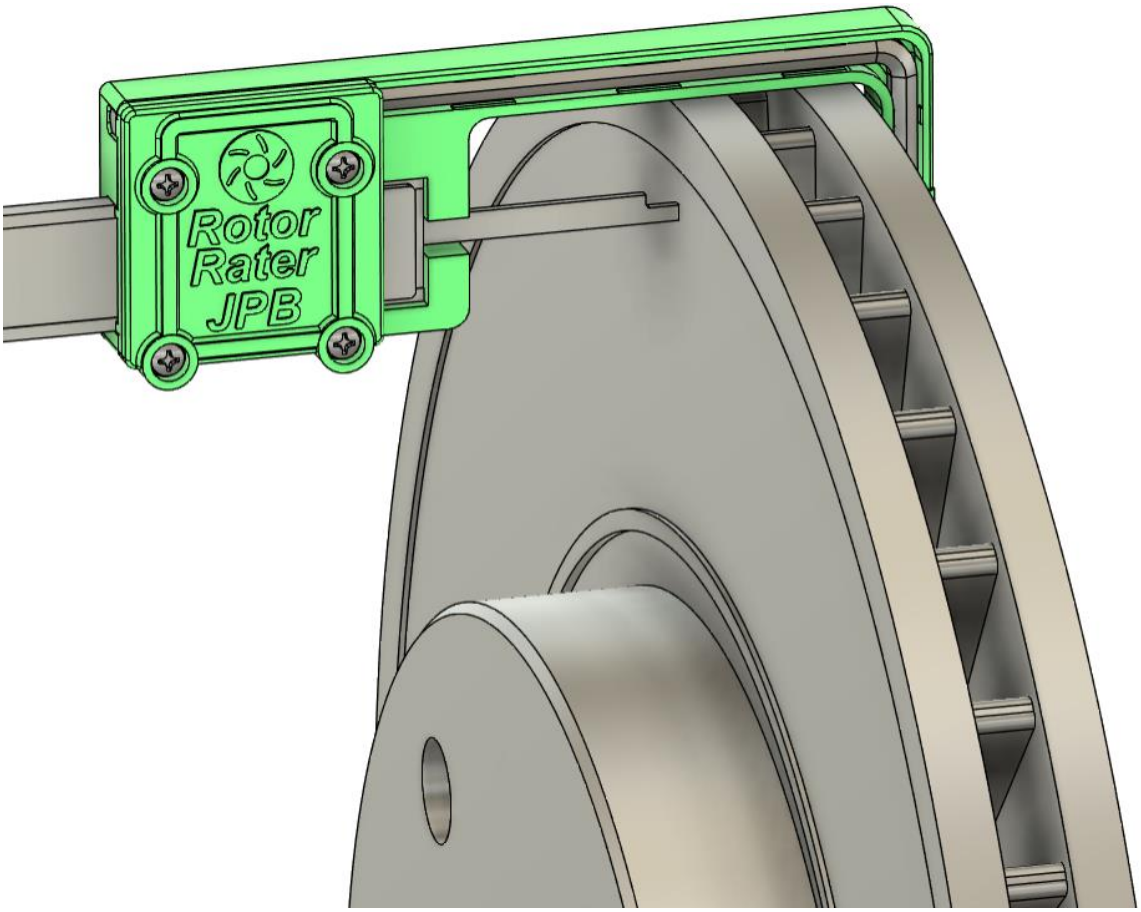
Mr. Barbetta's *"I don't care about 3D Printing. I just want the art credit."* class

Are you tired of being told you need new rotors?

Design a Rotor Rater



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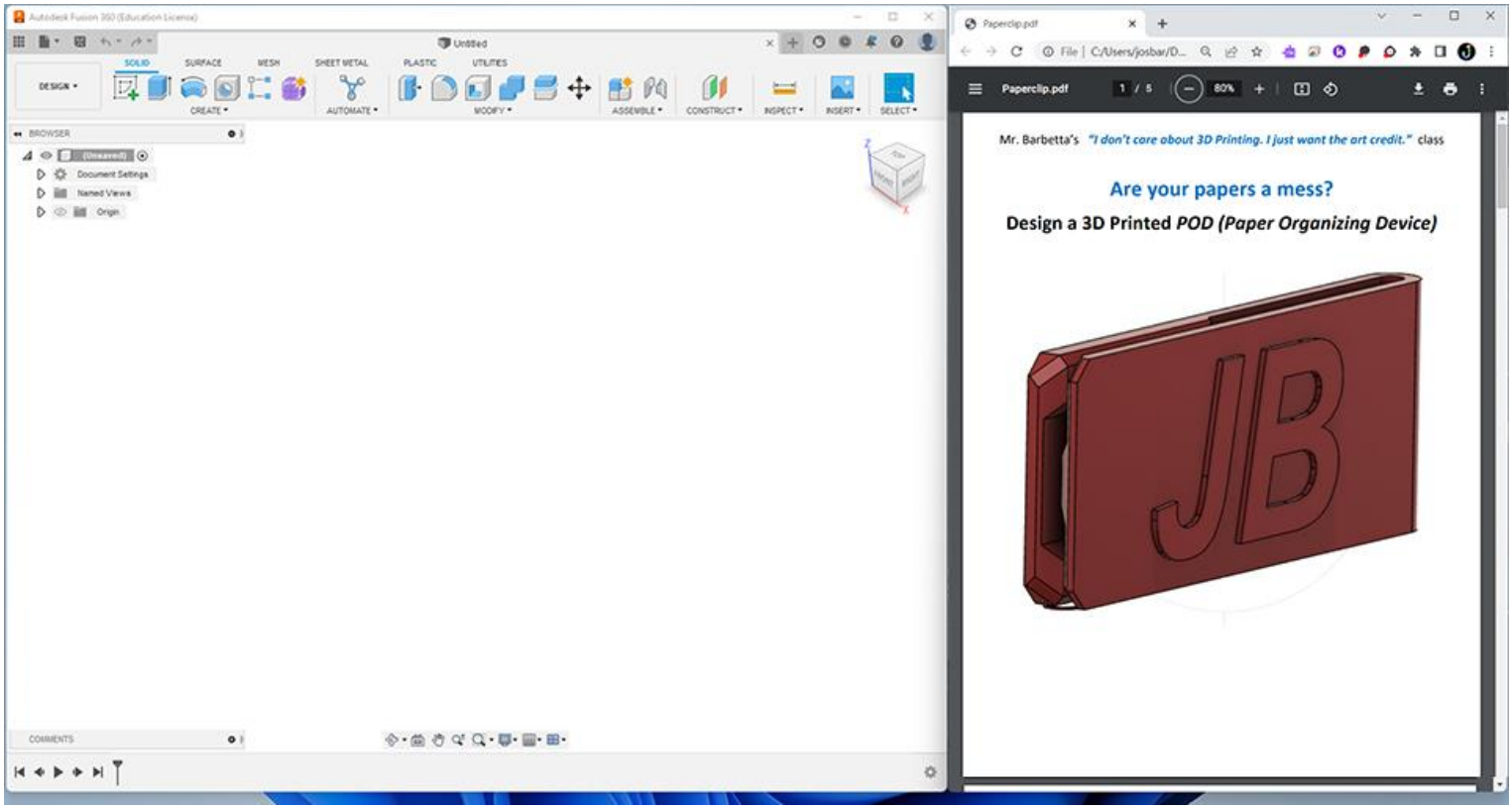
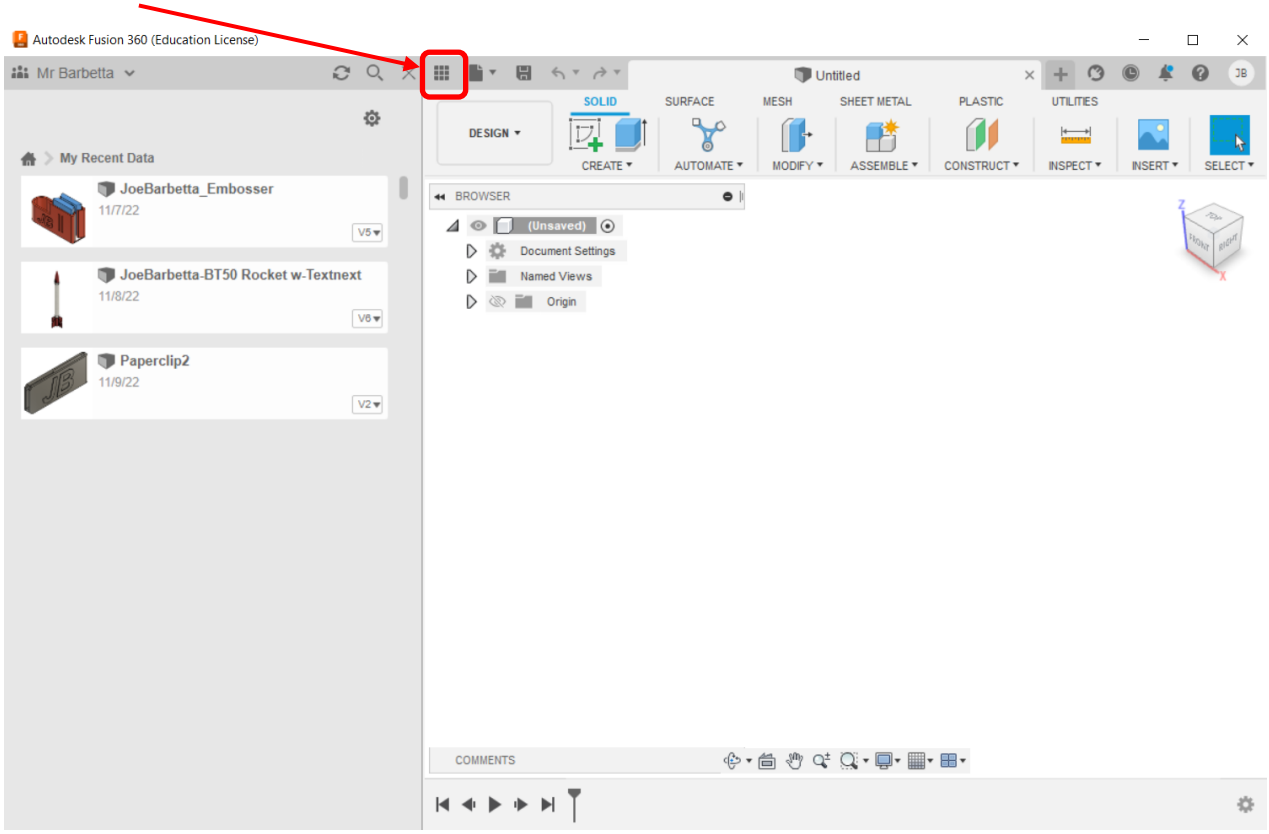
Contents

Using This Document

The best way to follow this document is to **reduce the width of the Fusion window** and have this pdf document open in Chrome browser as shown below. This document can be **downloaded from Schoology and then dragged into Chrome** and scaled down to 80%.



