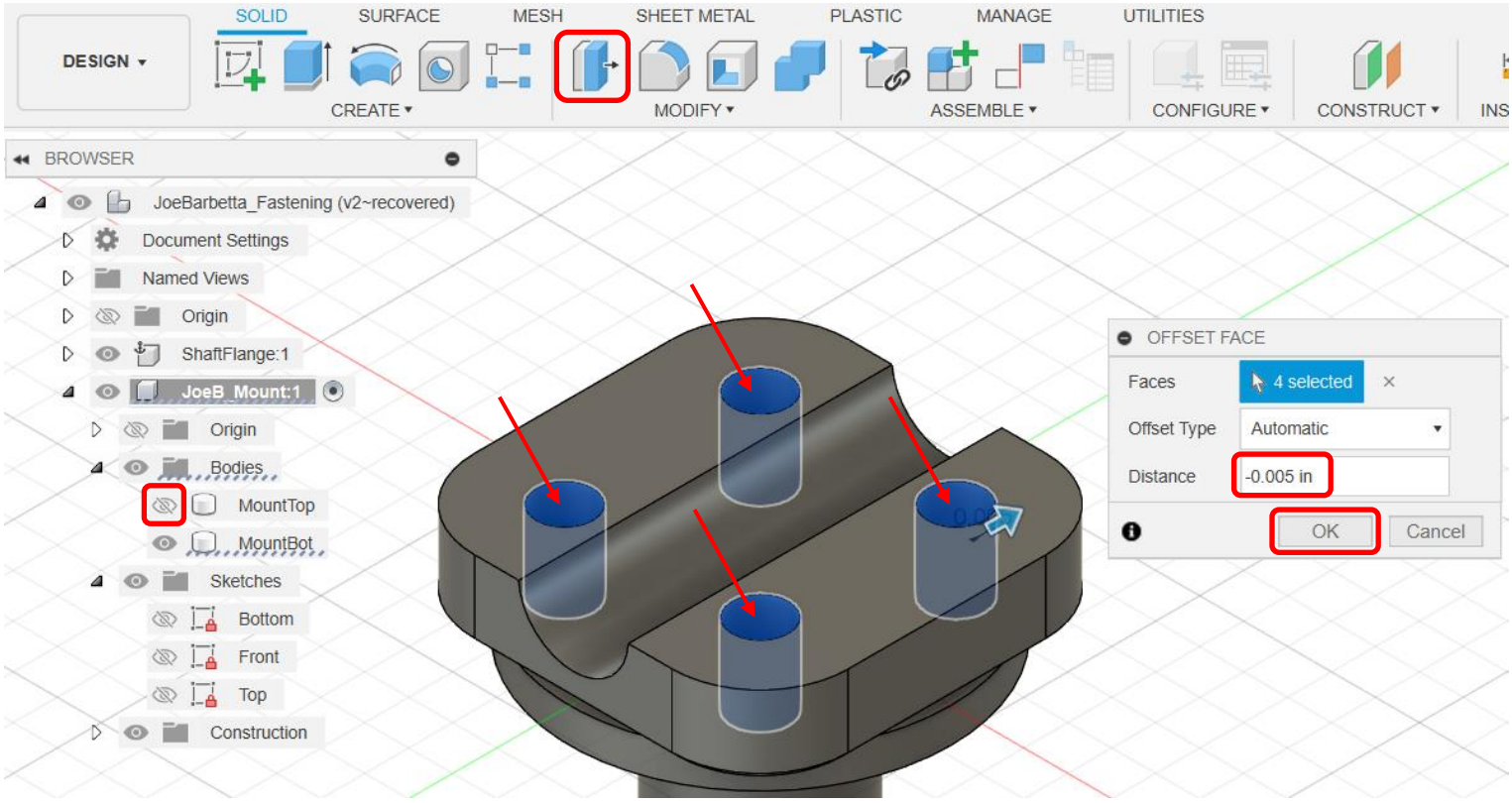
- ensure **Tangent Chain** is checked
- click on the **top and bottom edges** and enter a value of **0.015** and click **OK**



- select the **Chamfer** tool again
- click on **each edge of the hexagonal pockets** and the **edges of the 2 holes** and use a value of **0.015**. Note that each of the 6 edges for each hexagon needs to be selected.
- click **OK**

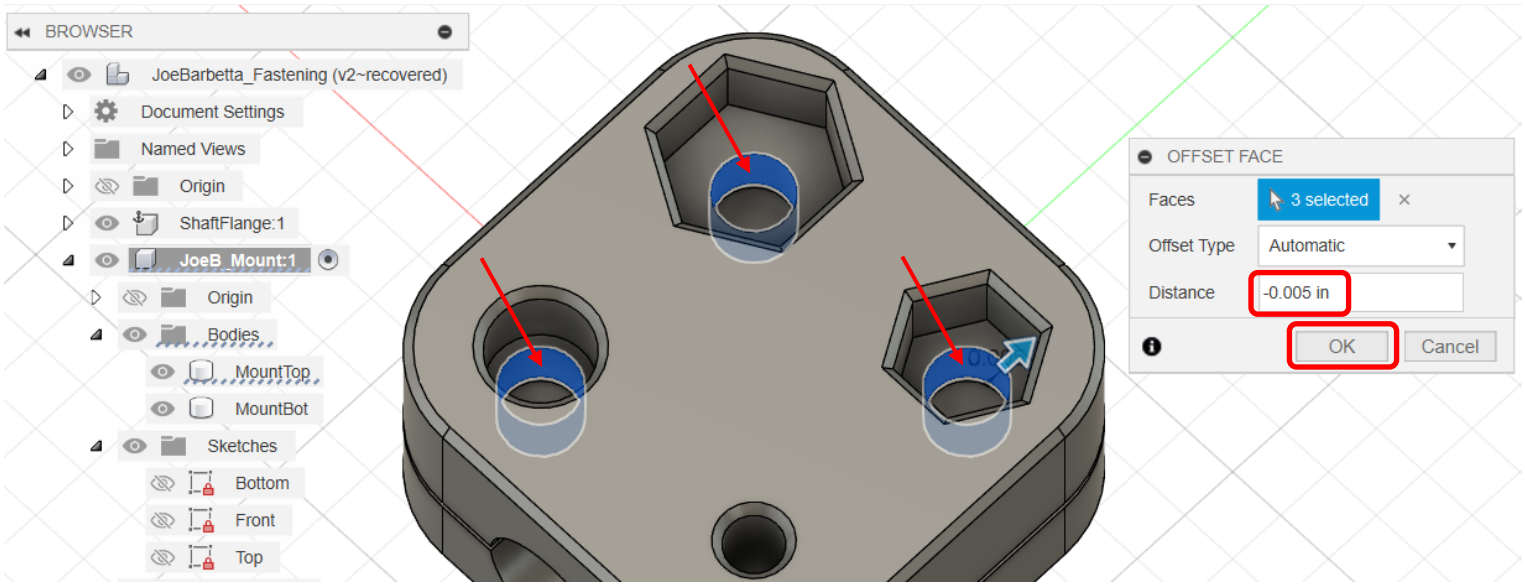


- click on the **eye** icon of the **MountTop** body to hide it
- select the **Press-Pull** tool. If it is not visible, find it in the **MODIFY** menu.
- click on the **inside of each hole** to highlight their inner surface blue
- enter **-0.005** (note the minus sign) and click **OK**

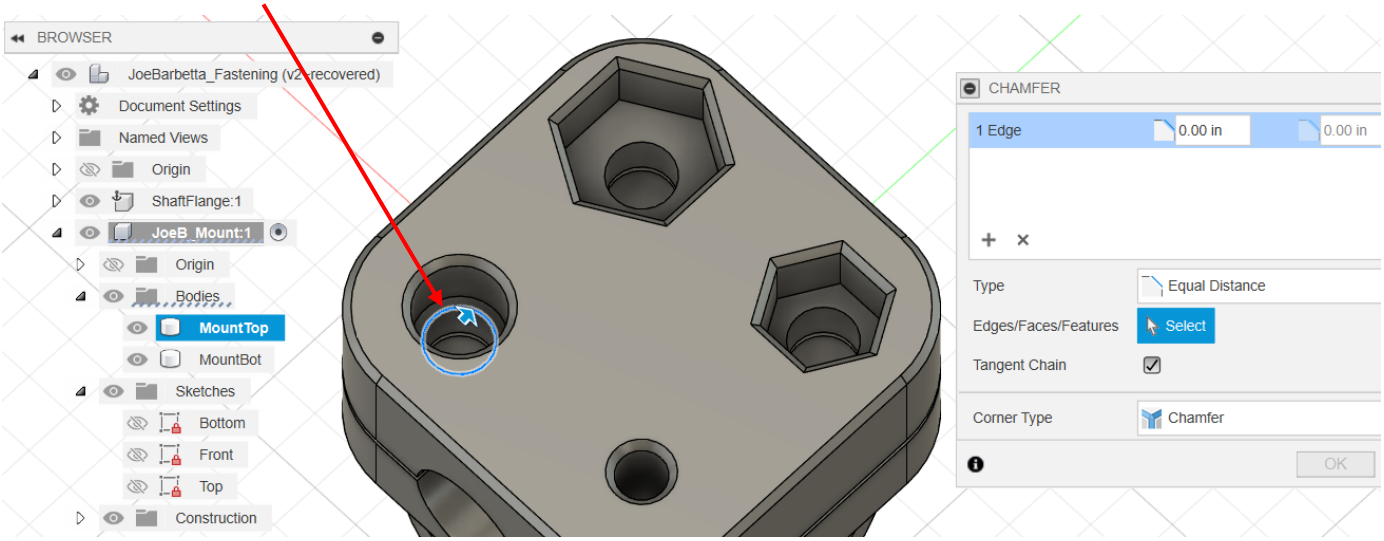


- click on the **eye** icon for **MountTop** again to make the top visible again
- use the **View Cube** to adjust the view to access the holes as shown below
- select the **Press-Pull** tool again
- click on the **inside of the 3 holes** indicated to highlight their inner surface blue
- enter **-0.005** (note the minus sign) and click **OK**

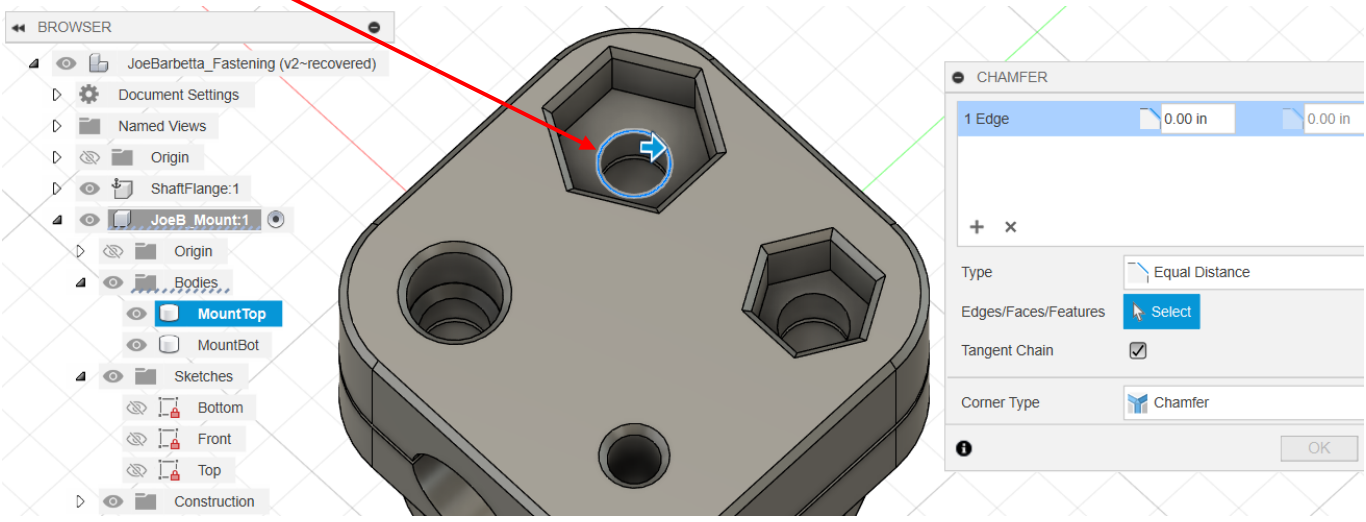
These operations expanded the holes by 0.01” to allow some extra clearance so the screws easily slide throught these holes.