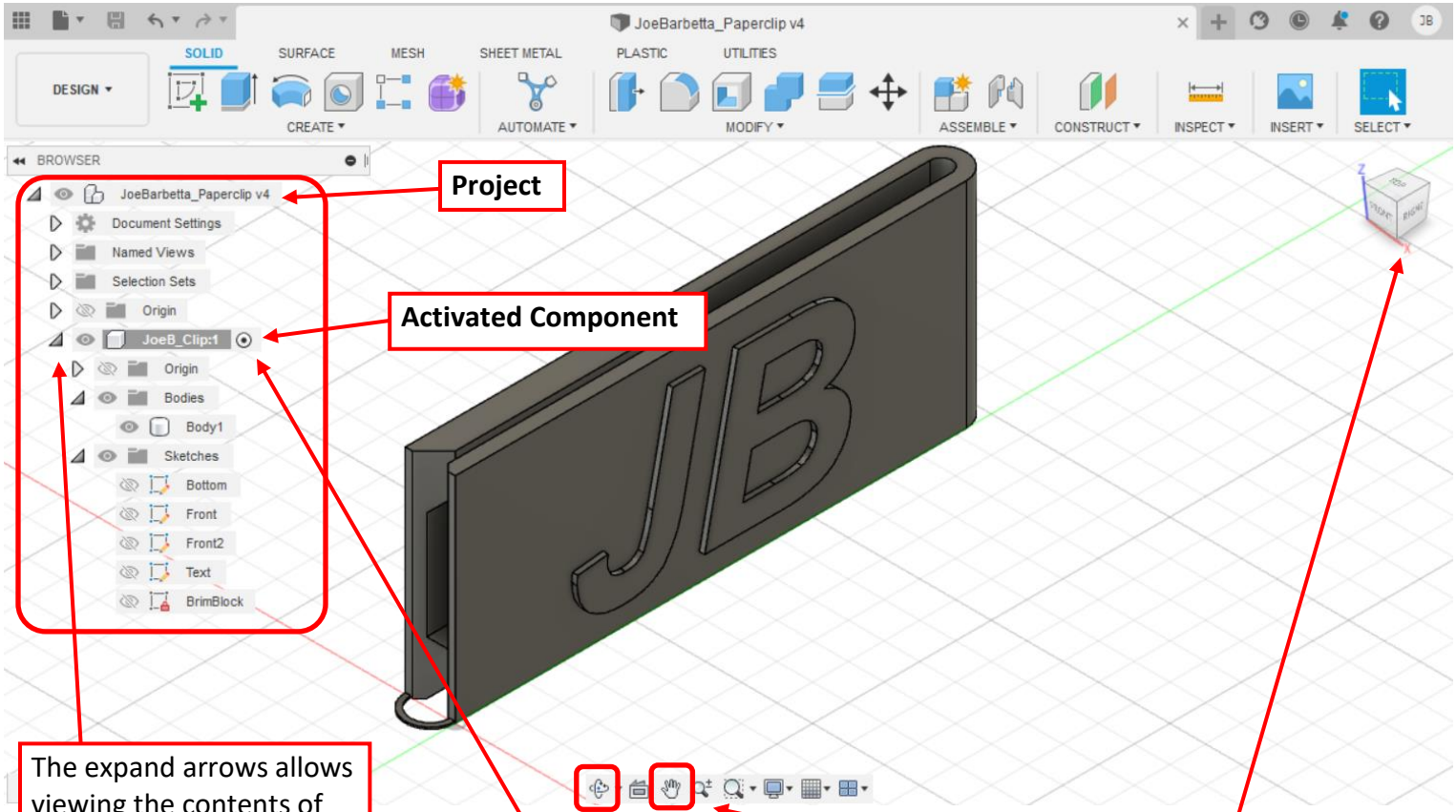
The Fusion window will not allow its width to be reduced much so for smaller computer screens a trick is to click on the **Data Panel icon** and then move the window to the left with the Data Panel off the screen.



The Fusion User Interface

Review this page and the following for information on the user interface.
The actual instructions begin on page 7 (Starting a Design in Fusion).

This is what the final paperclip will look like in Fusion See comments after this screen shot.



The expand arrows allows viewing the contents of *Components* and *Bodies* and *Sketches* folders.

Note the **circled dot** of the *Activated Component*. When one wants to work on a Component, it must be Activated by right-clicking on the Component Name and selecting **Activate** at the top of the pop-up menu.
When a project is reopened, the Project at the top of the Browser is Activated and a Component may have to be Activated again.

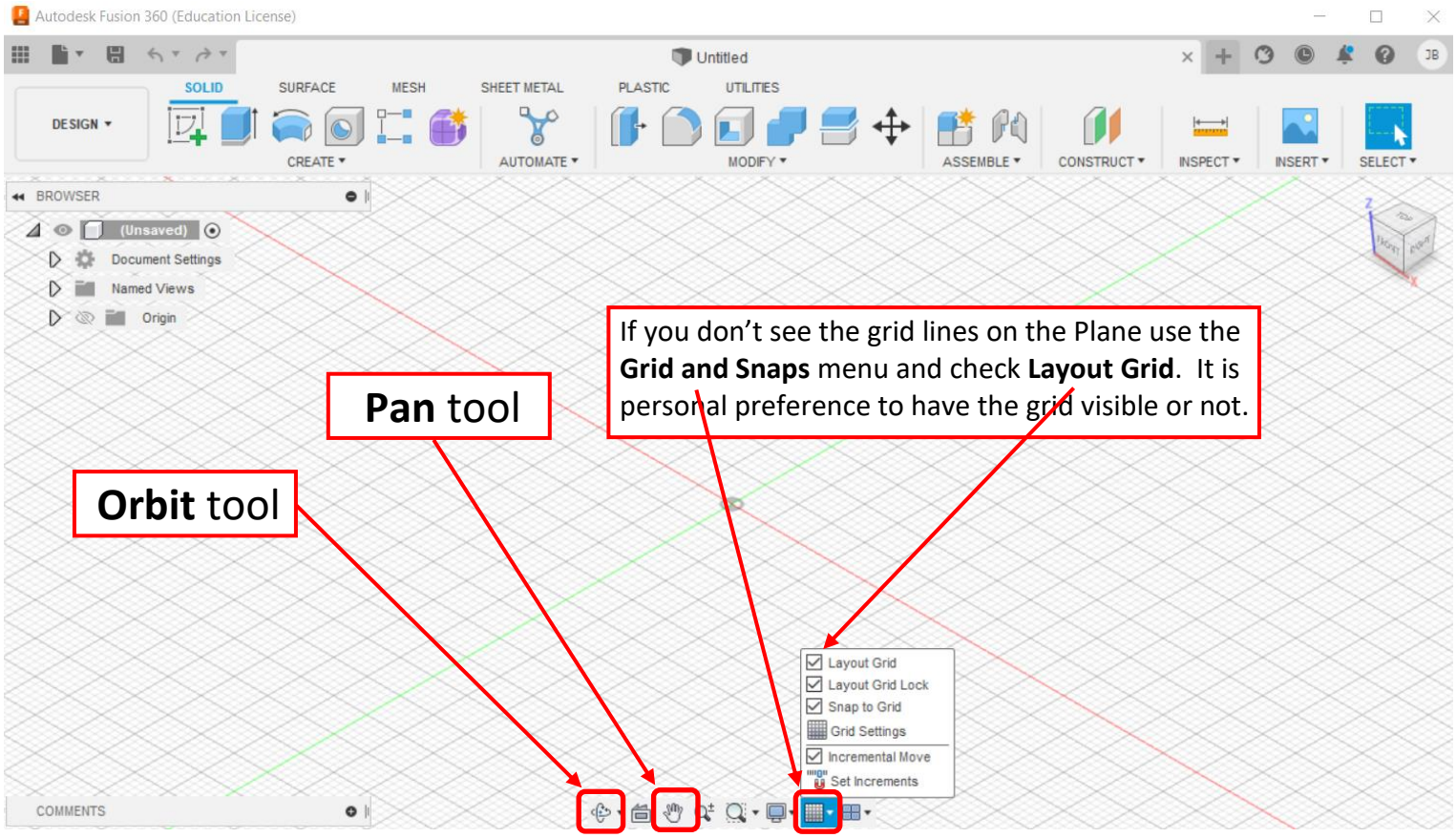
The **Orbit** and **Pan** (hand icon) tools are commonly used to adjust the view of the design. After using one the **Esc key** is needed to exit the operation.
The top right **View Cube** is another method to rotate the view of the design.

The **mouse wheel** can be used for zooming in and out. The **mouse wheel** can alsoe be pressed to pan.

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

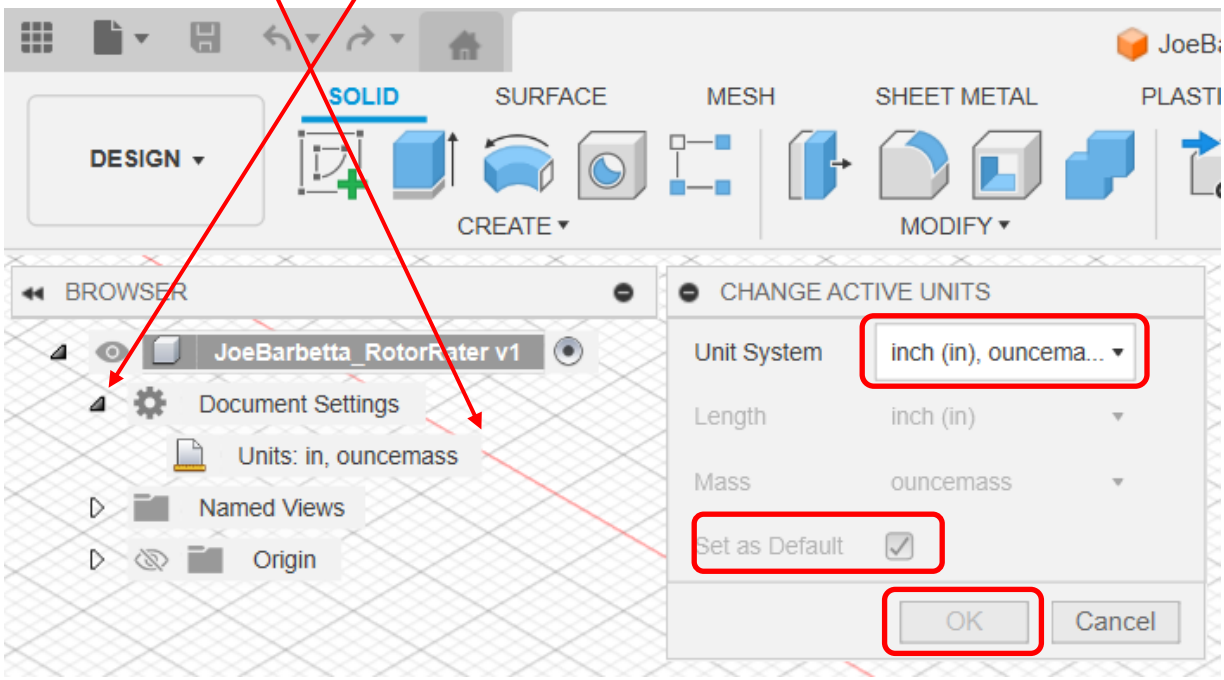
Starting a Design in Fusion

- 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 **_RotorRater** e.g. **JoeBarbetta_RotorRater** (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 can be opened using "**Open…**" from the top **File** icon.

As you work you may want to occasionally save your work in case Fusion 360 crashes.

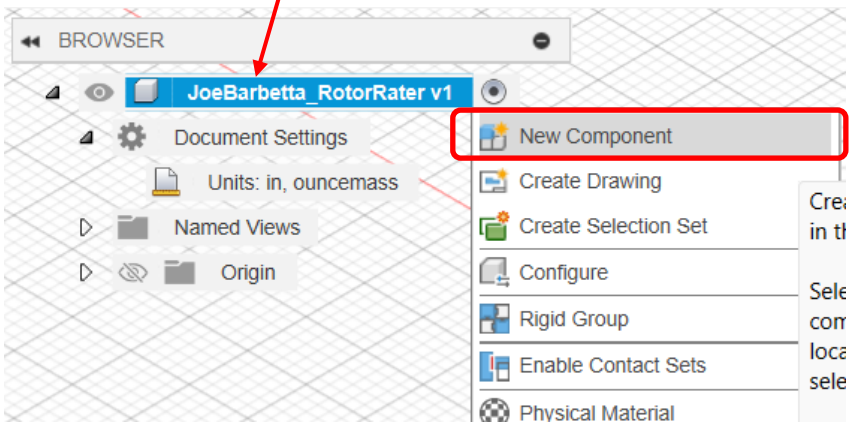
- in the left **"BROWSER"** click the **arrow next to Document Settings**
- click on the **edit icon** that appears to the right when you hover over **Units**
- ensure **Active Units** are set to **Units: in, ouncemass** and click **OK**. You can also enable **Set as Default** if it is not grayed out.

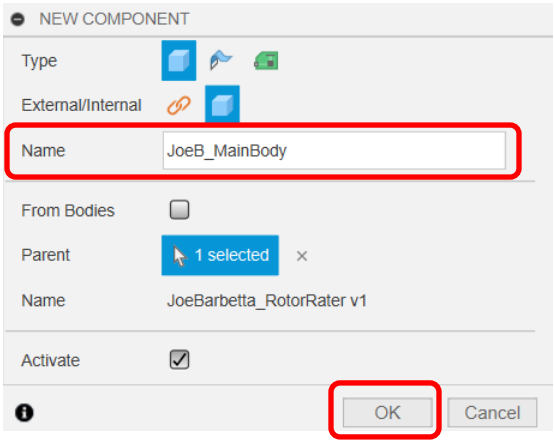


Creating a New Component

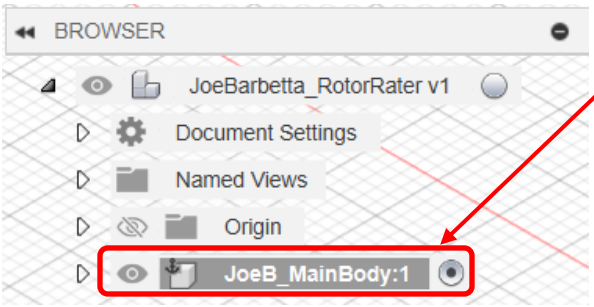
To keep a project organized it is recommended to create a new Component for each part.

- right-click on the **Project Name** at the top of the BROWSER and select **New Component**
- set **Name** as **Your first name and last name initial followed by _MainBody** e.g. **JoeB_MainBody** and click **OK**



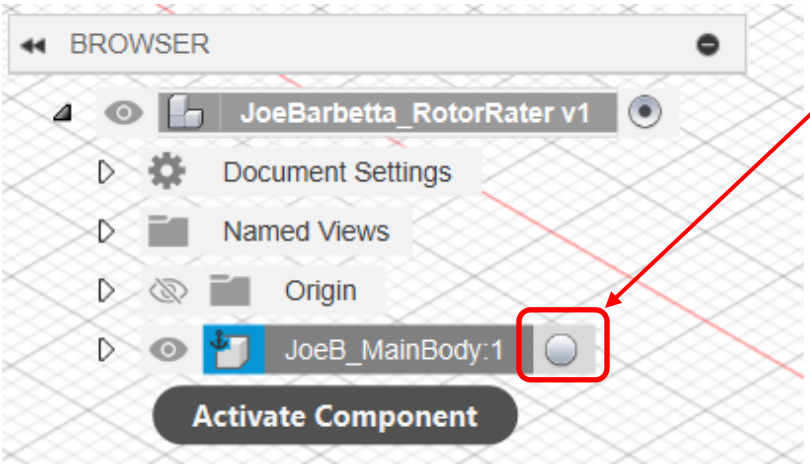


The new *Component* should show in the *Browser*.
Note that the area around the name is darkened and the circle with the dot. This indicates that it is the ***Active Component***.



One reason to hate Fusion!

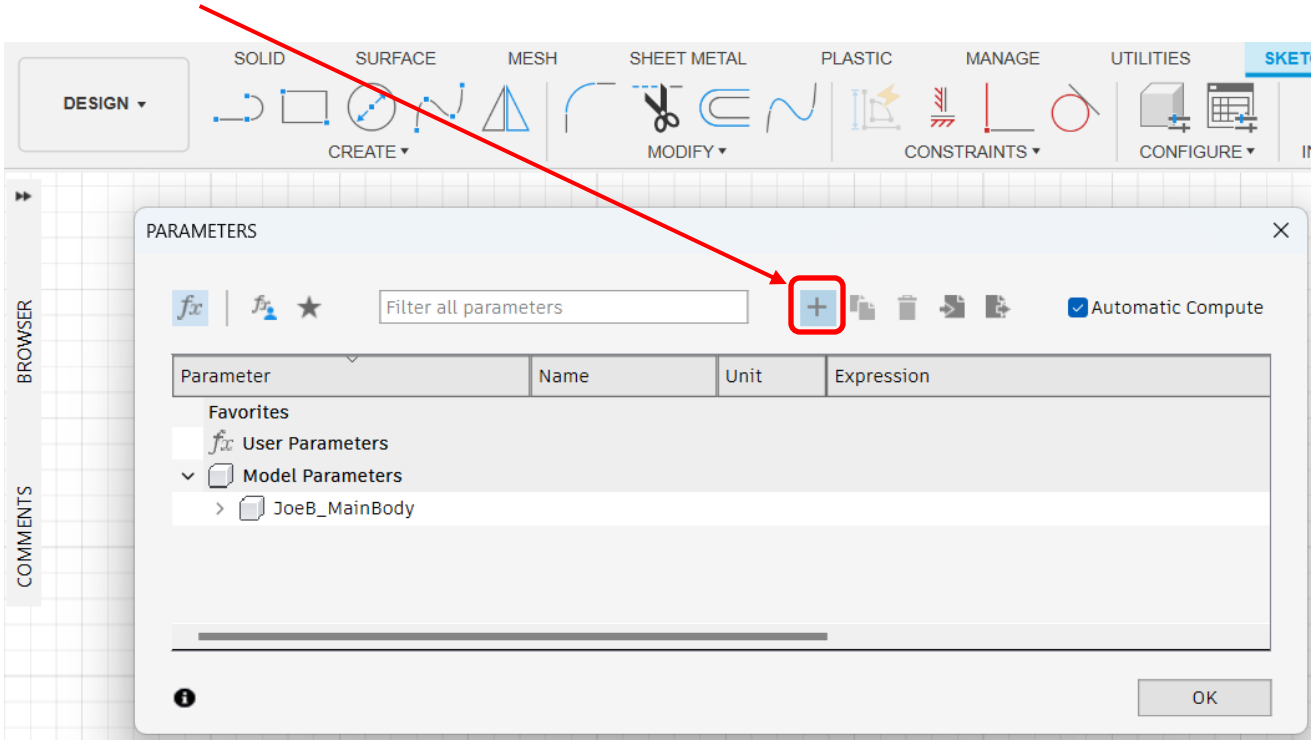
When a project is saved and then reopened, the Component that was Active is no longer Active.
If one wants to continue working on a Component, it must be reactivated by **clicking on the circle** o the right of the Component. Alternatively, one can **right-click on the Component Name** and selecting **Activate**.



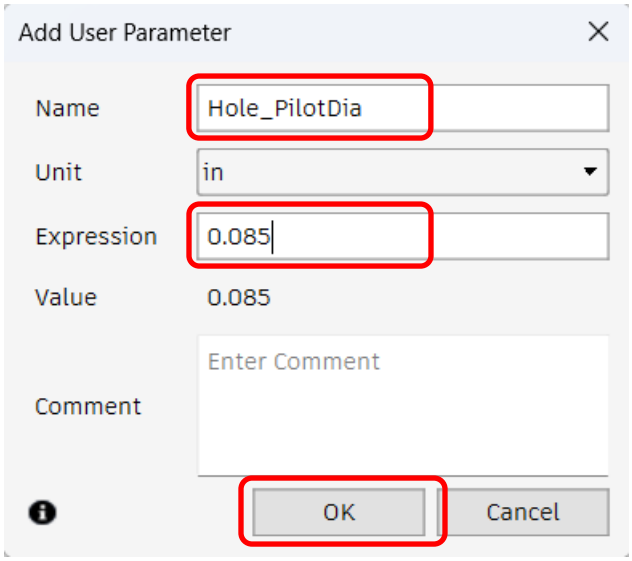
Setting Parameters

We will start by setting parameters for dimensions we may wish to change later on.

- near the bottom of the **MODIFY** menu select **Change Parameters**. If a window pops up about Text Parameters, click its bottom **OK** button to close it. You should see the below window.
- click on the **+ icon**, which should open a small **Add User Parameter** window, shown lower down.



- set the **Name** as **Hole_PilotDia** and set its **Expression** to **0.085**, and click **OK. Do Not click OK on the main window yet.**

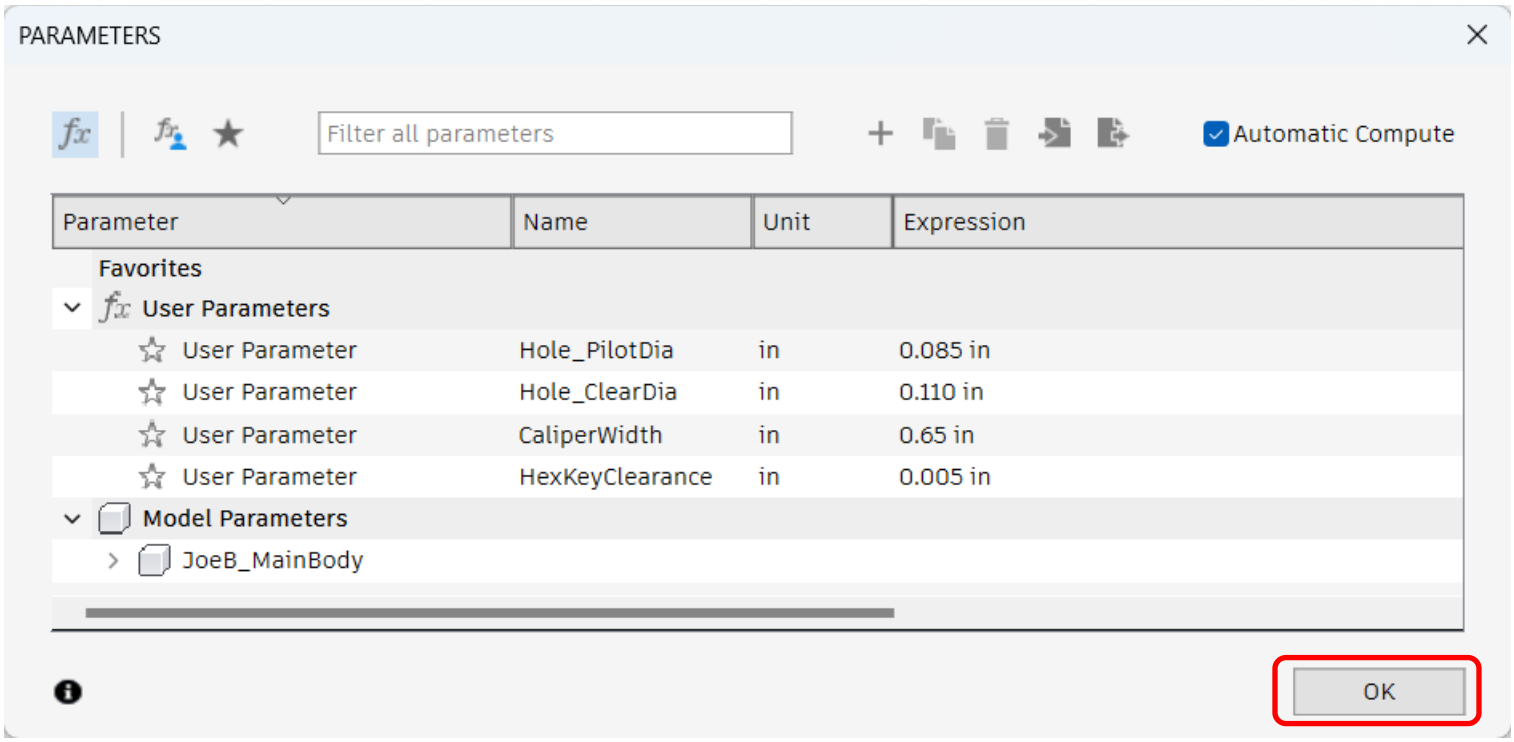


Self-Tapping screws will be used to assemble the Rotor Rater. A Pilot Hole is the hole that the screw will thread in to and it will have a smaller diameter than the screw. The resultant hole in the 3D print can vary from that specified and this value may have to be changed. Note that slicer software, such as Cura, has a few "Horizontal Expansion" settings, which can affect this.
If the pilot hole is too small, it will be difficult to insert the screw and its head may get stripped or the plastic may crack. If the hole is too large, the screw may pull out.
Just like our planets distance from the sun, there is also a "Goldilocks zone" for pilot hole sizes.