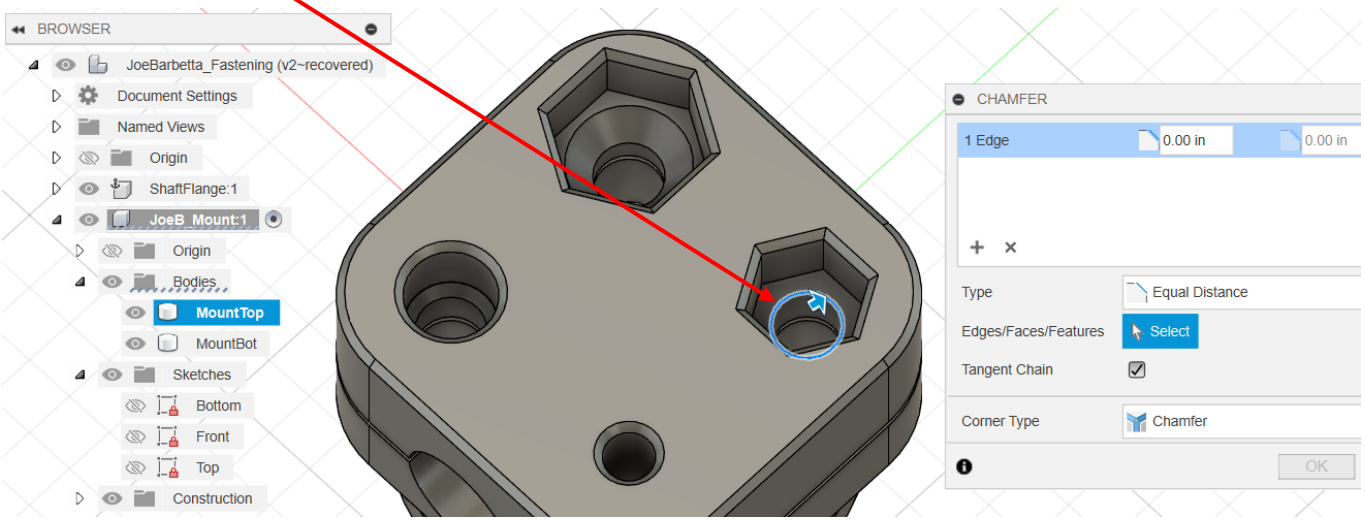
- from the **MODIFY** menu select **Chamfer**
- click on the **inside edge of the hole** and use a value of **0.02**



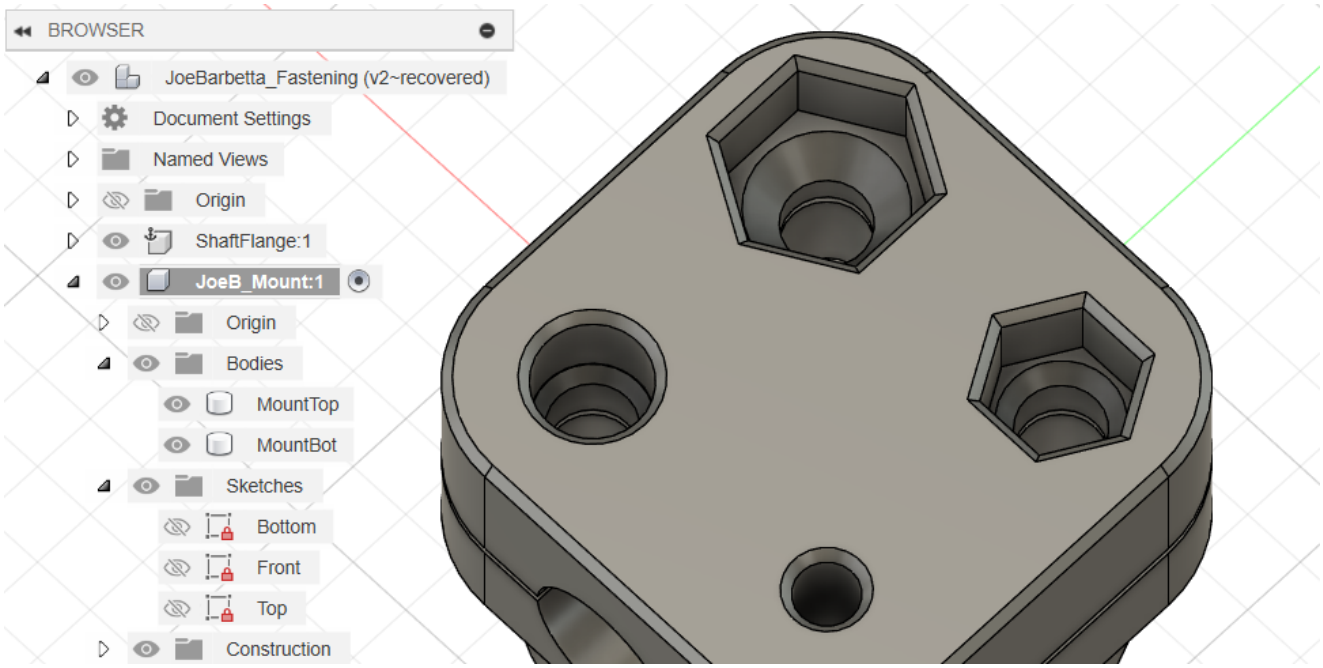
- select the **Chamfer** tool again
- click on the **inside edge of the hole** and use a value of **0.05**



- select the **Chamfer** tool again
- click on the **inside edge of the hole** and use a value of **0.025**



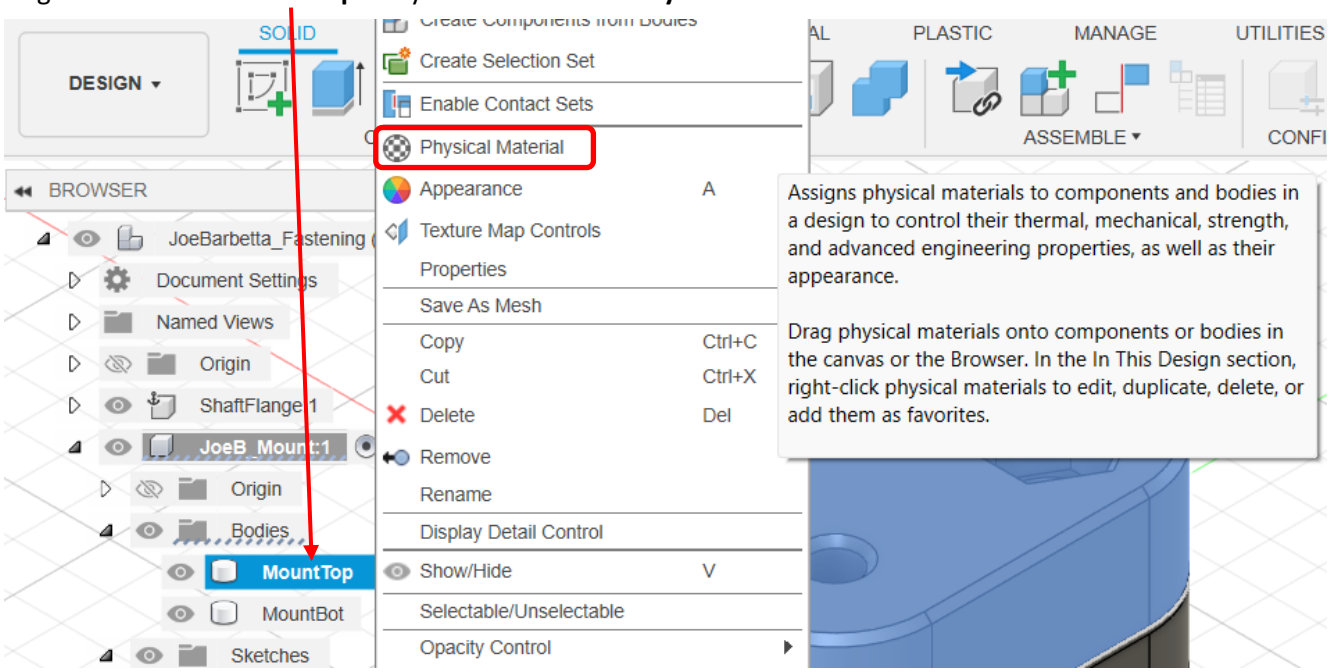
This is how the chamfers look. The reason for creating these chamfers is to reduce the amount of bridging if the top is printed with the top surface on the built plate of the 3D printer. It is always best to design a part so that supports are not needed. For the hexagonal pockets the nuts may get pulled down very slightly into the chamfer region, but this will not be a problem.
It should be noted that even without these chamfers, the pockets can still print nicely. The chamfers just clean up the pockets. Note that one could chose to print the MountTop body with its bottom surface on the print bed. The top of the 0.25" hole will be slightly distorted, but the part can still be functional.
Of course, one can perform a test print to evaluate these features.



Setting the Material

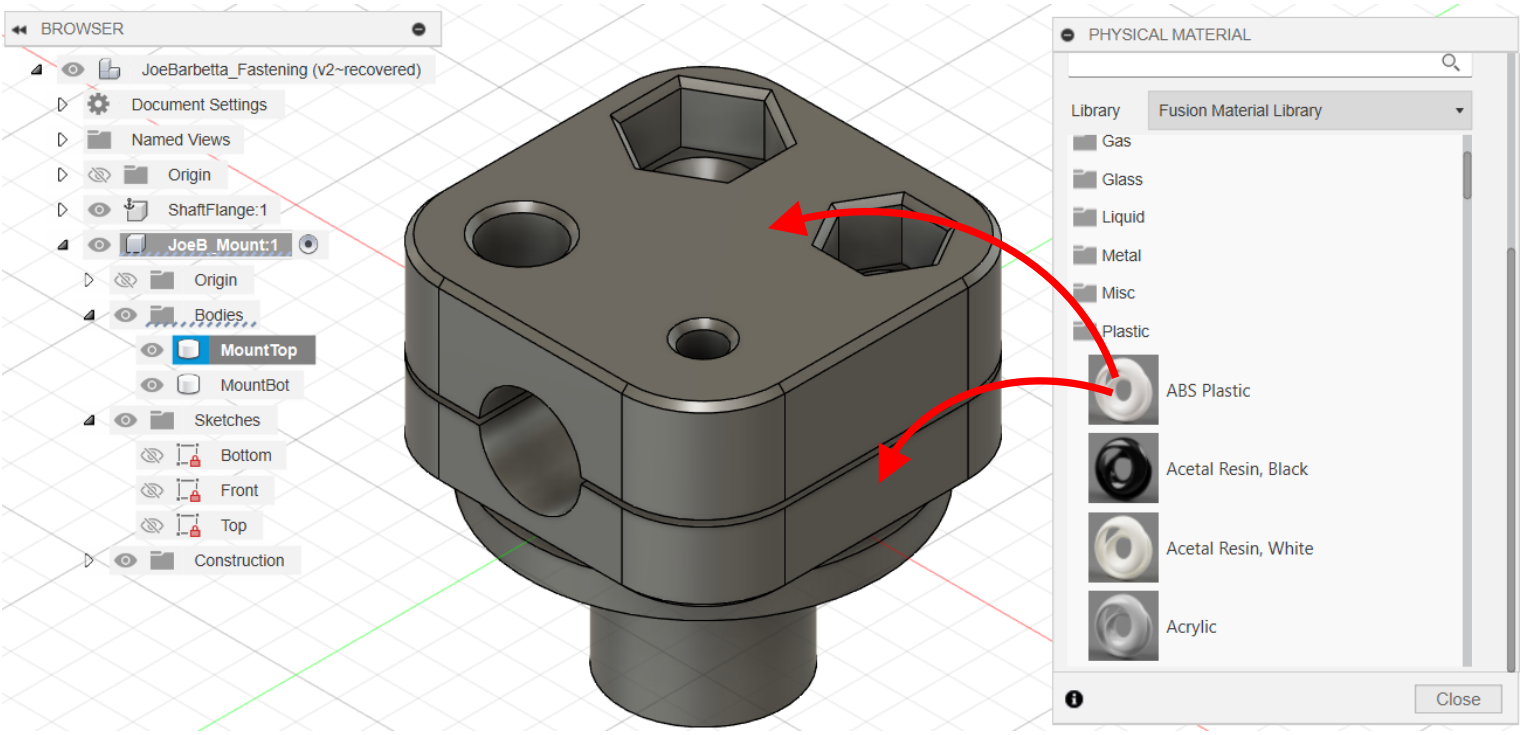
The color of the final part will be determined by the filament in the 3D printer and thus the material and color assigned in CAD is not important. However, it does make it easier to identify parts if materials and colors are assigned. The default material in Fusion is steel. Don't expect your 3D printer to spit out a gray steel part if you don't change the material.

- right-click on the **MountTop** body name and select **Physical Material**

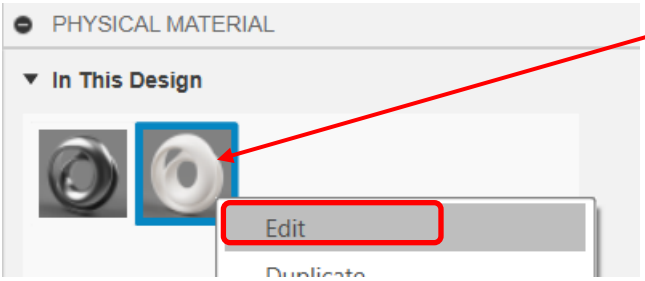


- scroll down to the **Plastic folder** and click on it
- drag the **ABS Plastic** icon onto both the MountTop and MountBot bodies, which should turn them white

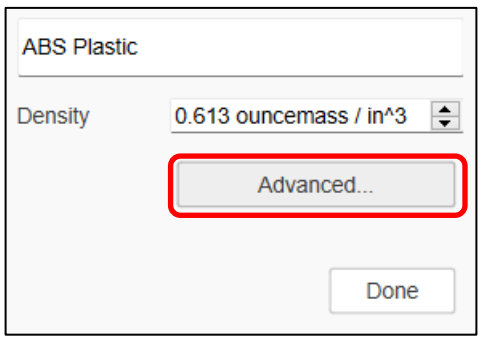
PLA is the most common material for 3D printed parts. At the time of writing Fusion does not have PLA in its list of materials so ABS Plastic will suffice. Do note that ABS can be used for 3D printing. It can be a little more difficult to print, but handles higher temperatures much better.



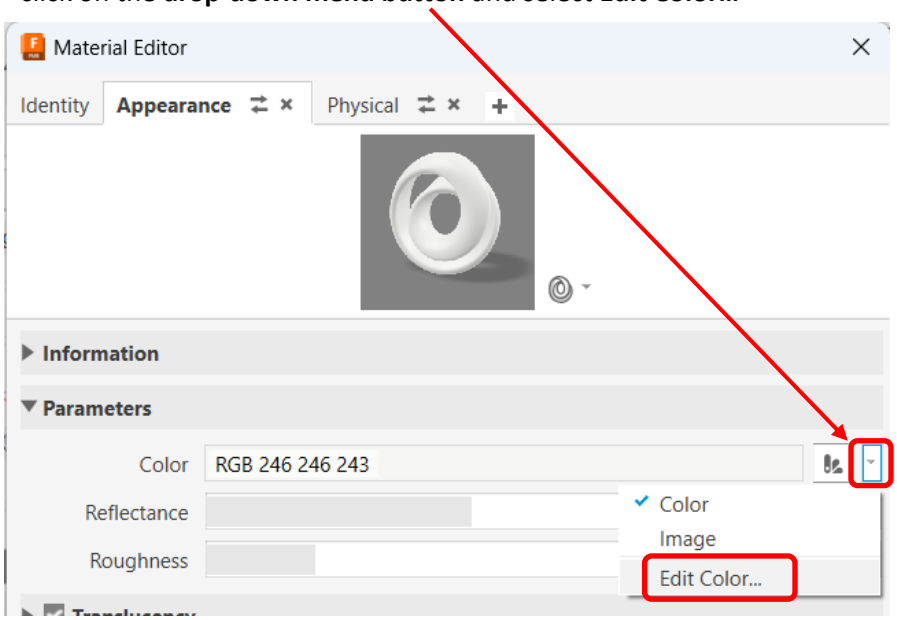
- at the top of the PHYSICAL MATERIAL window right-click on the **ABS icon** and select **Edit**



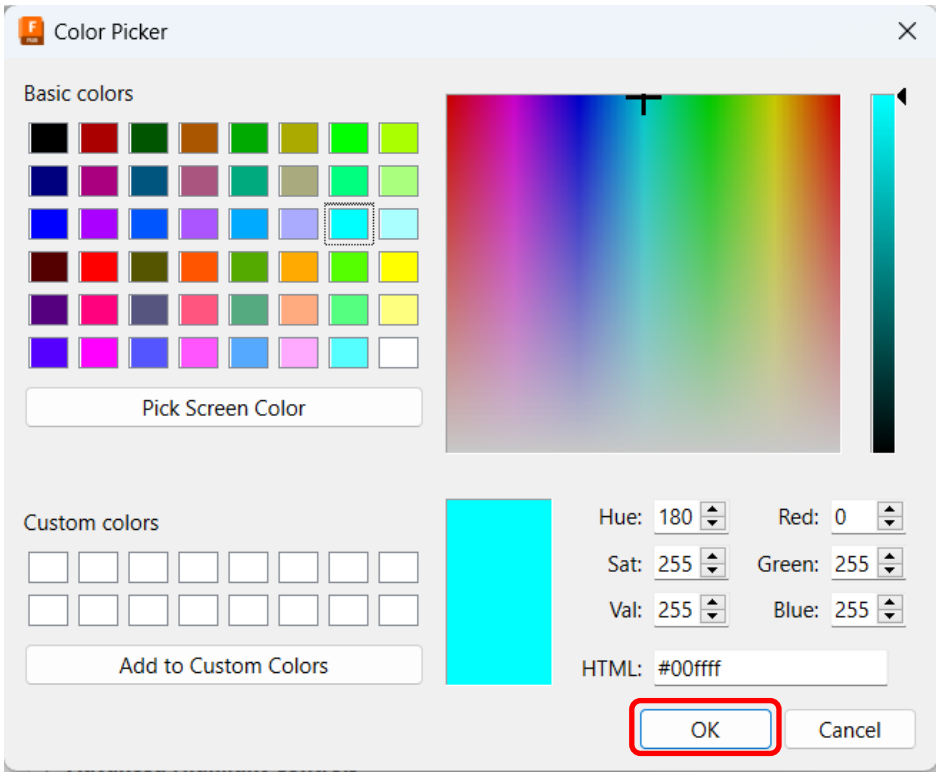
- click on the **Advanced** button



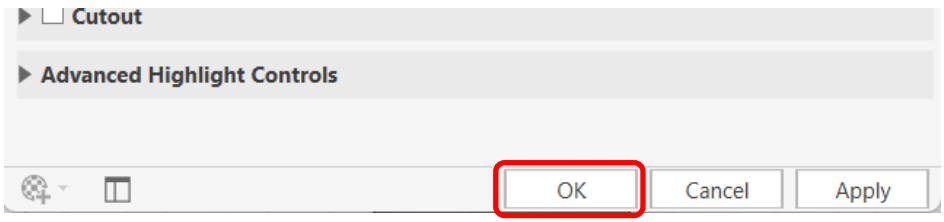
- click on the **drop-down menu button** and select **Edit Color…**



- using either the **Basic colors boxes** or the **color spectrum box**, select your **favorite color**.
The brightness bar on the right can be adjusted to brighten or darken the color.
- click **OK**

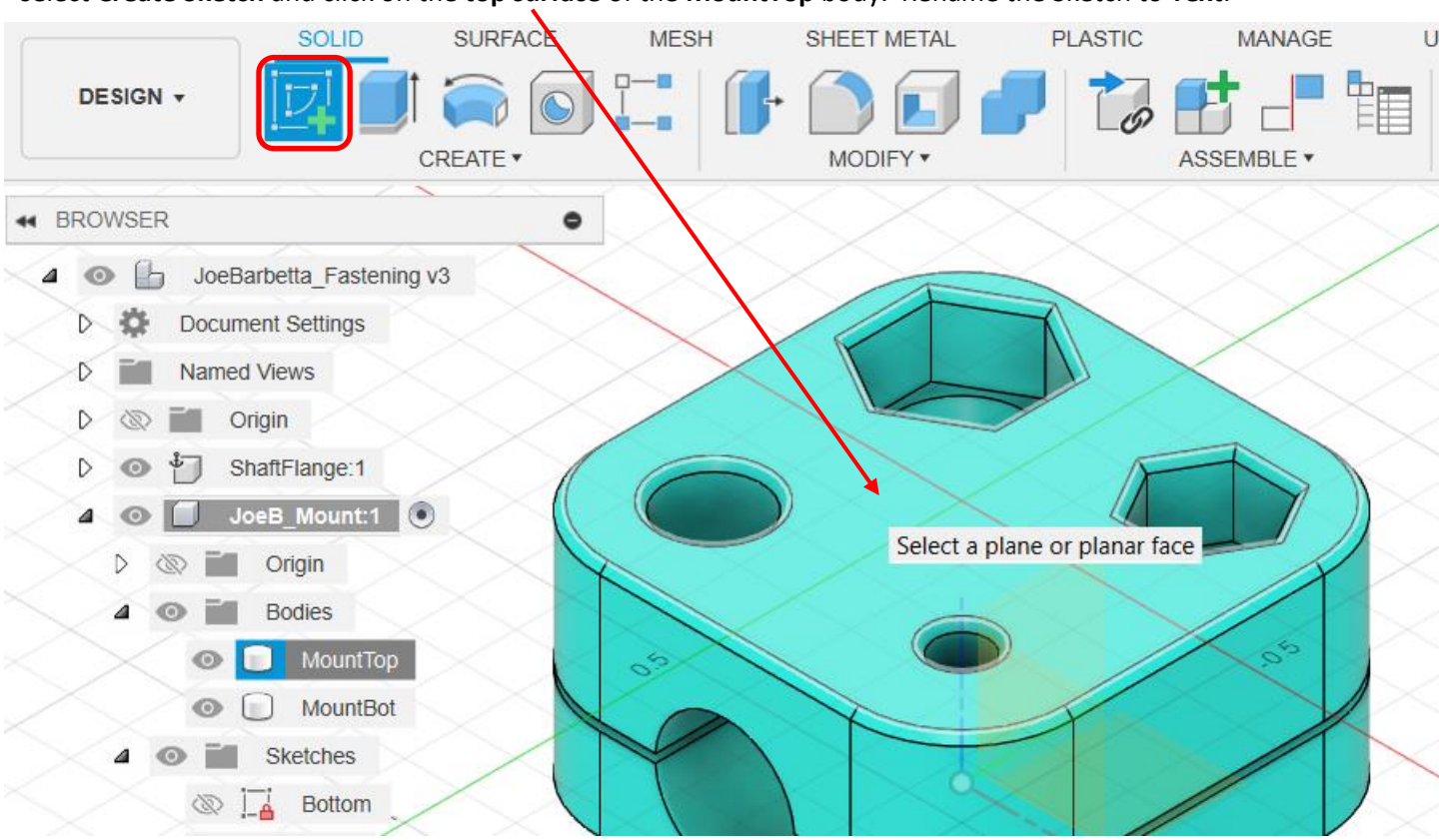


- at the bottom of the PHYSICAL MATERIAL window, click **OK**

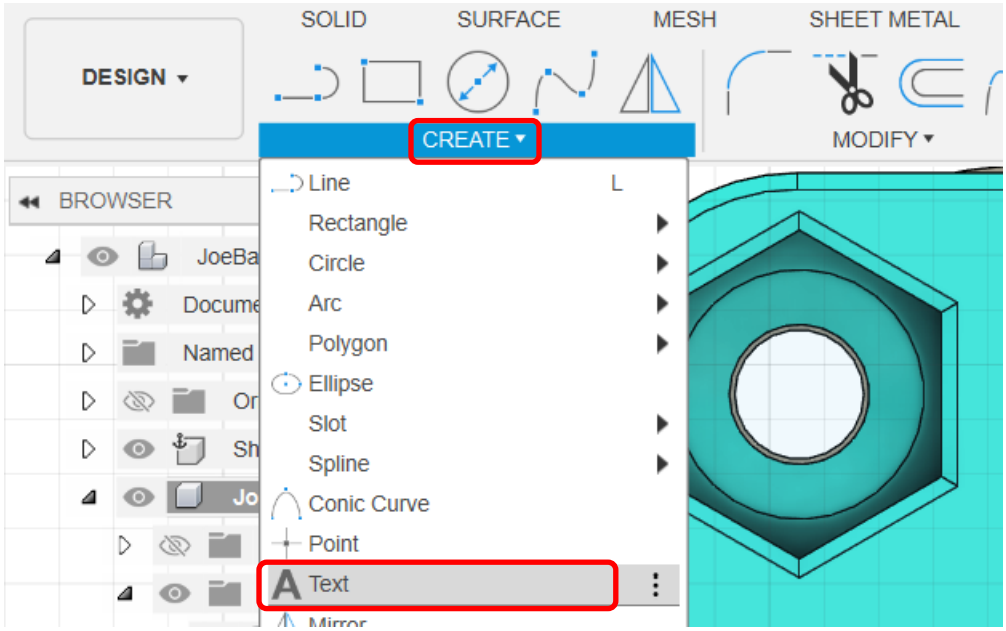


Adding Text

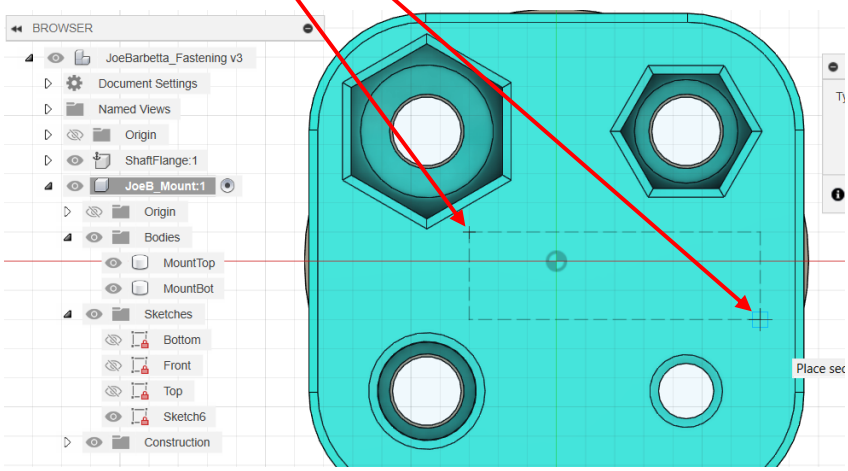
- select **Create Sketch** and click on the **top surface** of the **MountTop** body. Rename the Sketch to **Text**.



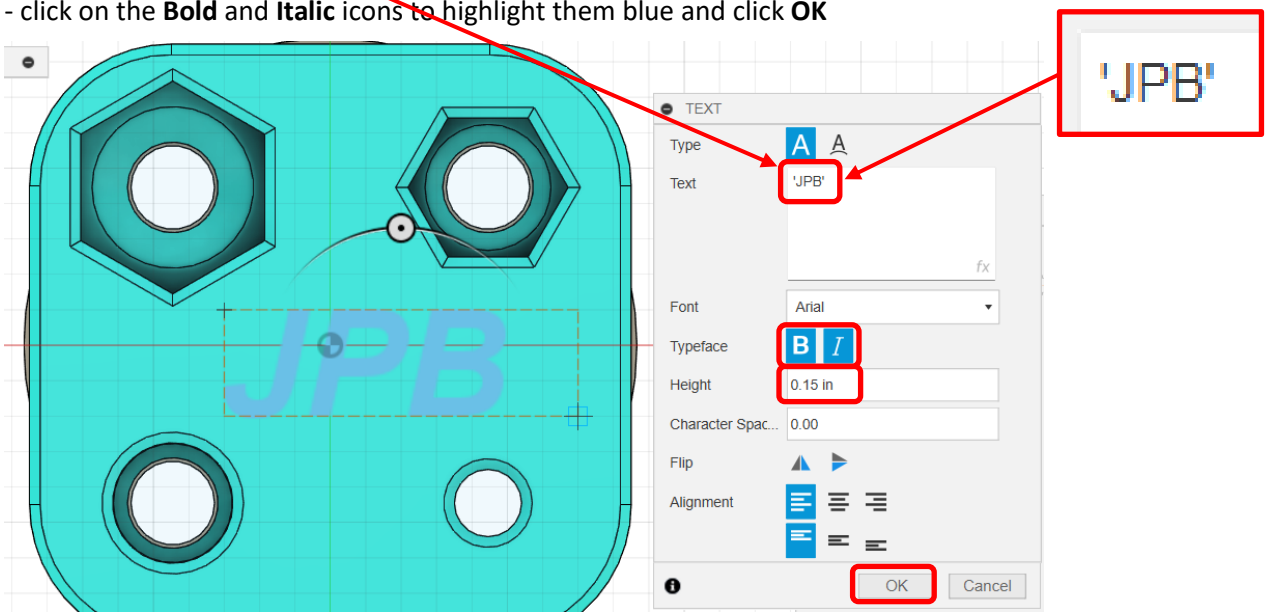
- from the **CREATE** menu select **Text**
- if a window about Parameteric Text appears click its **OK** button