- using the **+ icon** again, set a few other parameters

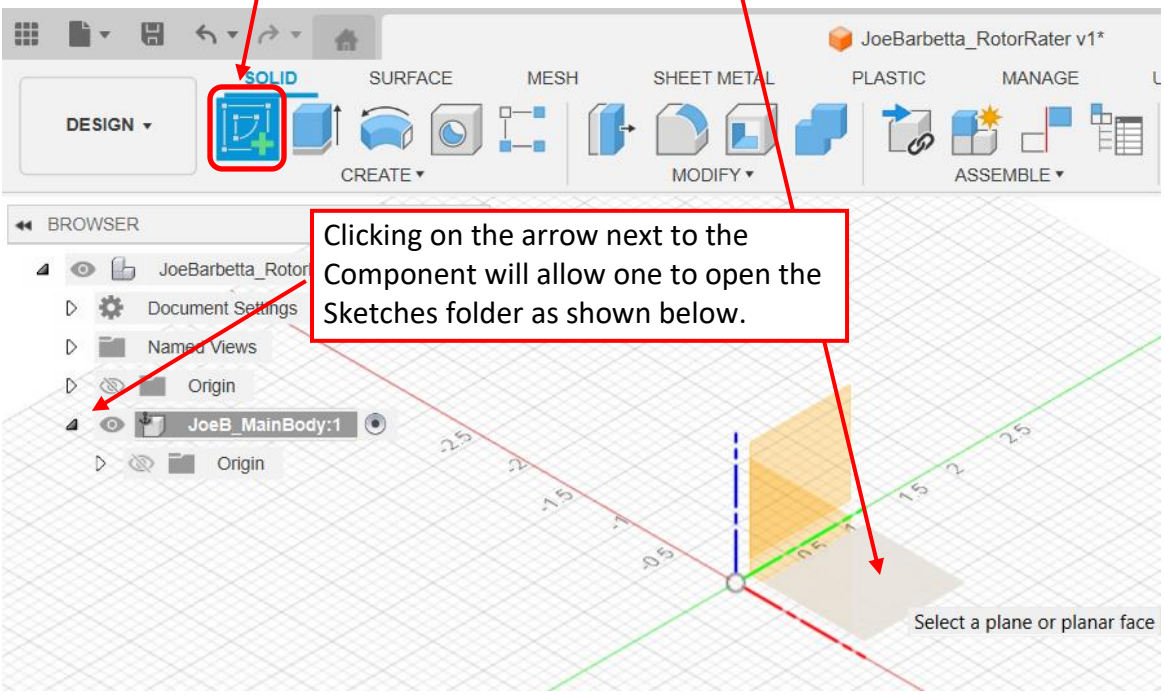
Hole_ClearDia = **0.110**
CaliperWidth = **0.650**
HexKeyClearance = **0.005**

The PARAMETERS window should look like that below.
- click **OK**



Creating the Bottom Sketch

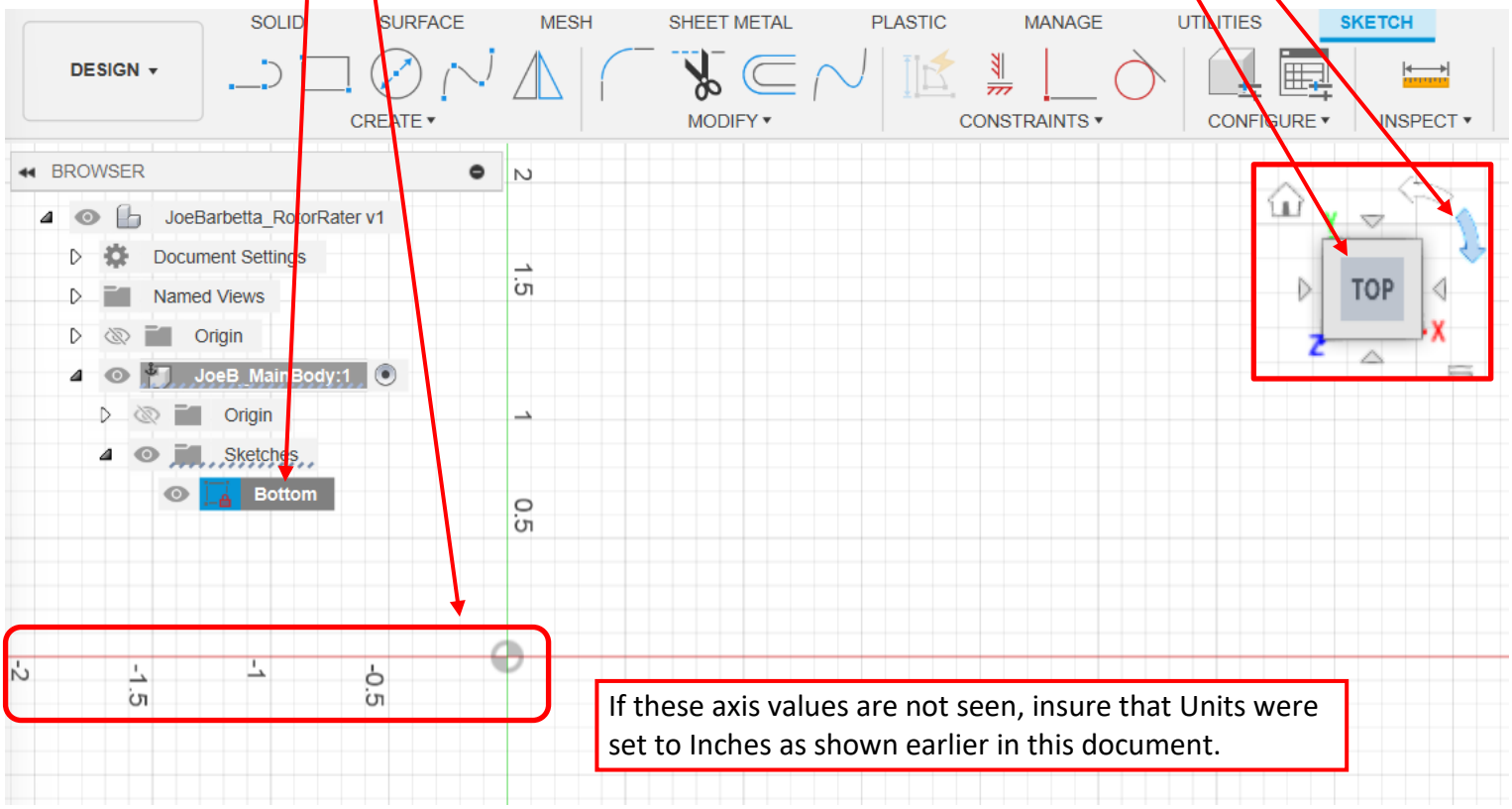
- select the top **Create Sketch** tool and click on the **bottom rhombus** to select the X-Y Plane.
If a tool can't be found, one can always look in the **CREATE** and **MODIFY** menus for it.



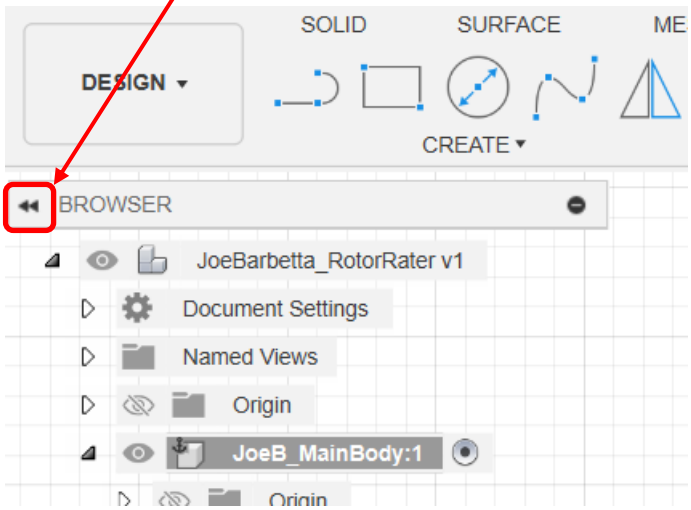
- **zoom in and pan** (hand icon) to achieve a view similar to below. You may have to use the **curved arrow** to rotate the view. If the curved arrows don't show, click on the center of the View Cube.

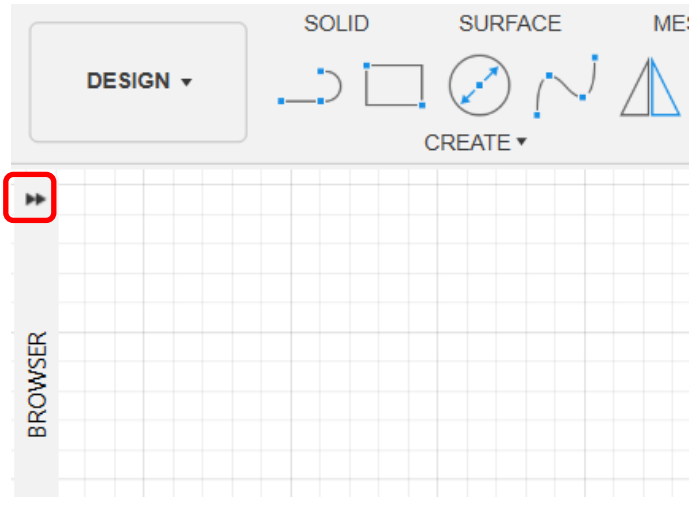
- note the **values on the red axis** as an indication of the desired view and the position of the **View Cube**

- right-click on the **Sketch Name** and select **Rename** and name it **Bottom**



- click on the **Arrows** on the BROWSER to hide it. This gives one extra room to work. As the right picture shows, there are arrows to reopen the BROWSER.