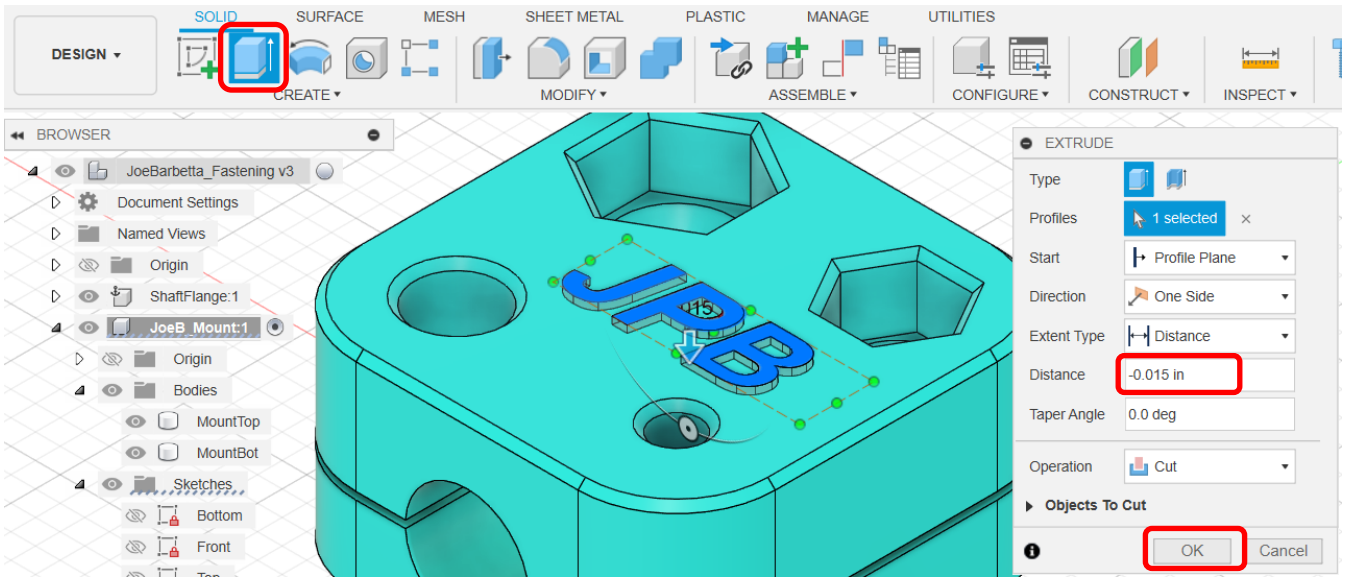
- click on a spot as indicated (the exact position is not critical) and extend the rectangle down and to the right
- click again on the spot as indicated. Again the position is not critical.



- at first the text may be very large. Enter **0.15** in the **Height** box to allow 3 letters to fit.
- type your **3 initials** in the **Text** box. The initialis must be **preceeded and followed by a single quote** as shown here.
- click on the **Bold** and **Italic** icons to highlight them blue and click **OK**

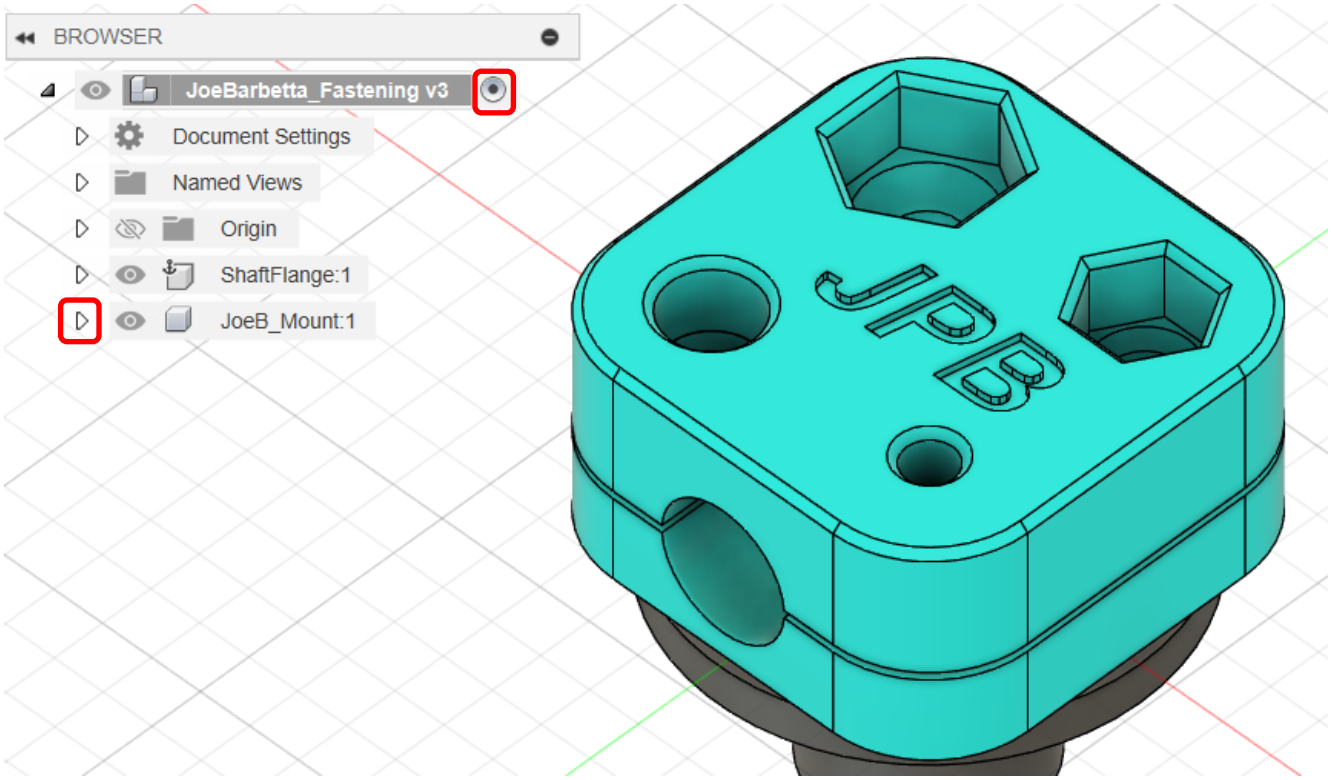


- click on the text to select it
- select the **Extrude** tool, enter **-0.015** (note the minus sign), and click **OK**

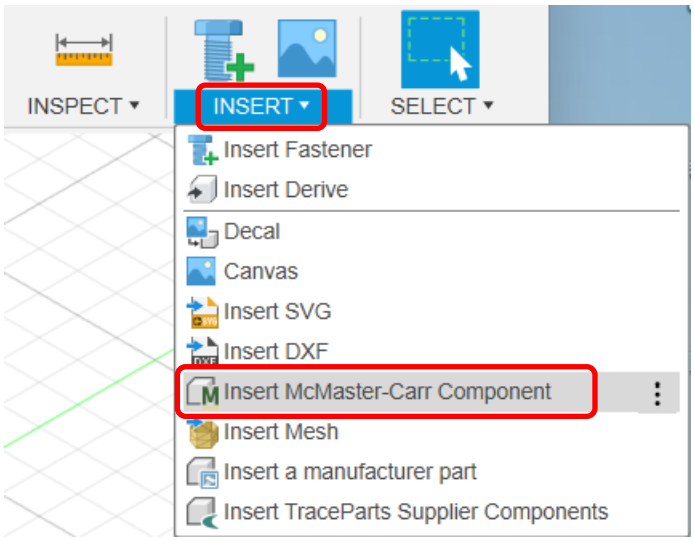


Inserting Components

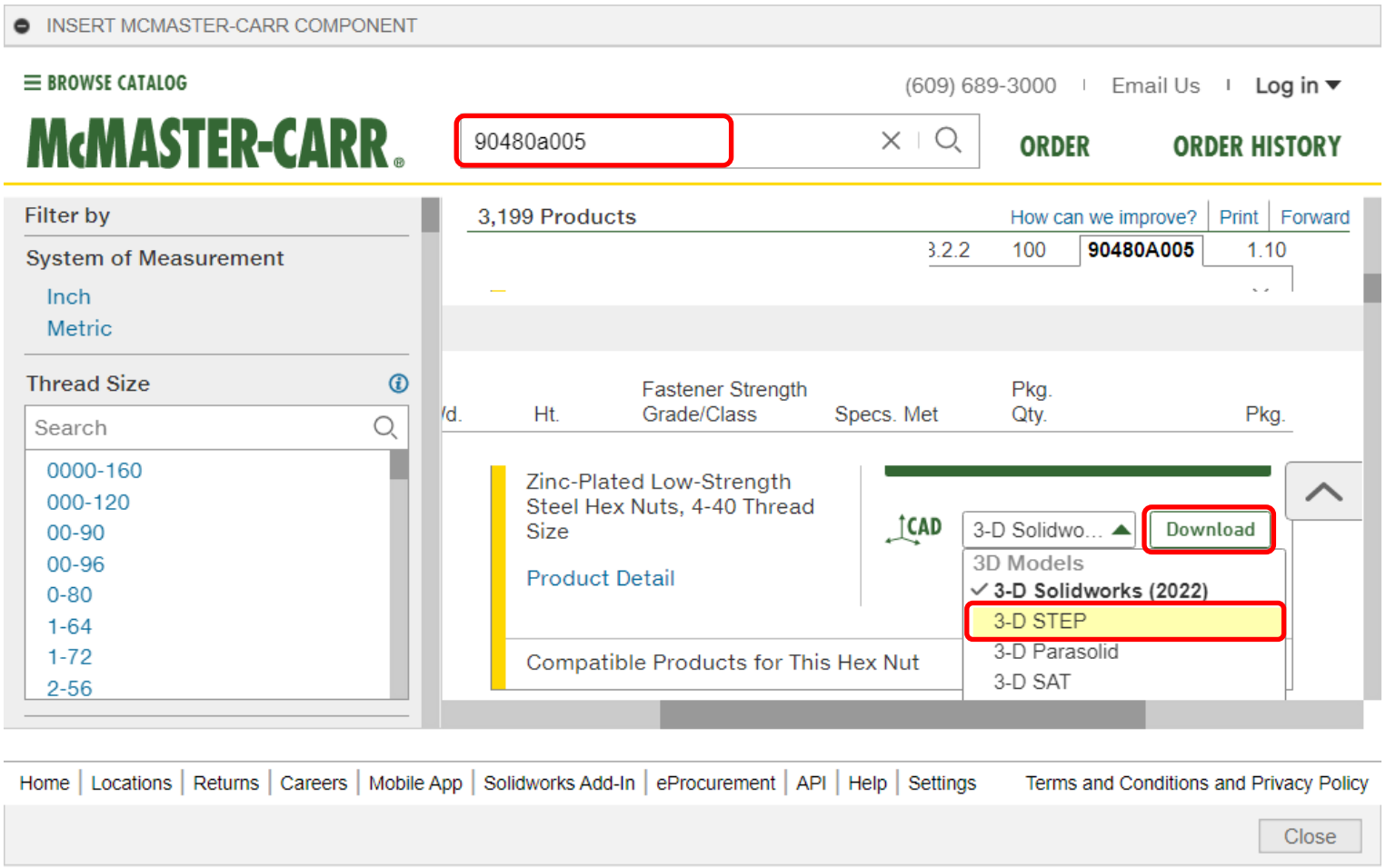
- click on the **arrow** next to the Mount Component to close it and click on the **circle** for the **Project Name** to activate it
- adjust the view to allow access to the edges of the chamfered holes



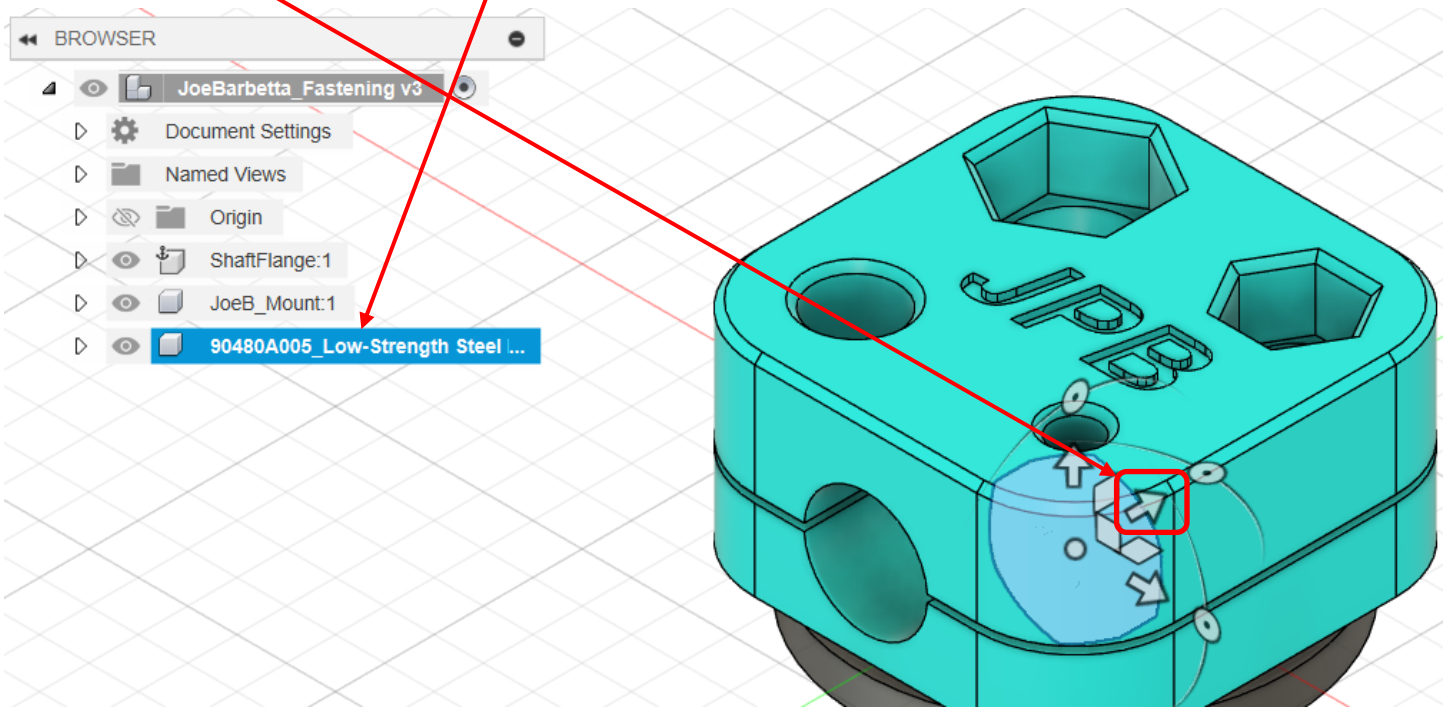
- from the **INSERT** menu select **McMaster-Carr Component**