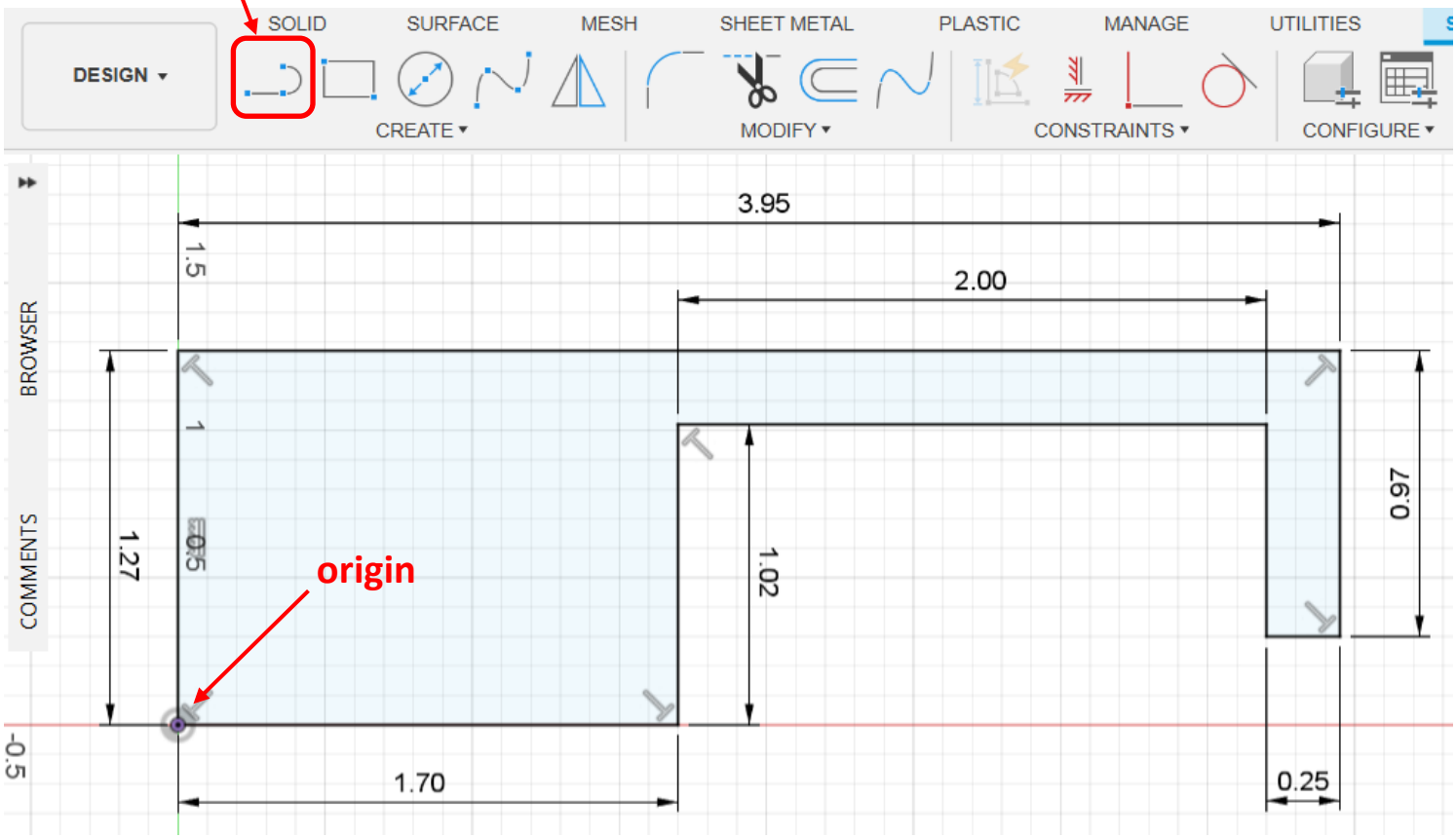




Unlike previous assignments, this more advanced assignment will provide less detailed steps. You are an expert now.

- click on the **Line** tool and then click on the **Origin** (where the green and red lines cross) to start the line.
- draw the sktech as shown below. As you should know, it is OK if the dimension lines look different and you can drag the values to move them. It is the thicker blue lines that are important.

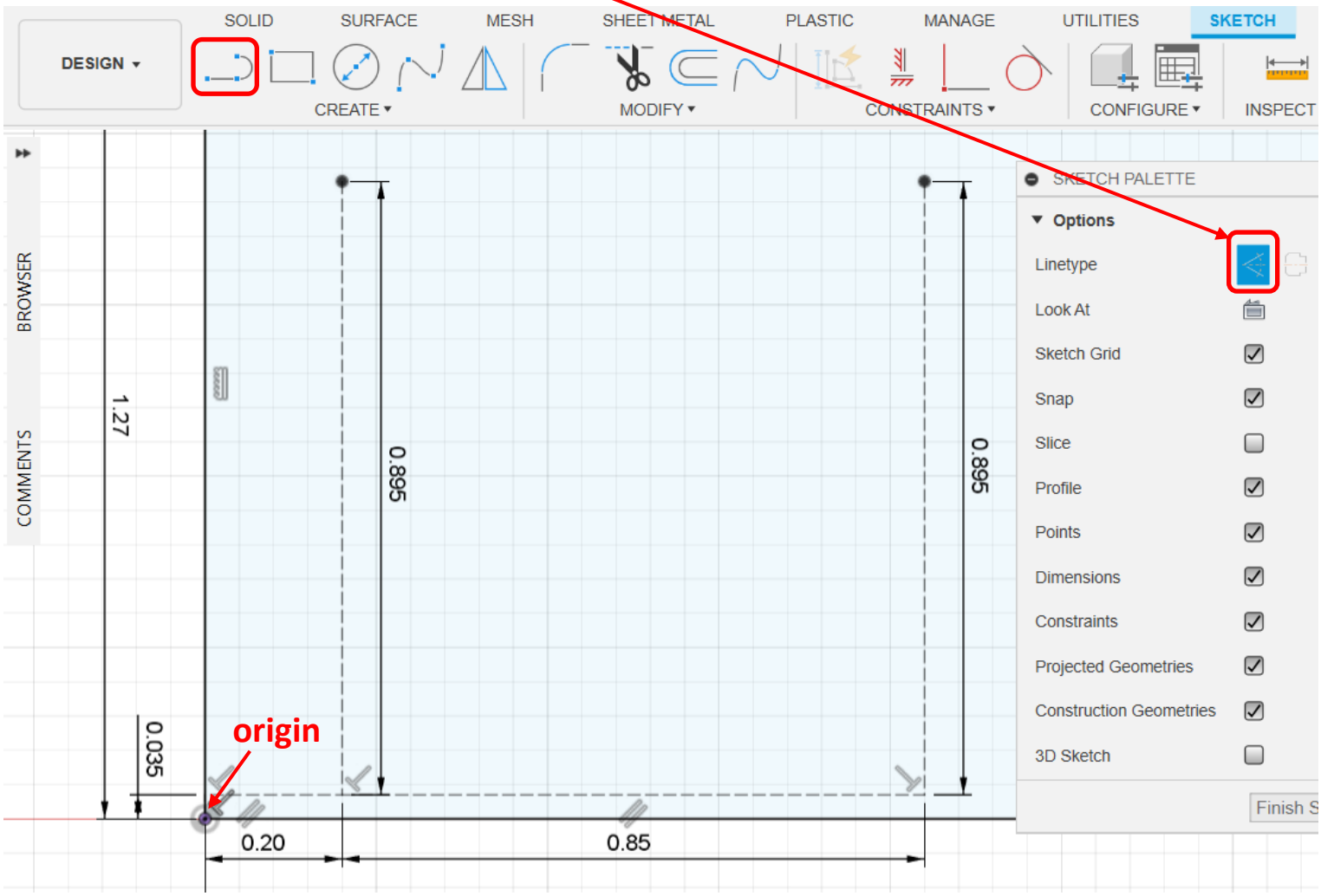
- **Do Not click Finish Sketch**. There is more to do.



This step will use Construction lines (dashed lines) to specify screw hole positions. Construction lines are used for positioning other elements and the Construction Lines themselves do not define edges.
Note that “normal” lines could be used, but it looks more professional when Construction Lines are used. However, one **must remember to turn off the Construction Line feature when starting to create normal lines.**

- click on the **Construction icon** for Linetype to highlight the icon
- starting at the Origin, extend a line upward by **0.035** and then to the right by **0.020**. Draw the three other lines as shown.
- click on the **Construction icon** again to turn it off.

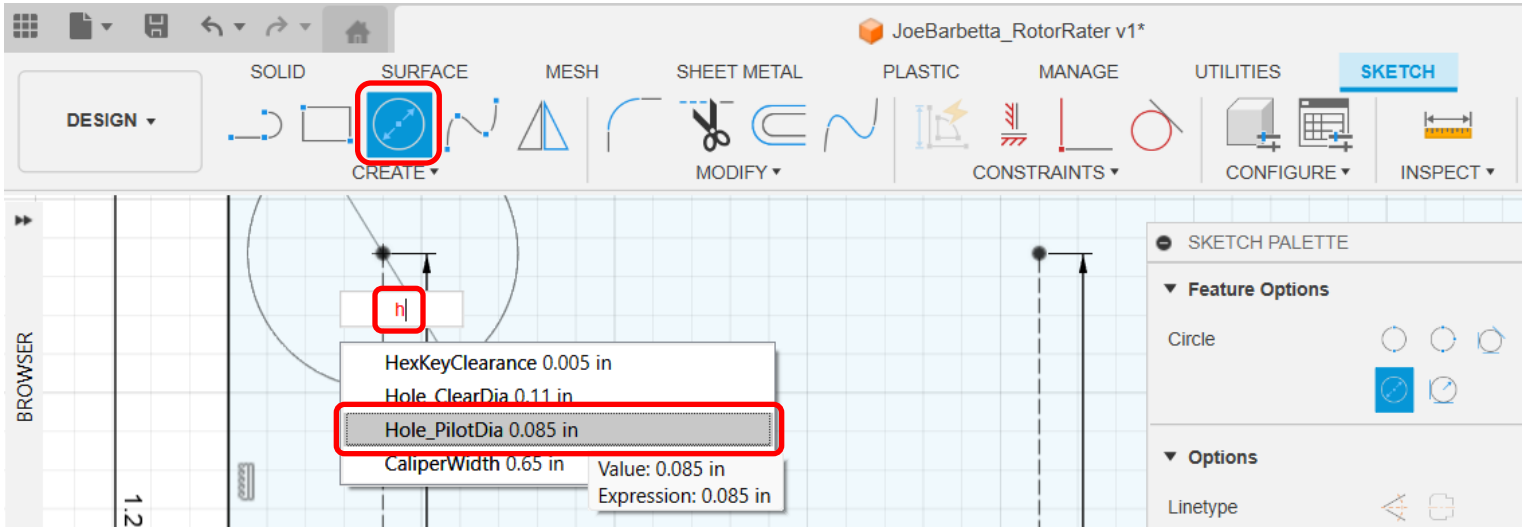
Do Not click Finish Sketch.