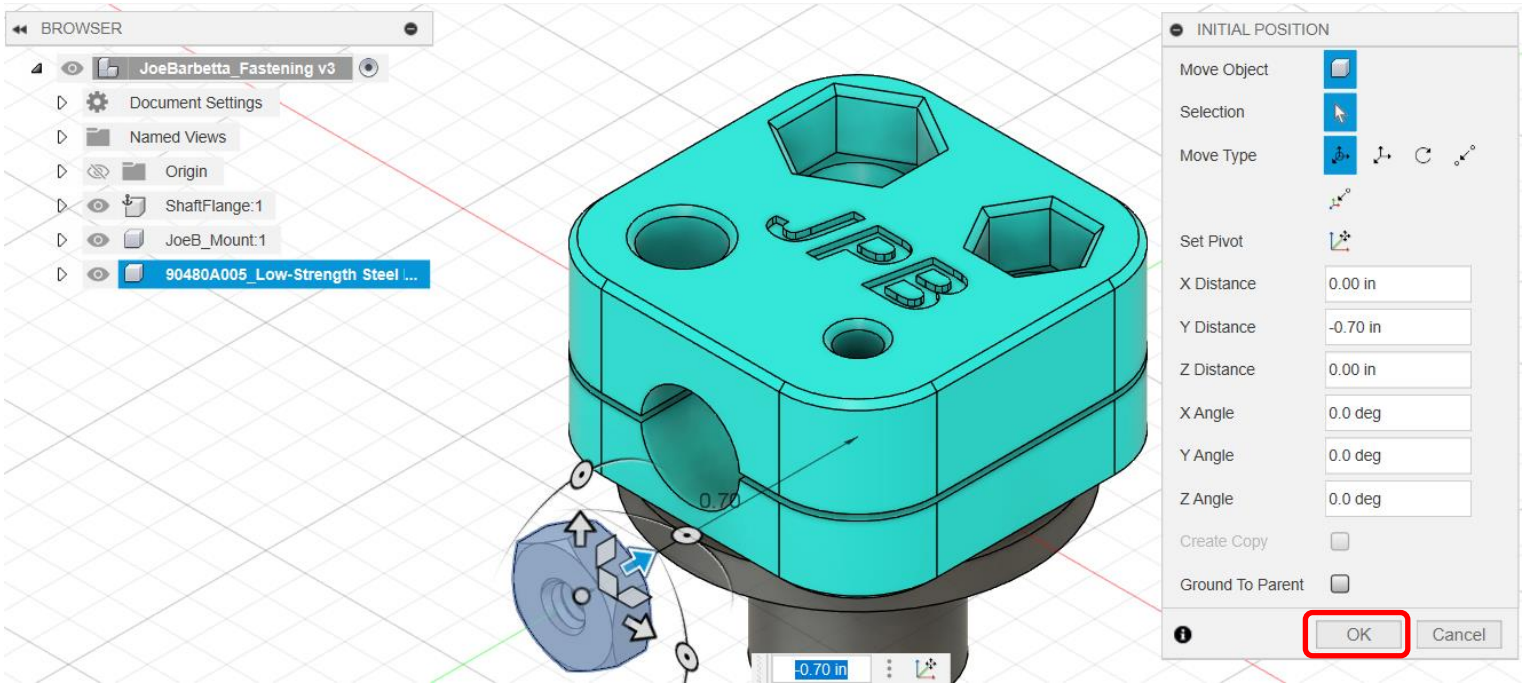
- type **90480A005** (or copy and paste) in the search box and press the **Enter key**
- one may need to scroll a little to the left and down to access the CAD options and Download button
- select **3-D STEP** and click the **Download** button



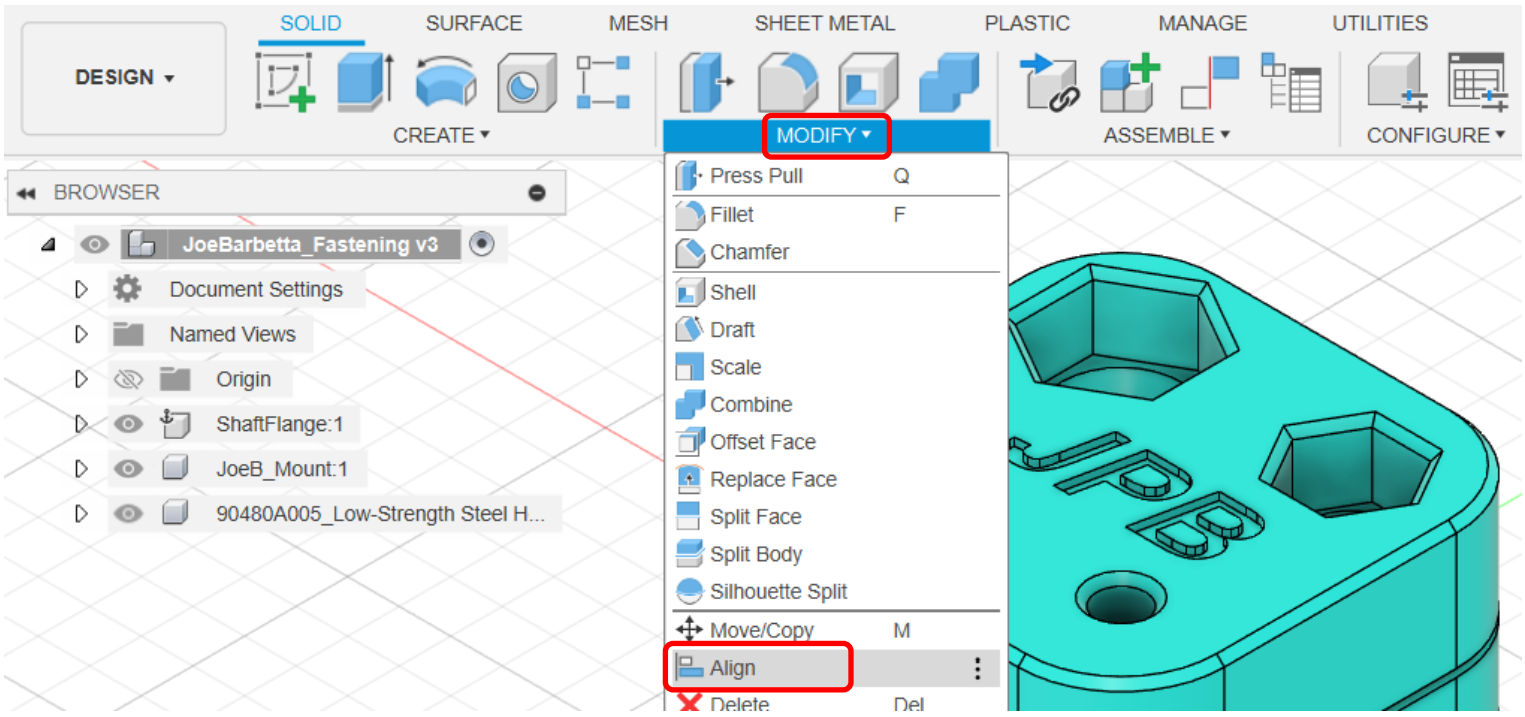
- click on the **arrow** indicated of the inserted component and drag it to the left until it is outside of the mount body
If needed, one can click on the **Component Name** in the Broswer to select it.



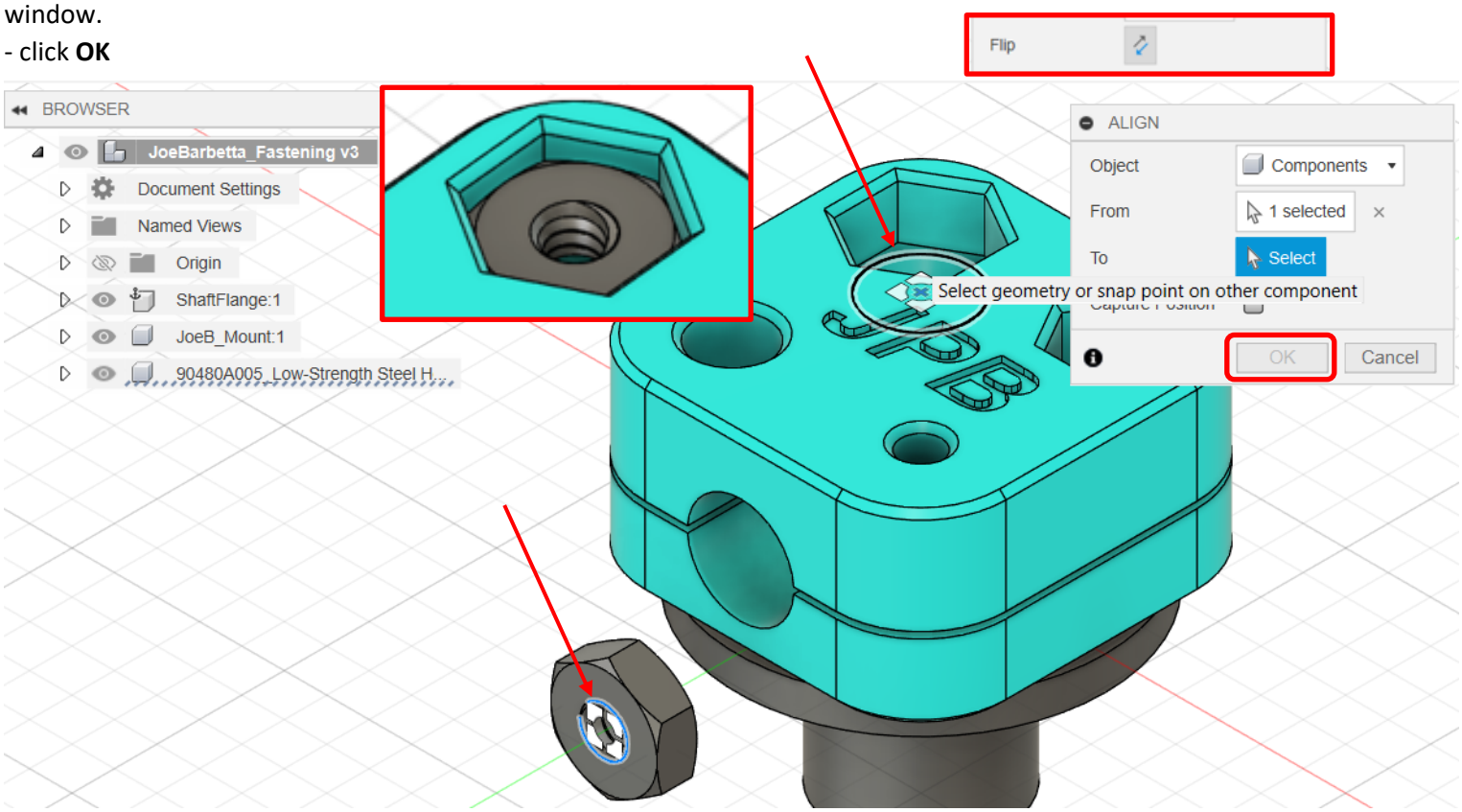
- after moving the Component, click **OK**



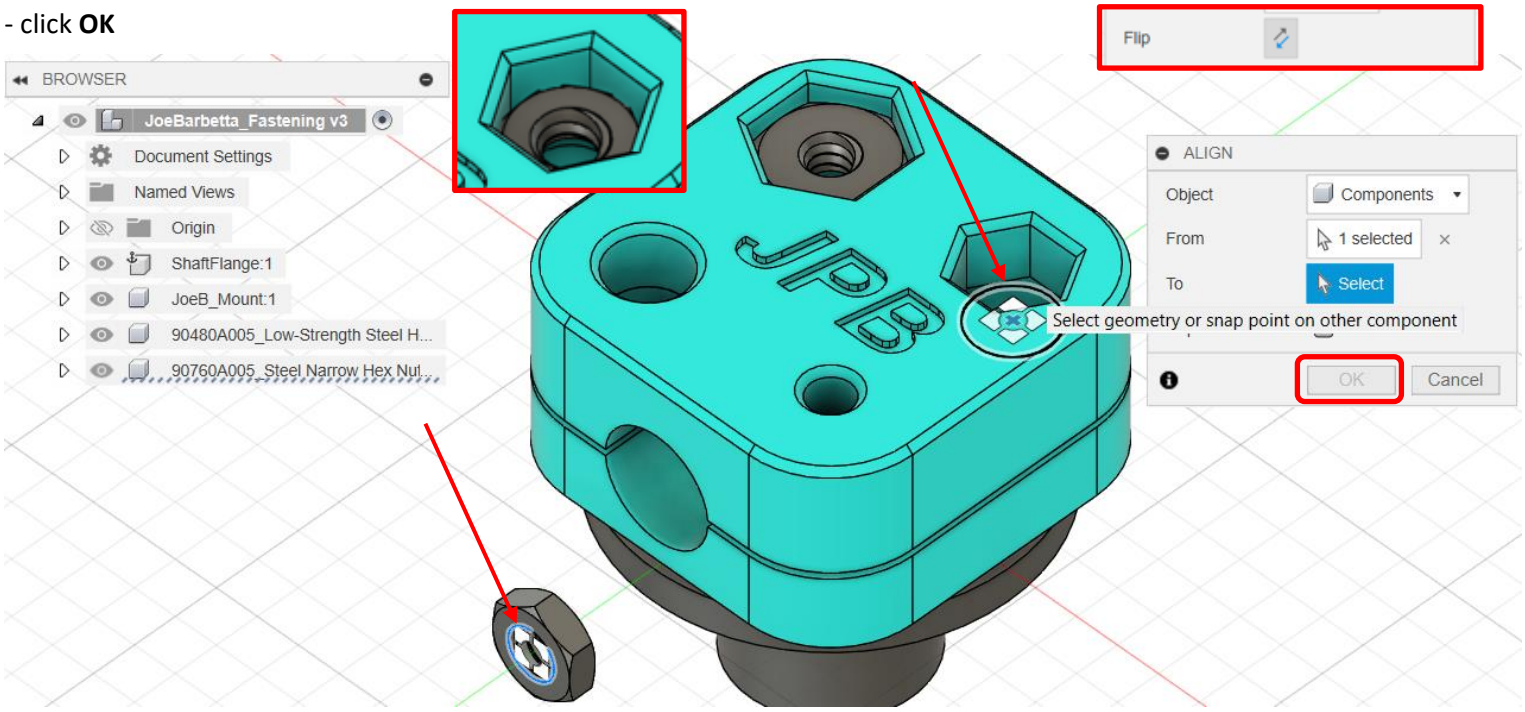
- from the **MODIFY** menu, select **Align**



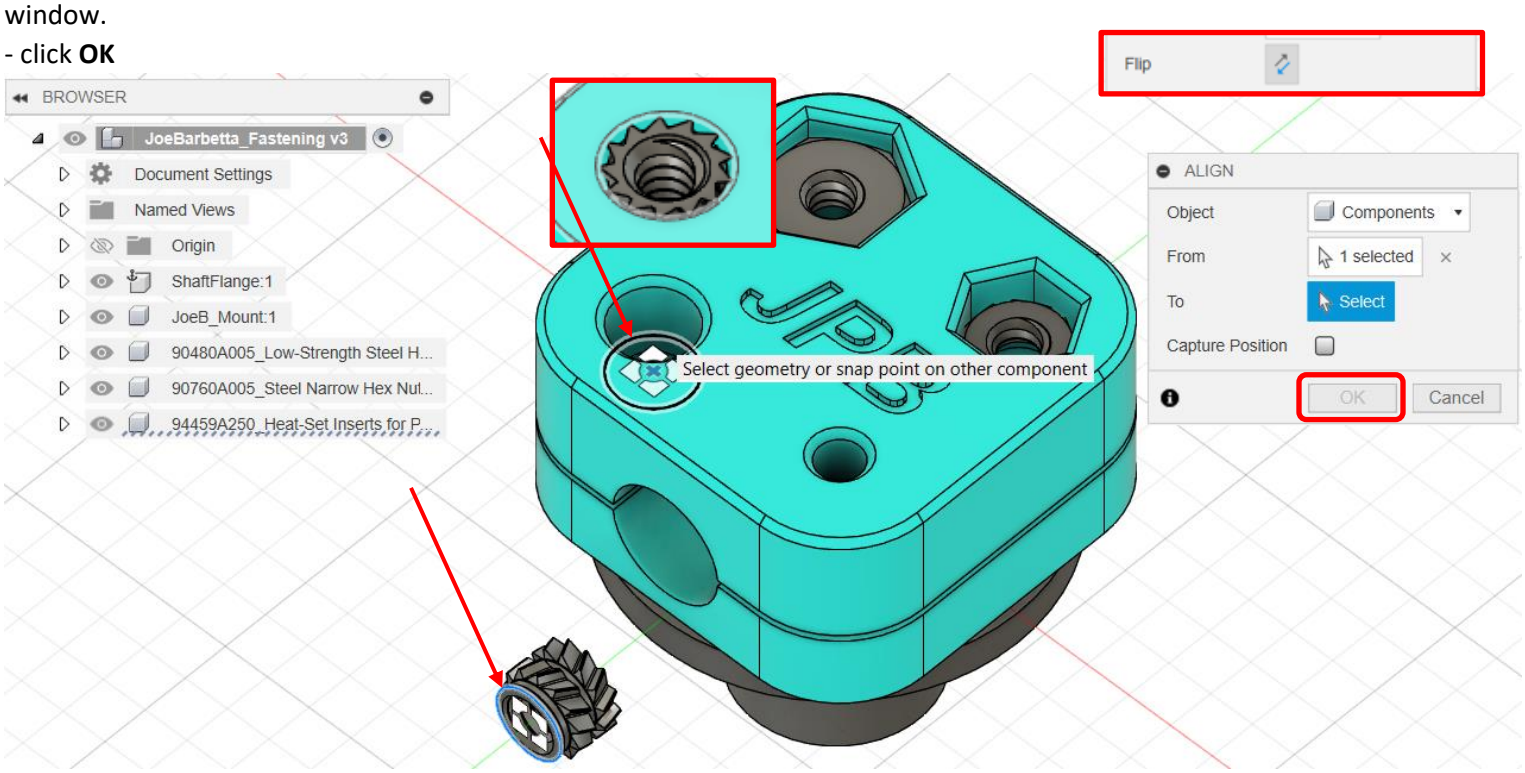
- click on the **edge of the hole of the nut** so that the target icon appears
- click on the **top edge of the chamfer** in the hexagonal pocket, which should cause the nut to jump into the pocket, as shown in the inset image. If the nut seems deeper than that in the inset picture, click on the **Flip icon** that will appear in the ALIGN window.
- click **OK**



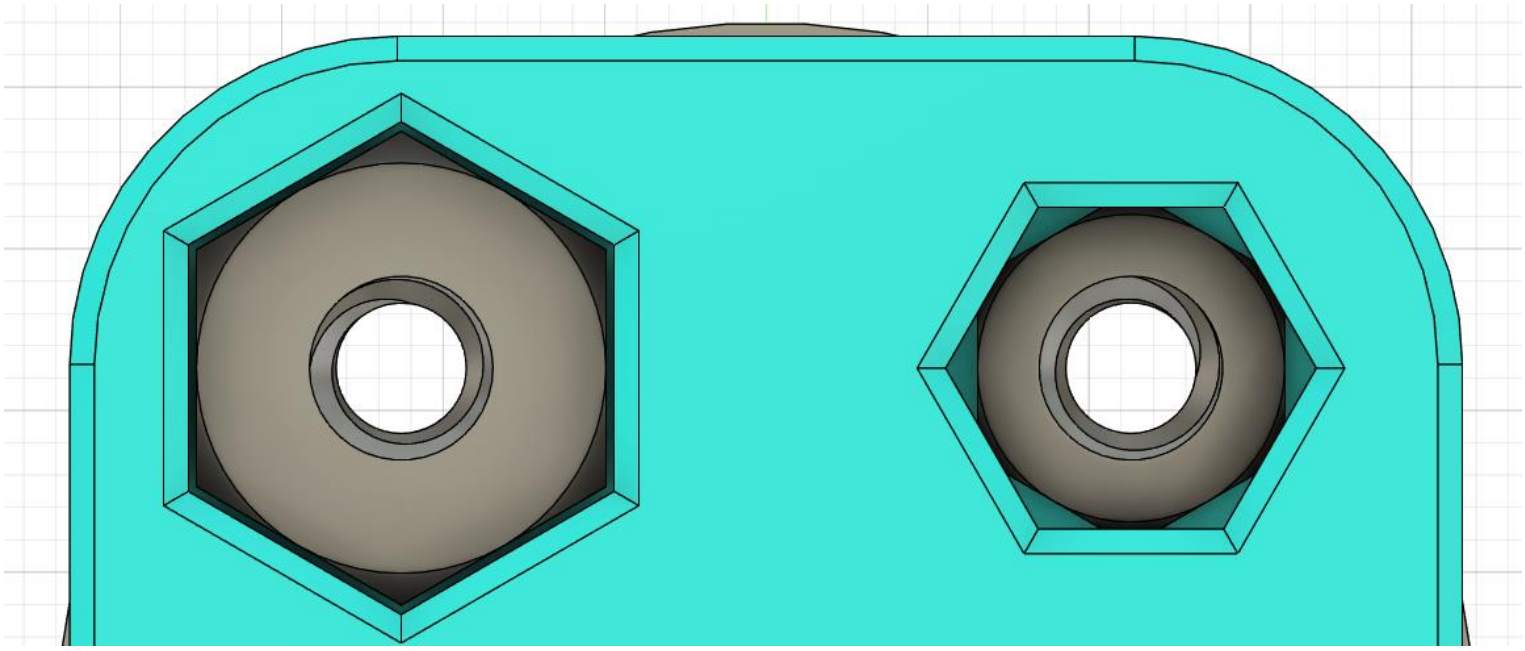
- perform the same **Insert McMaster-Carr Component** and **move** operation to import the part number **90760A005**
- use the **View Cube** to adjust the view to allow access to the top edge of the chamfer in the small hexagonal hole
- click on the **edge of the hole of the nut** so that the target icon appears
- click on the **top edge of the chamfer** in the hexagonal pocket, which should cause the nut to jump into the pocket, as shown in the inset image. If the nut seems deeper than that in the inset picture, click on the **Flip icon** that will appear in the ALIGN window.
- click **OK**



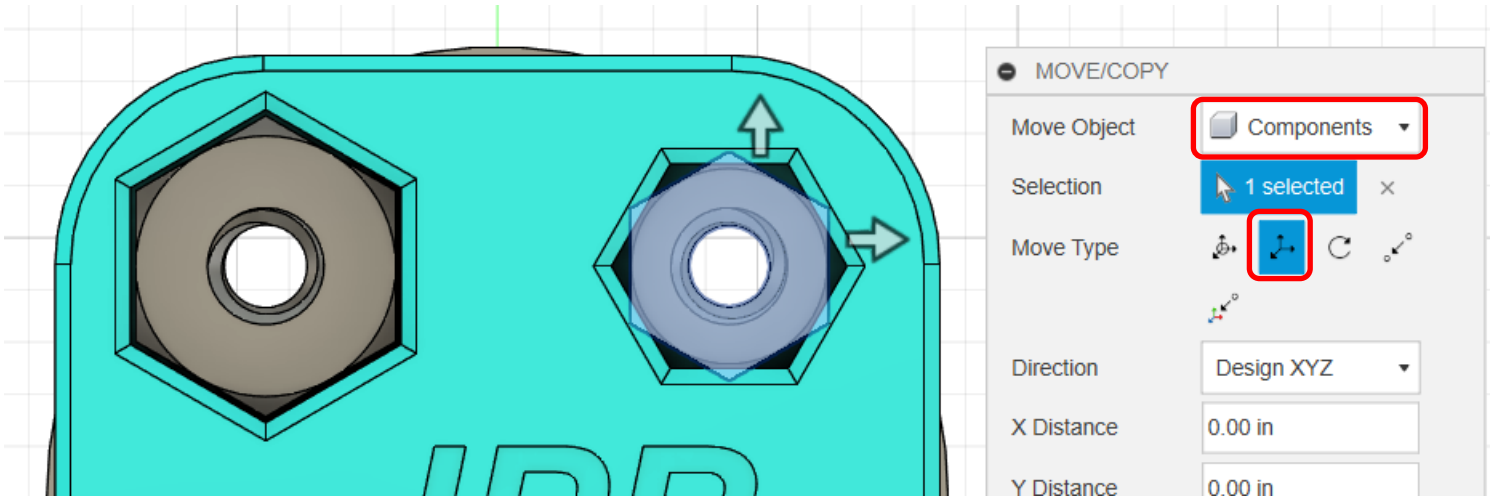
- perform the same **Insert McMaster-Carr Component** and **move** operation to import the part number **94459A250**
- click on the **edge of insert** so that the target icon appears
- click on the **top edge of the chamfer** in the hexagonal pocket, which should cause the nut to jump into the pocket, as shown in the inset image. If the nut seems deeper than that in the inset picture, click on the **Flip icon** that will appear in the ALIGN window.
- click **OK**



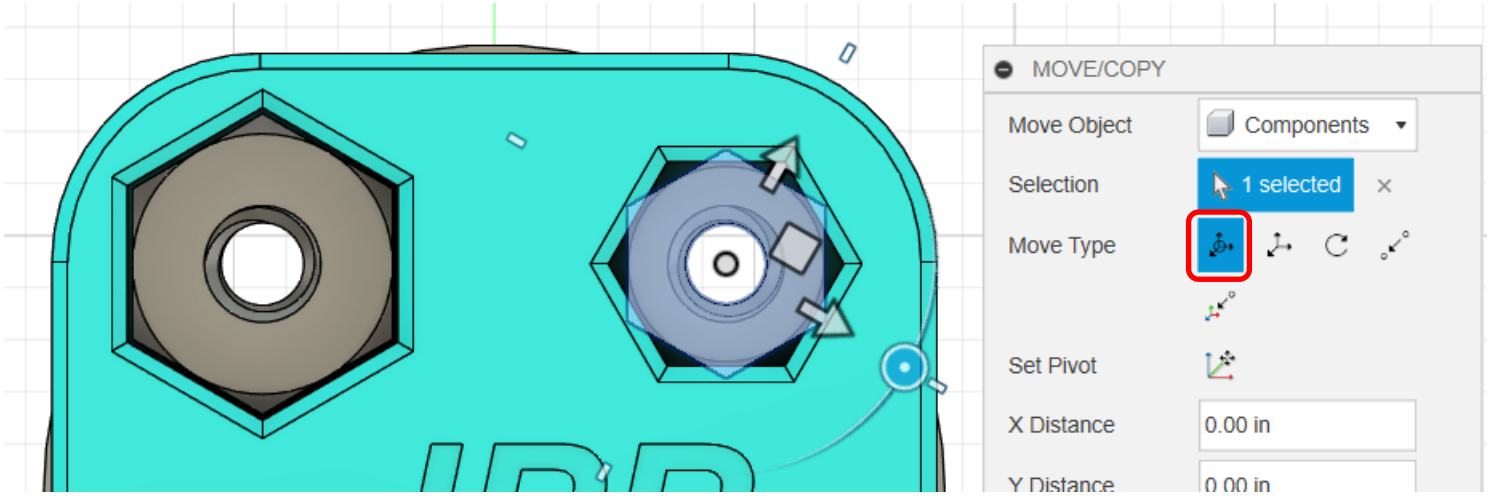
- click on the **TOP** face of the **View Cube** and zoom into the top of the mount body
- if there is a nut that needs to be rotated to align with its pocket, as is the case with the right nut here, use the rotate operation steps on the next page