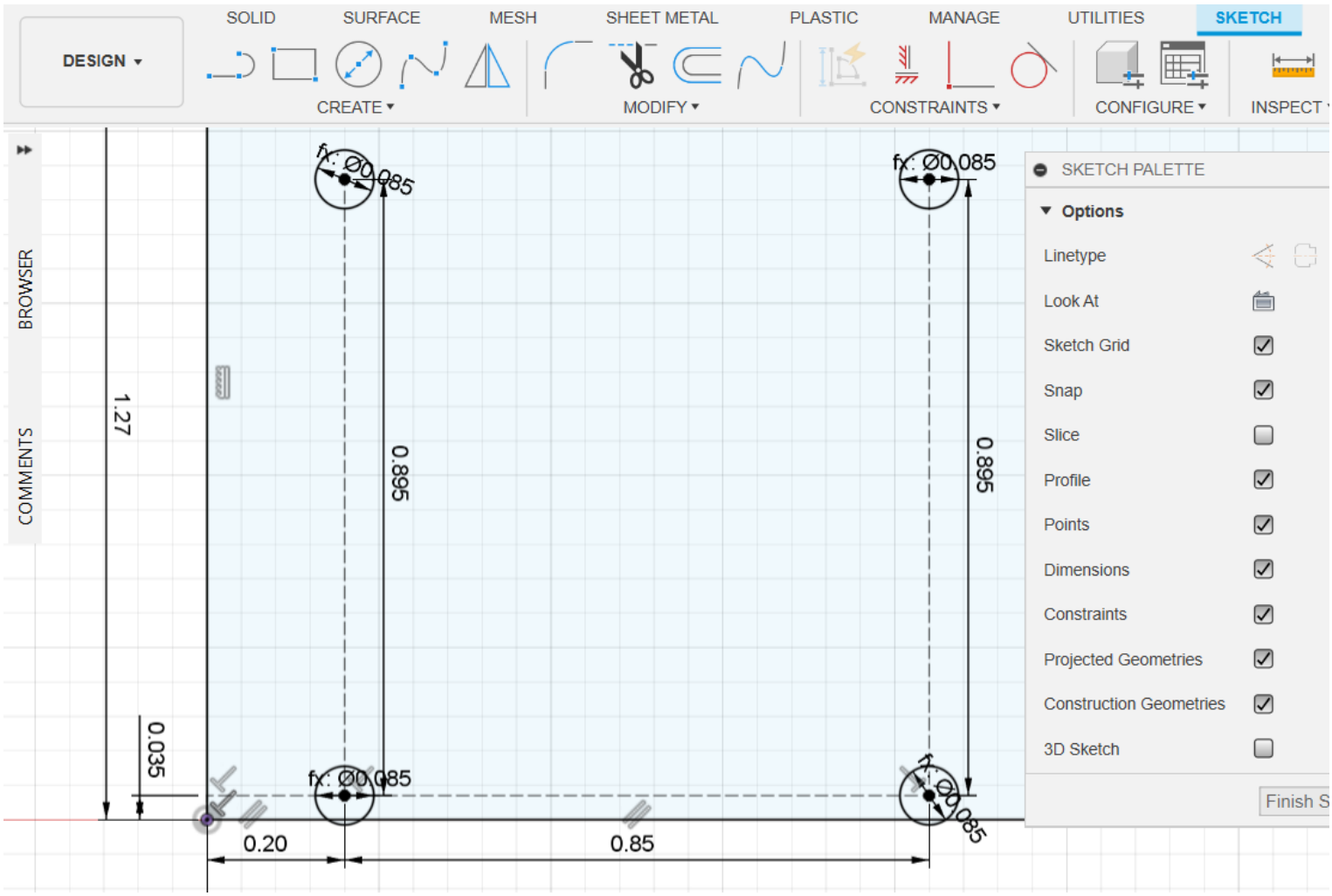
Where did all the crazy dimensions come from?
They are the result of many decades of rotor rater evolution.

This step will create the hole outlines.

- select the **Circle** tool. If the icon is not there, from the CREATE menu select Circle and then Center Diameter Circle.
- click on the **top of the first long constrction line**, extend the circle outward, type **h**, and then select **Hole_PilotDia**, and press the **Enter key**.

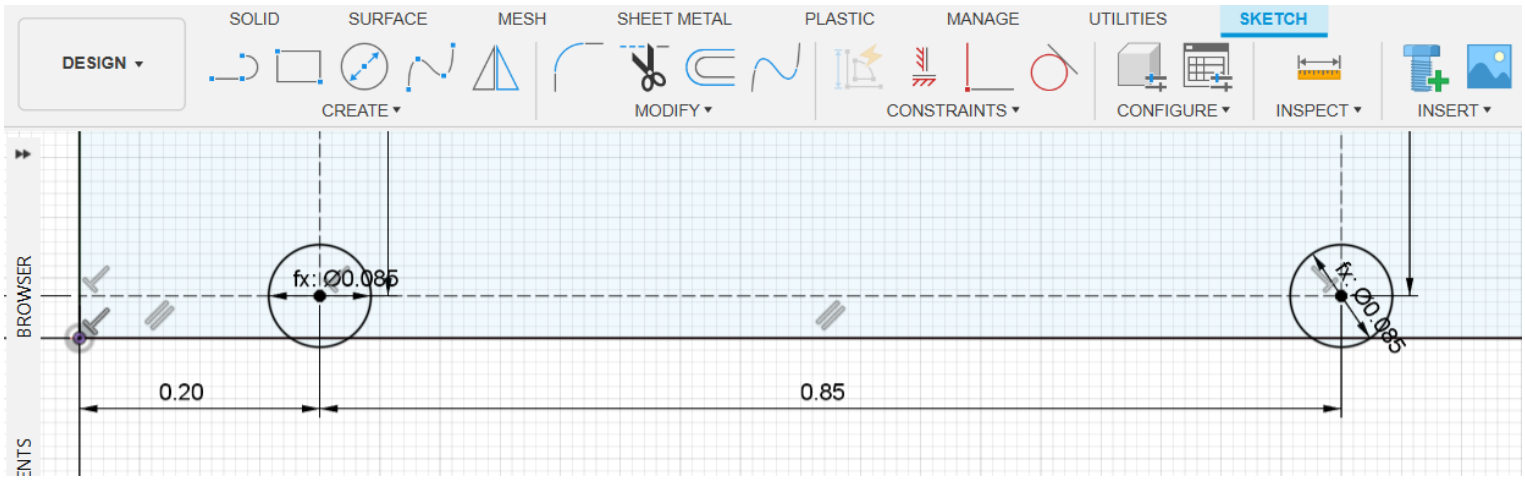


The circles just created should look like those below. Note that the circles have solid lines. If they are dashed, the Construction line feature was not turned off.

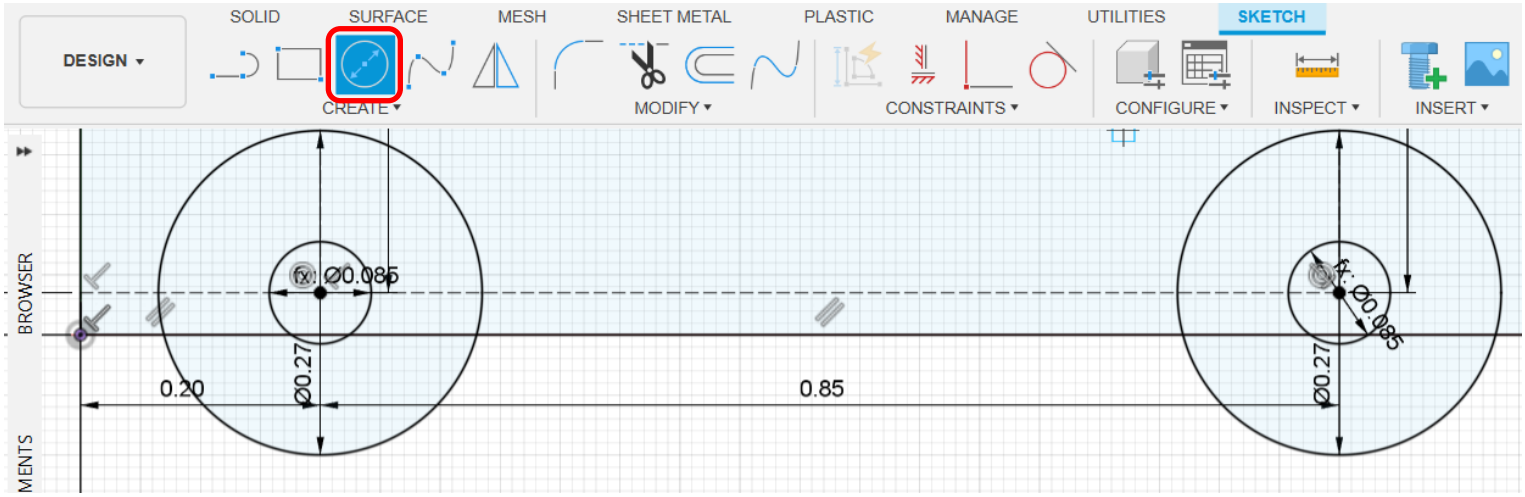


- zoom into the lower area to view the bottom two hole circles

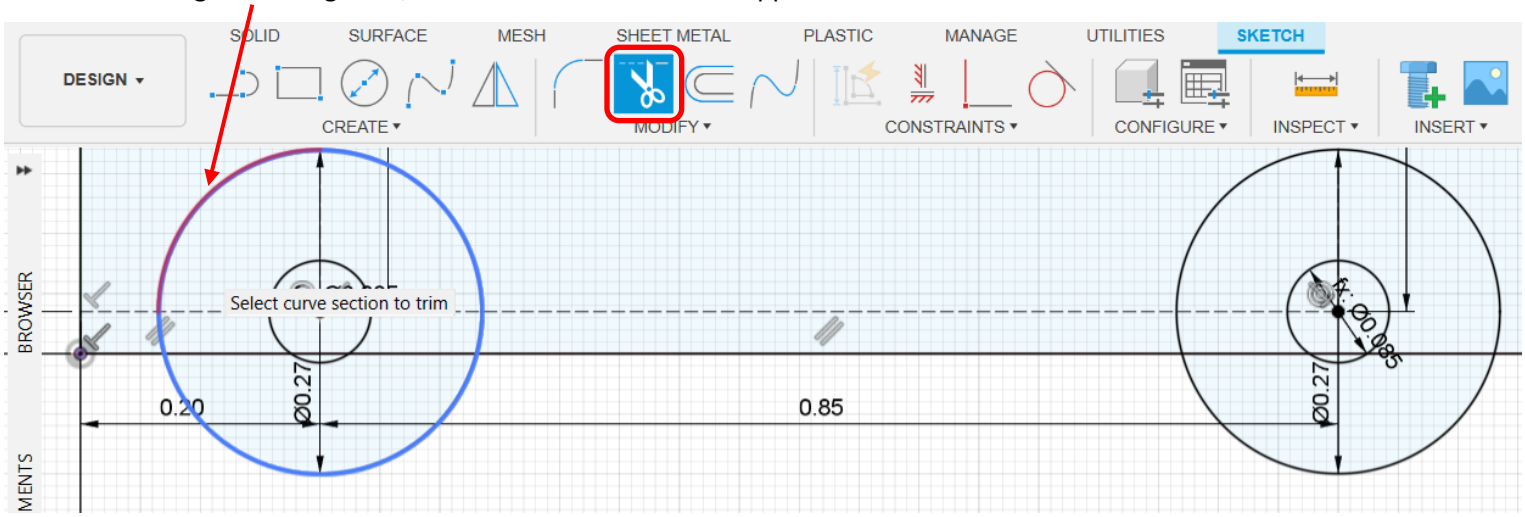
Part of the circle are off the edge and the resultant “partial” holes would barely hold a screw. What the sigma?



- select the **Circle** tool and starting at the center of each small circle (only the lower two as shown) and enter their dimensions as **0.27**.

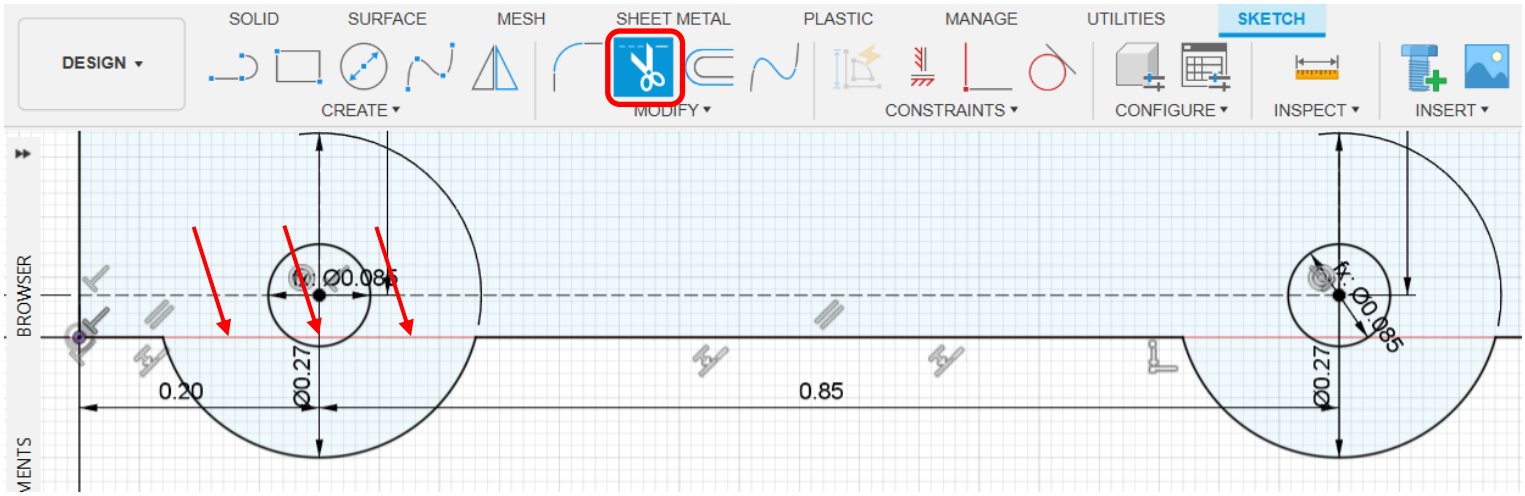


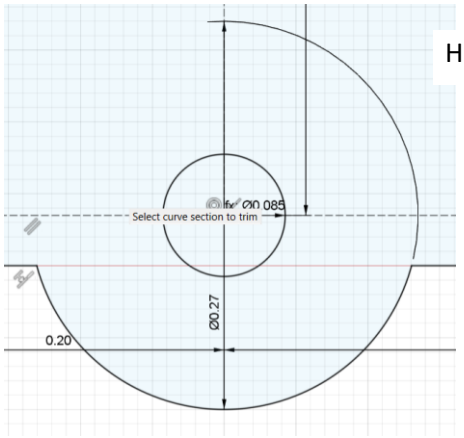
- select the **Trim** tool (if it’s not visible find it under the MODIFY menu)
- click on the large circle segment, which should cause it to disappear



- click on additional circle segments so that only the bottom arc is left
- click on the 3 horizontal line segments at each circle as well.

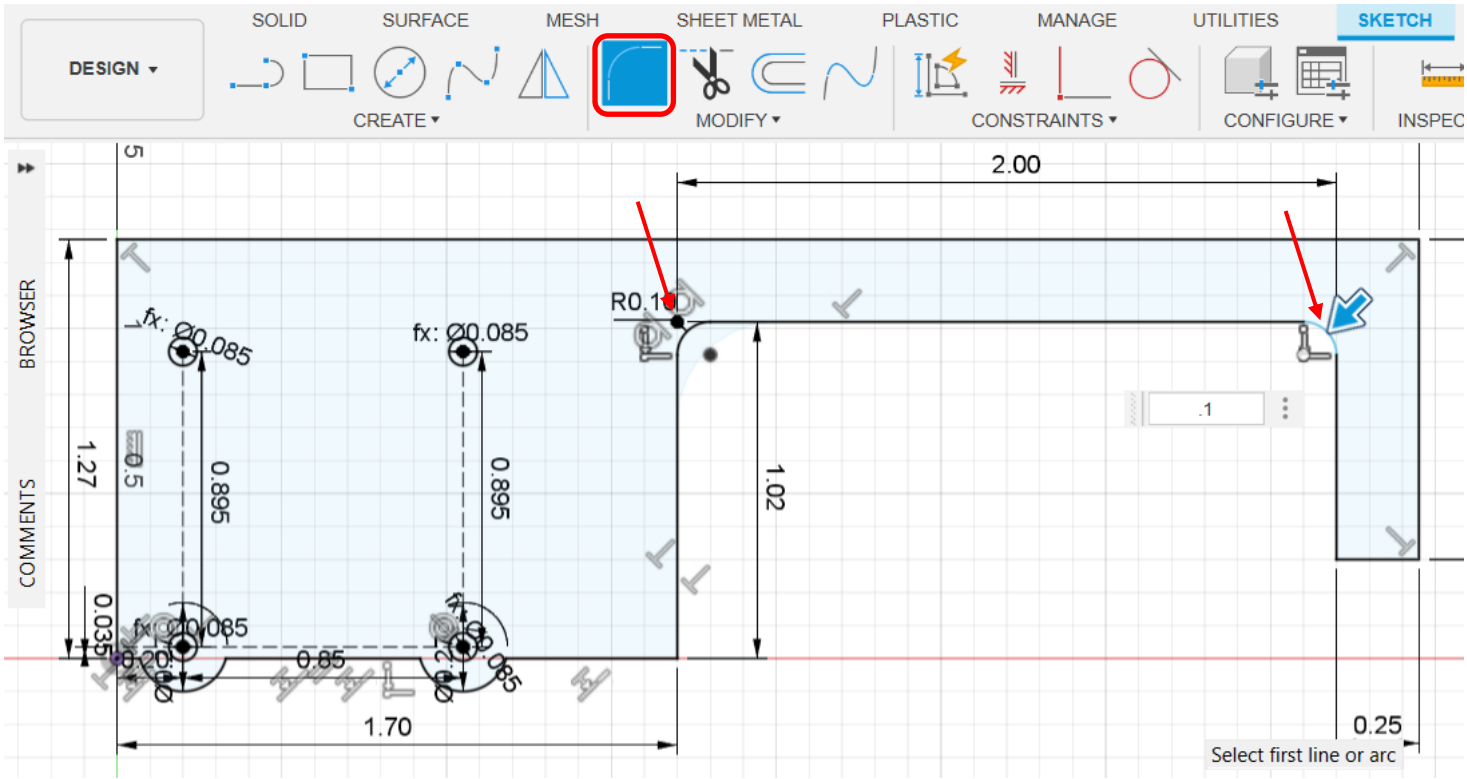
The result should look like that below. Note that there are still some arcs and lines left for dimensions.



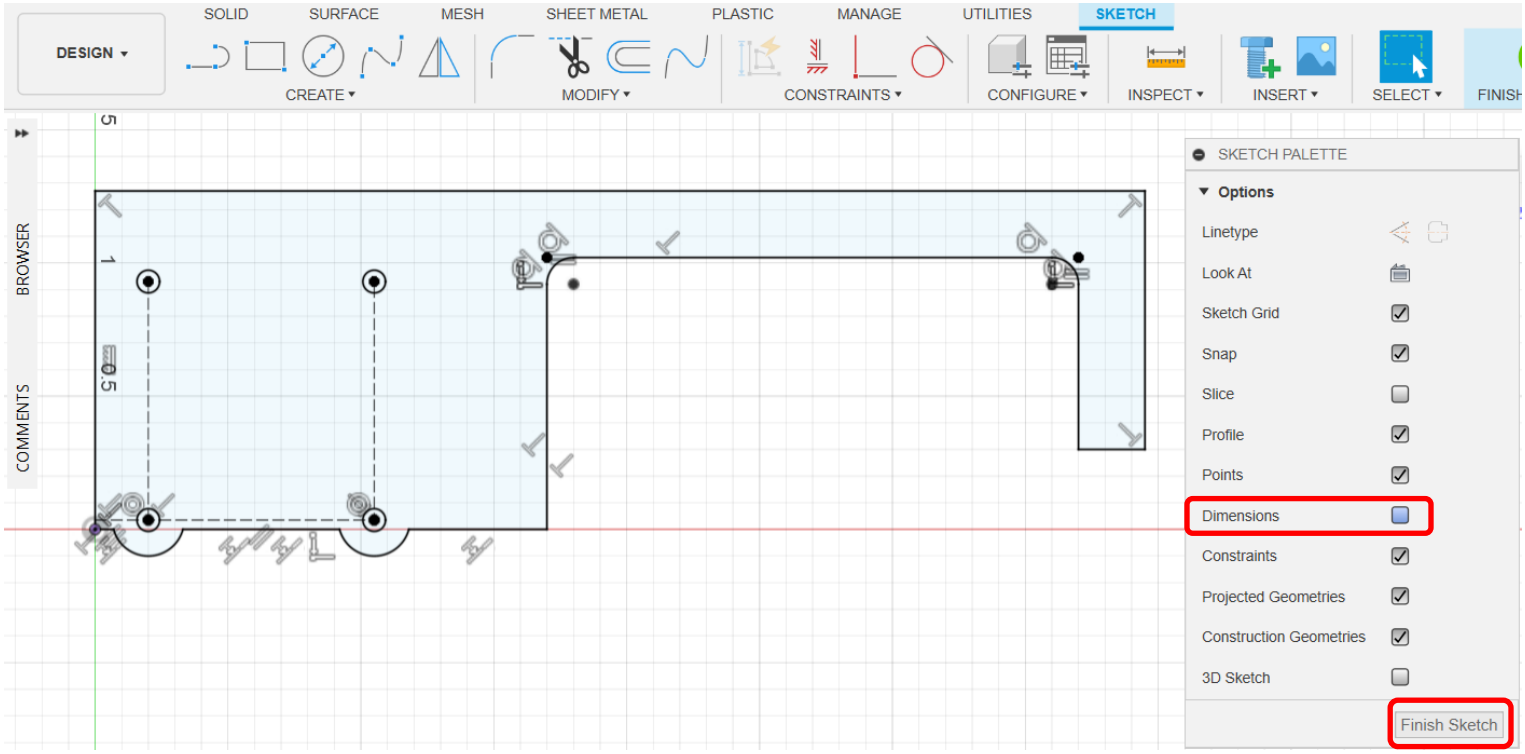


Here is a closeup of one hole.

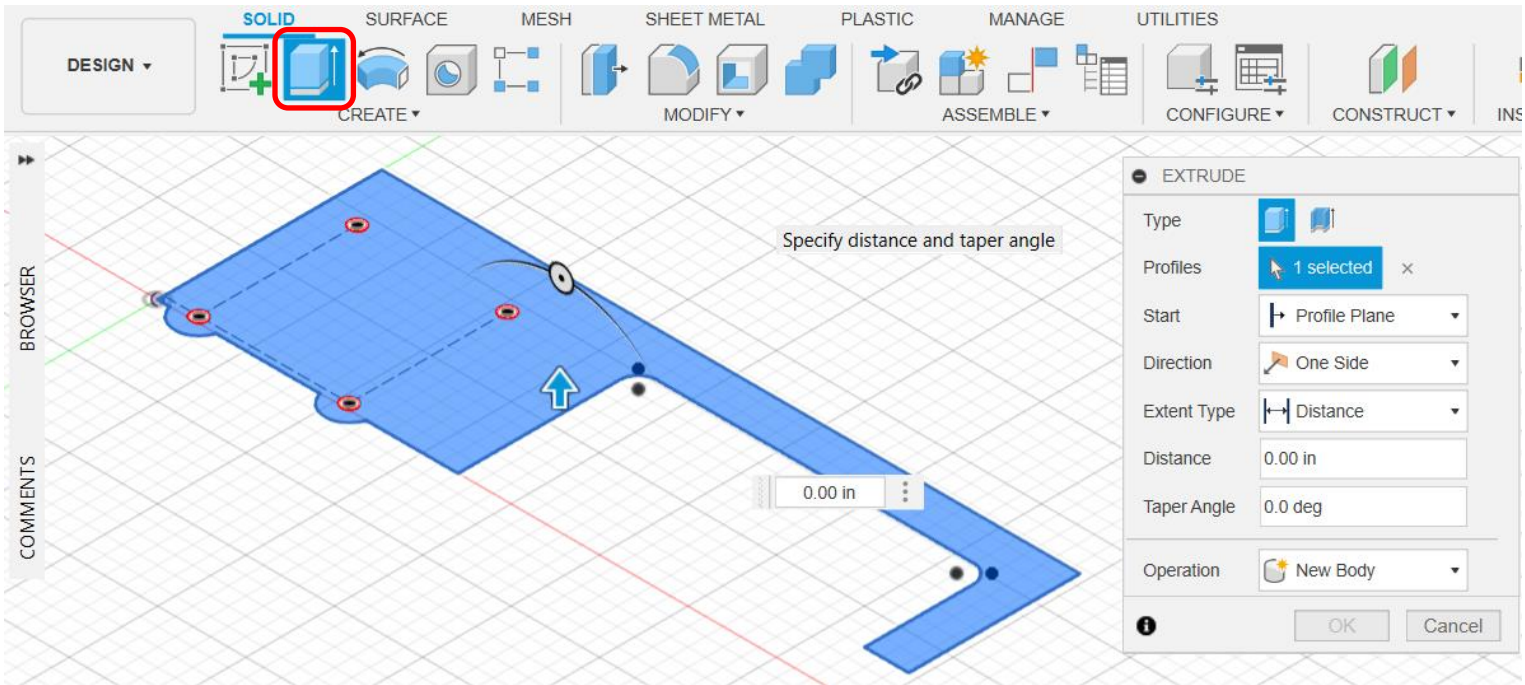
- zoom out and the sketch should look like that below.
- select the **Fillet** tool (if it is not visible find it under the MODIFY menu)
- click on the **two inner corners** and enter **0.1** for the radii.