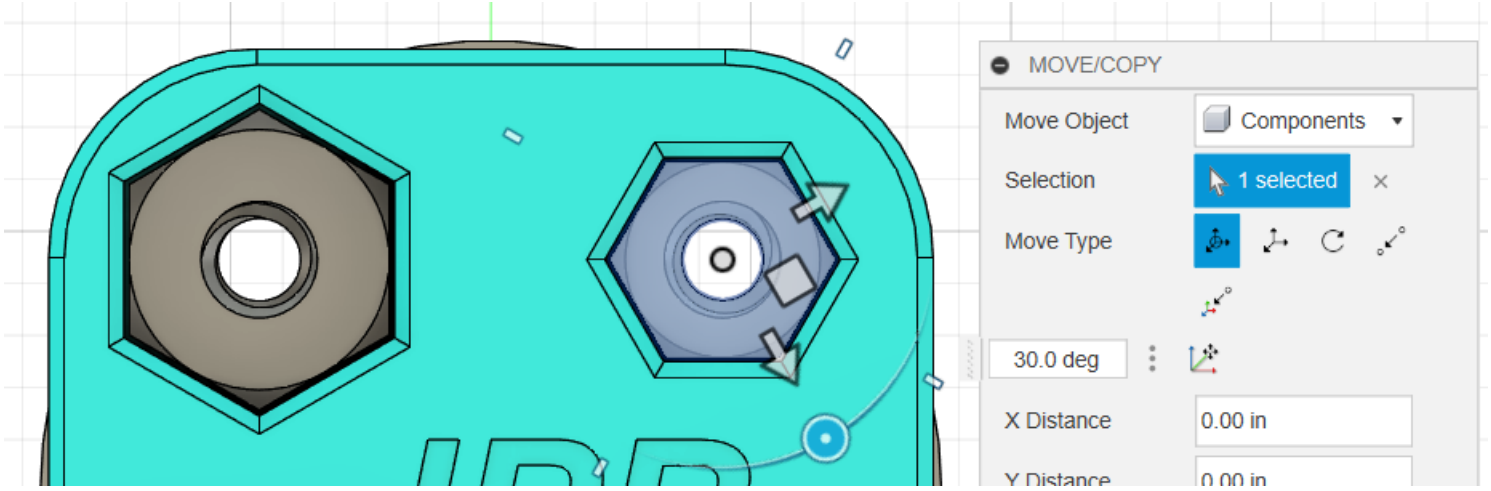
- from the **MODIFY** menu select **Move/Copy**
- ensure the **Move Object** is set to **Components** and click on the **Translate icon**
- click on the nut that needs to be rotated



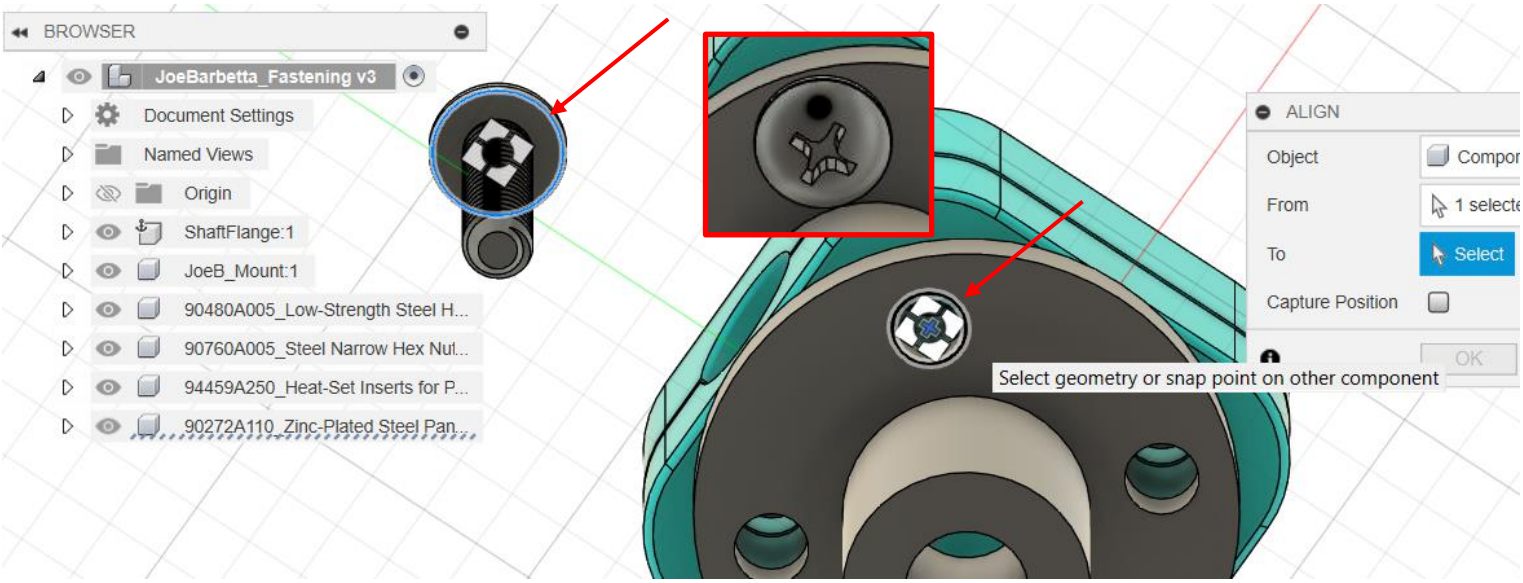
- click on the **Free Move** icon to allow the rotation arc to show
- drag the **rotation circle clockwise** until a value of **30.0 deg** shows, as shown on the next picture



- click on the **OK** button at the bottom of the MOVE/COPY window

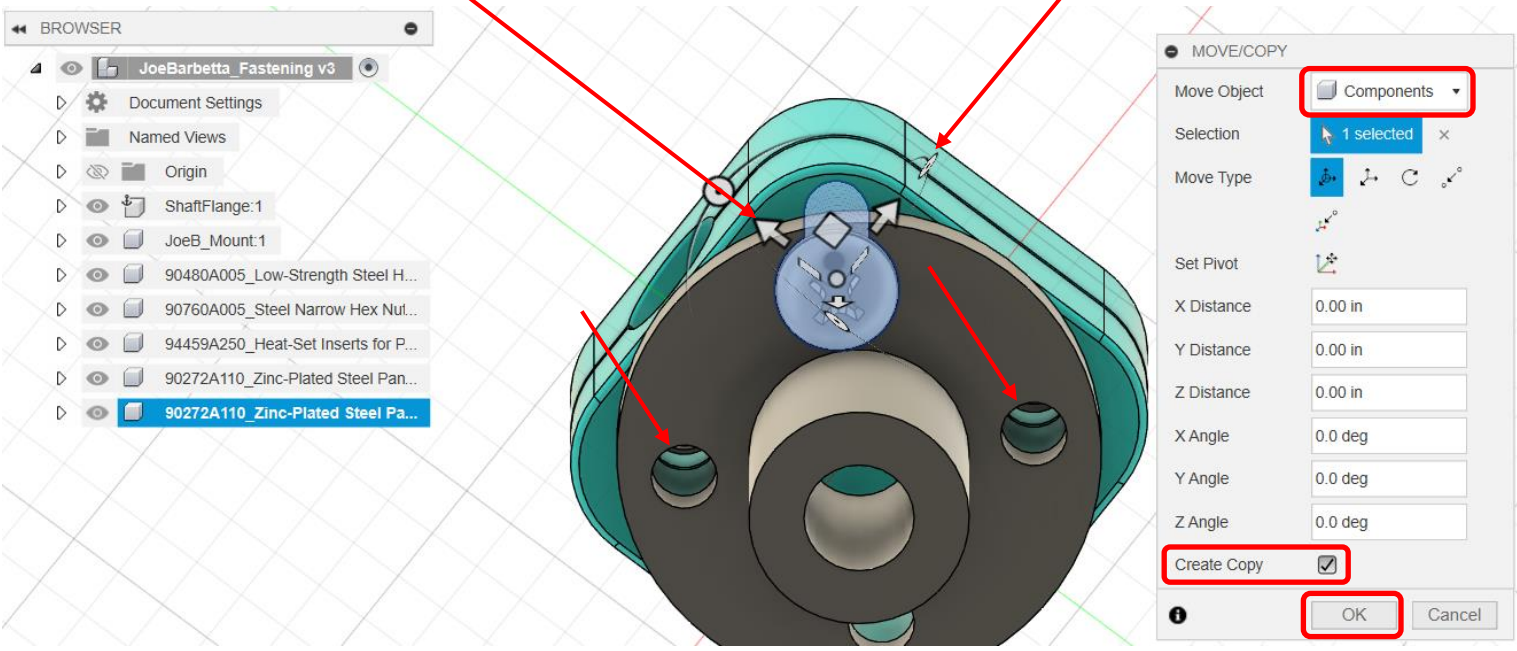


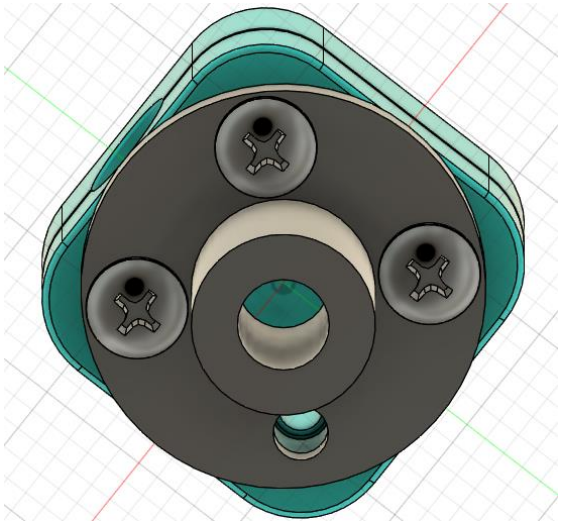
- perform the same **Insert McMaster-Carr Component** and **move** operation to import the part number **90272A110**
- yell **"I am tired of inserting McMaster-Carr components!"**
- use the **View Cube** to adjust the view to allow access to the bottom of the flange
- click on the **bottom edge of the screw head** and then on the edge of the hole, which should cause the screw to jump into the hole, as shown in the inset image. If the head is not visible, click on the **Flip icon** that will appear in the ALIGN window.



We need 2 more of the screws just inserted. As an alternative to using the import component feature, we will duplicate an already inserted screw.

- from the **MODIFY** menu select **Move/Copy**
- ensure the **Move Object** is set to **Components**
- click on **Create Copy** check box near the bottom of the MOVE/COPY window
- drag one of the **arrows** to move the copy of the screw off to the side and then use one of the **rotation arcs** to access the bottom of the head
- click **OK**
- use the **Align** tool as done before to align the screw with a hole indicated with an arrow
- perform these operations again to align a 3rd screw with the other hole indicated with an arrow

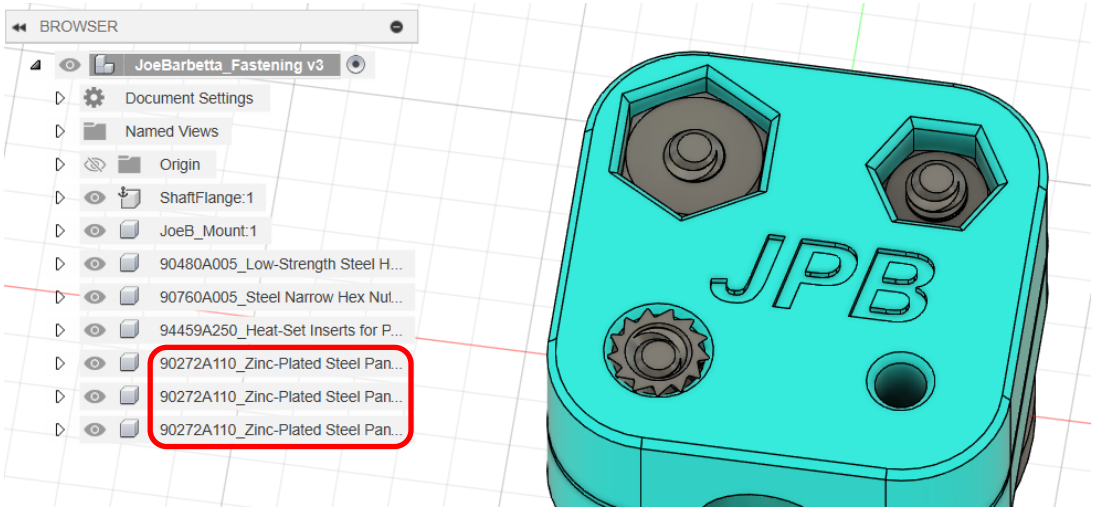




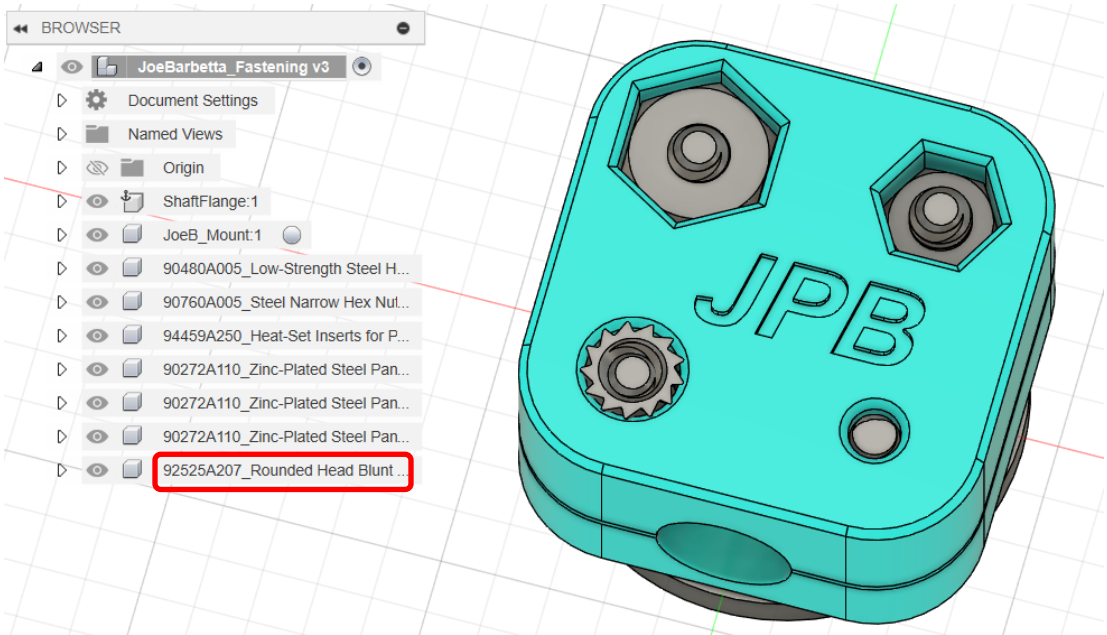
This is the result after 3 screws have been aligned.

Note that a screw was not added to the bottom hole yet because this hole needs a **self-tapping screw**. The screws just added were **machine screws**.

- return to the top of the mount to view the screws. Note how the hole without a nut or insert is empty. This is the hole for the self-tapping screw. The 3 bottom parts in the BROWSER are the machine screws.

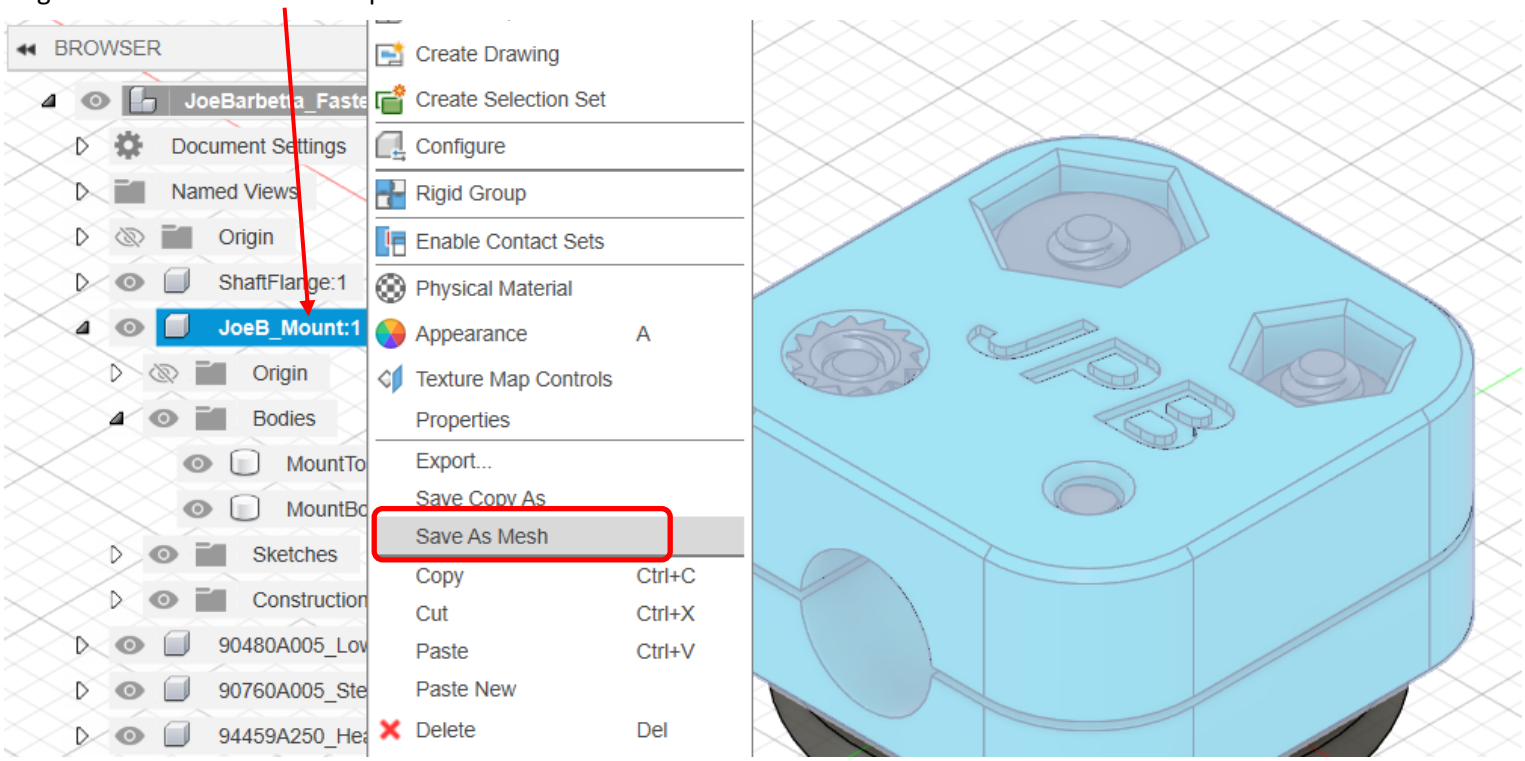


- perform the operations, as shown a few pages earlier, to insert a new McMaster-Carr component **92525A207** and align the screw with the last avaiable hole in the flange. The top view should then look like that below. Note the new part that will show in the BROWSER.

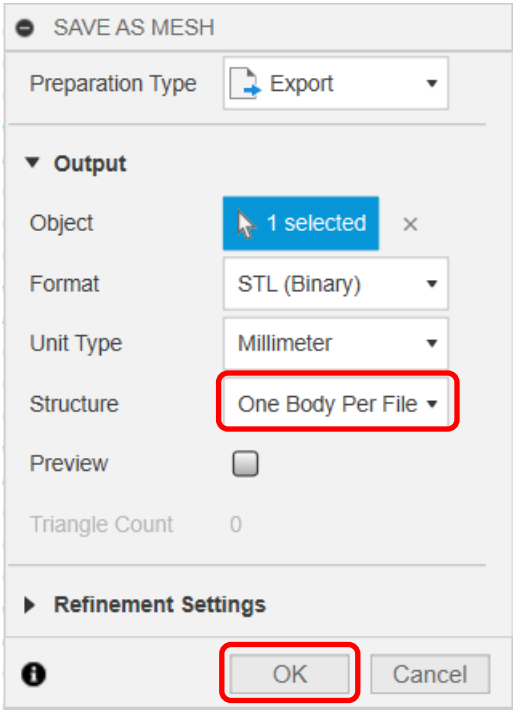


Exporting the STL Files

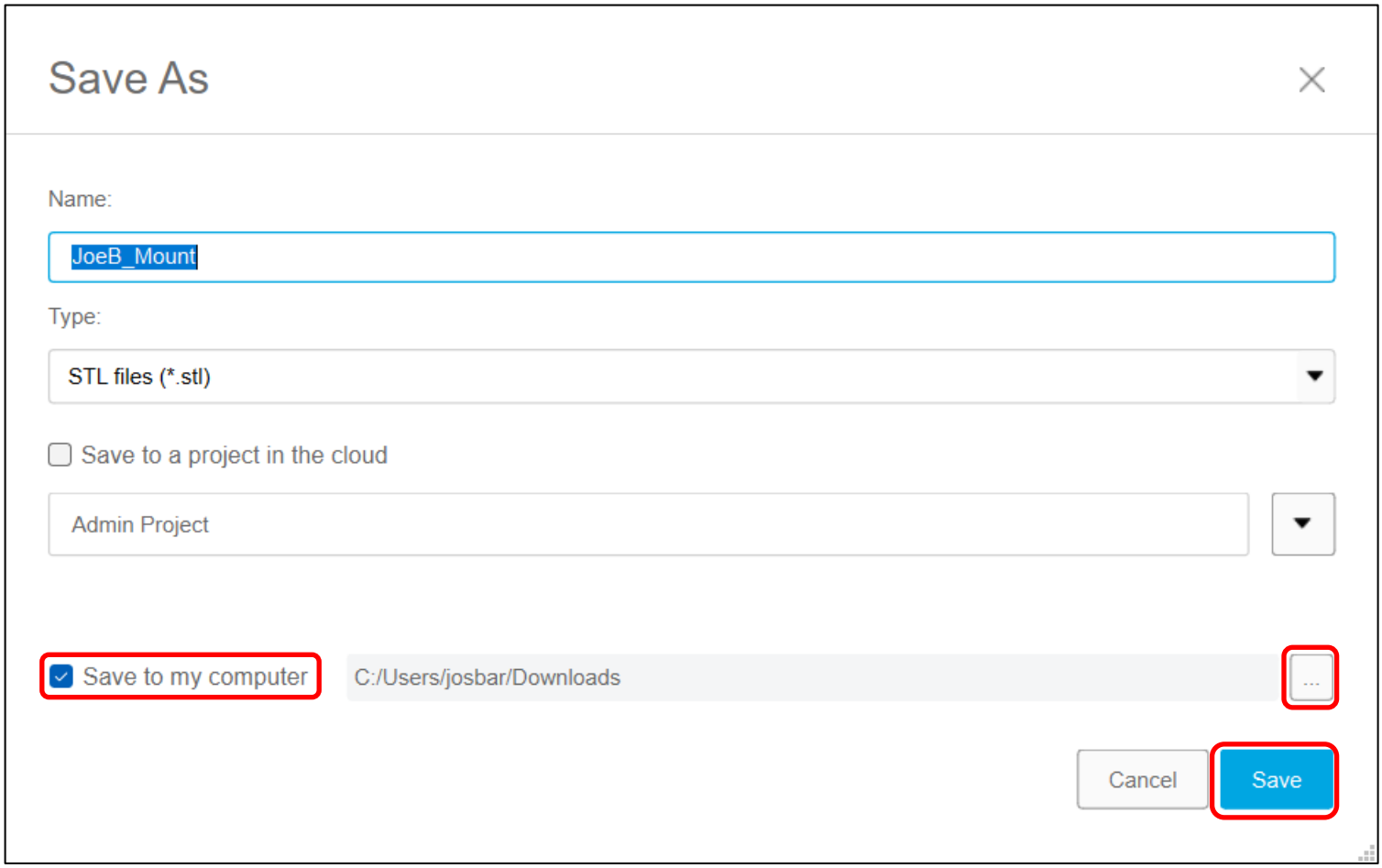
- right-click on the **Mount** component and select **Save As Mesh**



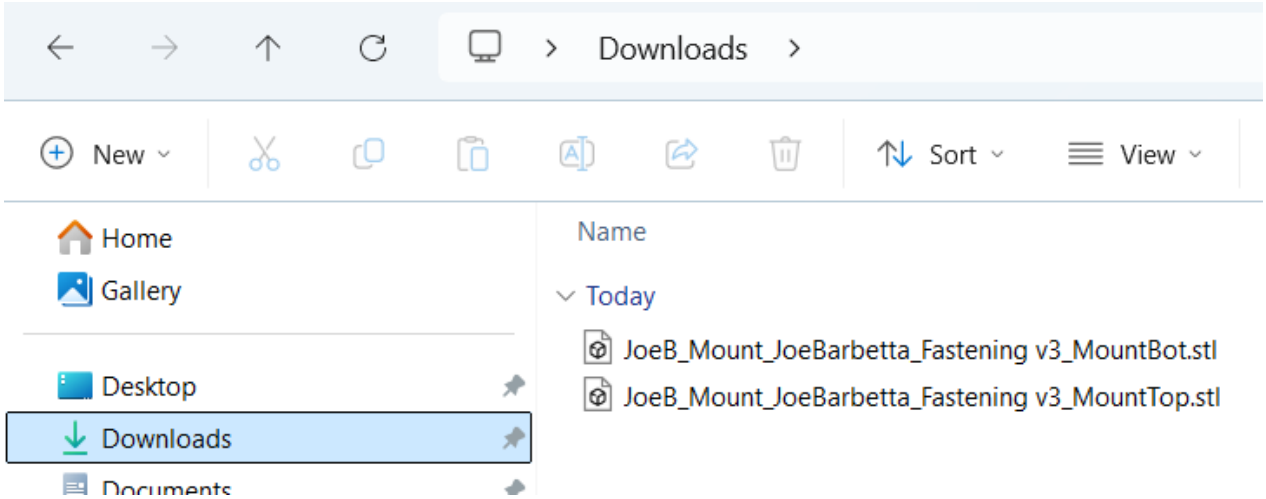
- set Format to **STL (Binary)**, Unit Type to **Millimeter**, and Structure to **One Body Per File**
- click **OK**



- ensure that **Save to my computer** is checked and note the save location of the file. By default it should be the **Downloads** folder. If desired it can be changed using the button with 3 dots after the file loacation bar.



- verify that 2 files were created in your Downloads folder



Viewing the Parts in Cura

This is the view of the two parts opened in Cura.

They could be printed this way and the top of the semicircle cutout will be slightly distorted. This could still work.

The lower picture shows the rotation tool in Cura being used to flip the left part. This is an option to allow the best printing for tha channel.