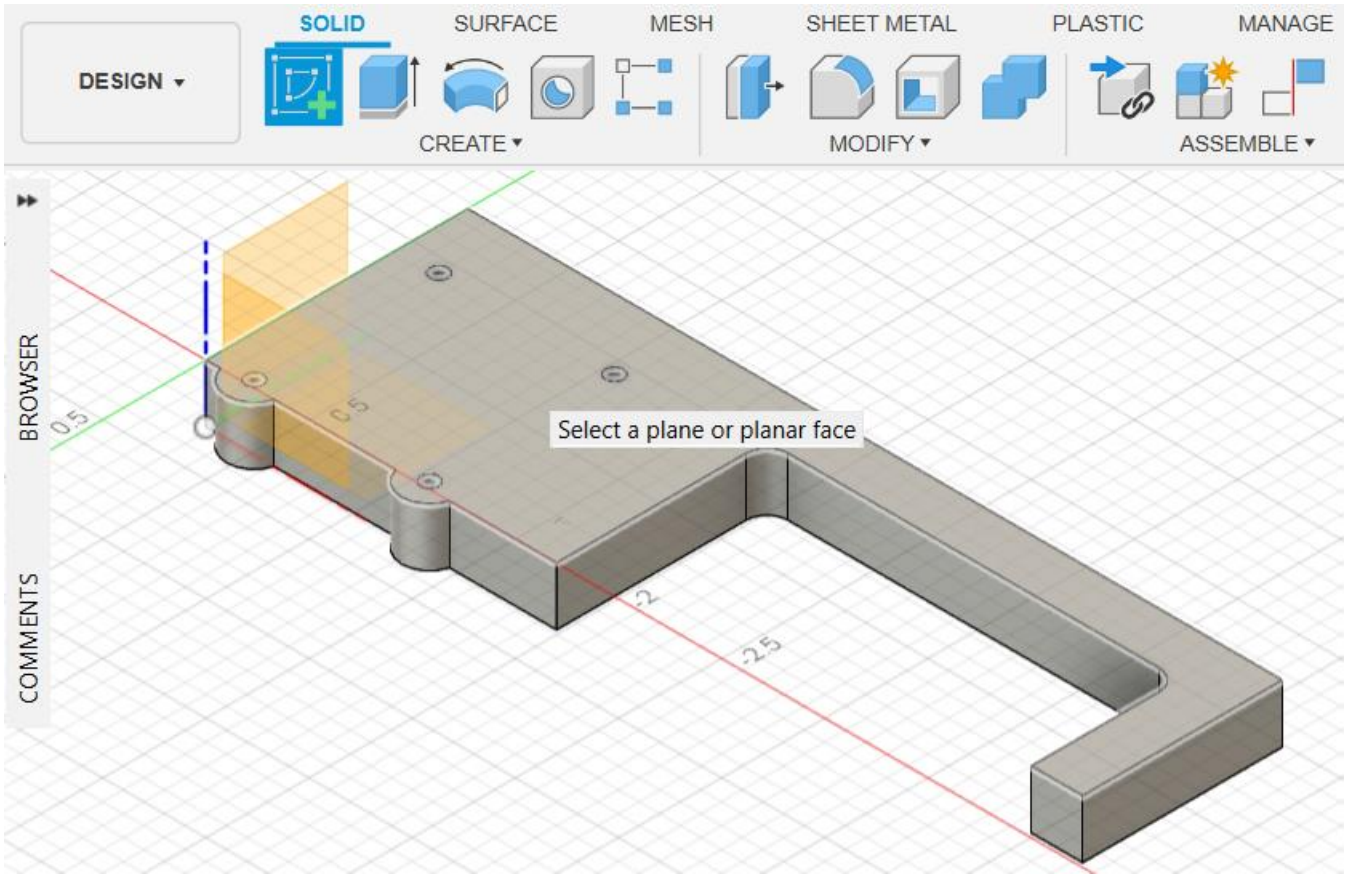
- click on the **Dimensions** check box and note how the dimensions disappear. They can be turned on or off as desired. If someone is looking over your shoulder, keep them on so your work looks more impressive.
- click **Finish Sketch**



- select the **Extrude** tool and extrude by **0.30**

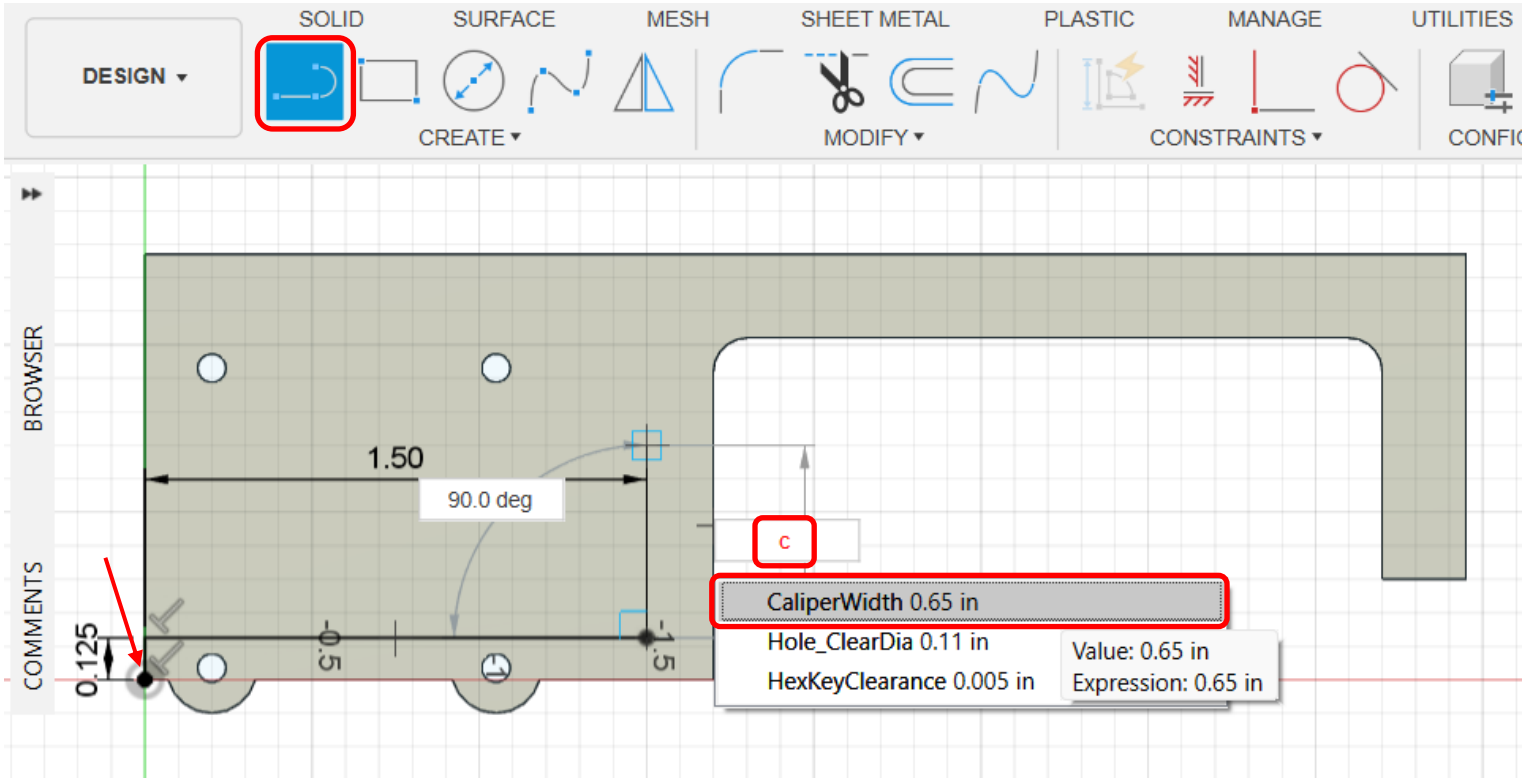


- start a Sketch on the top surface

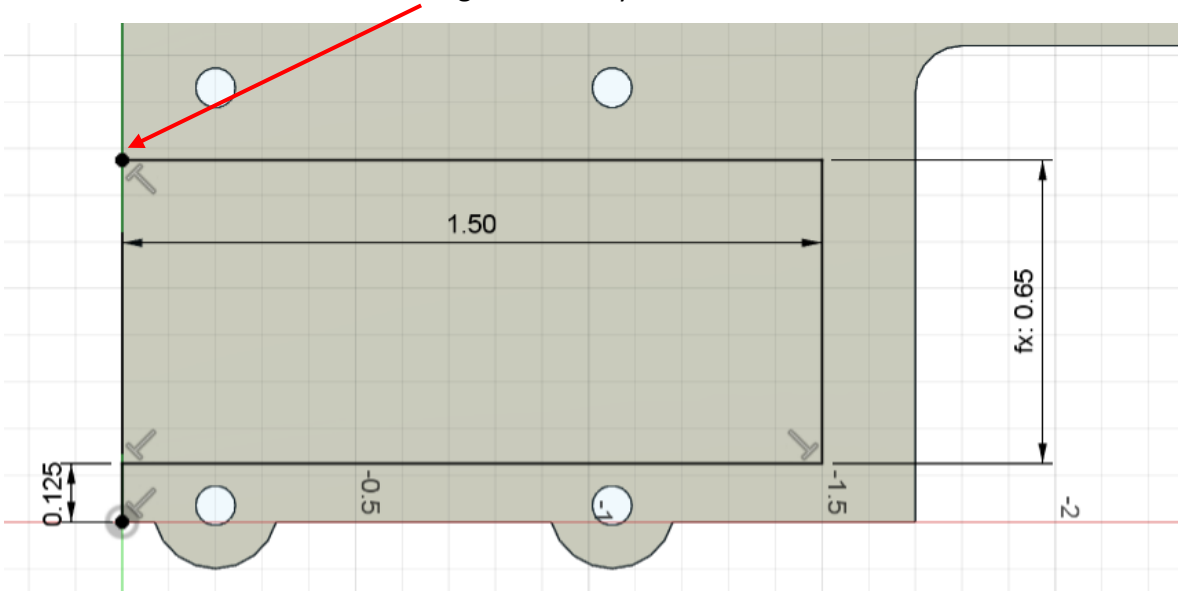


- from the **Origin** create a line upward **0.125** and then one to the right **1.50**
- from the **end of the last line** create a line upward, type **c**, select **CaliperWidth**, and press the **Enter key**

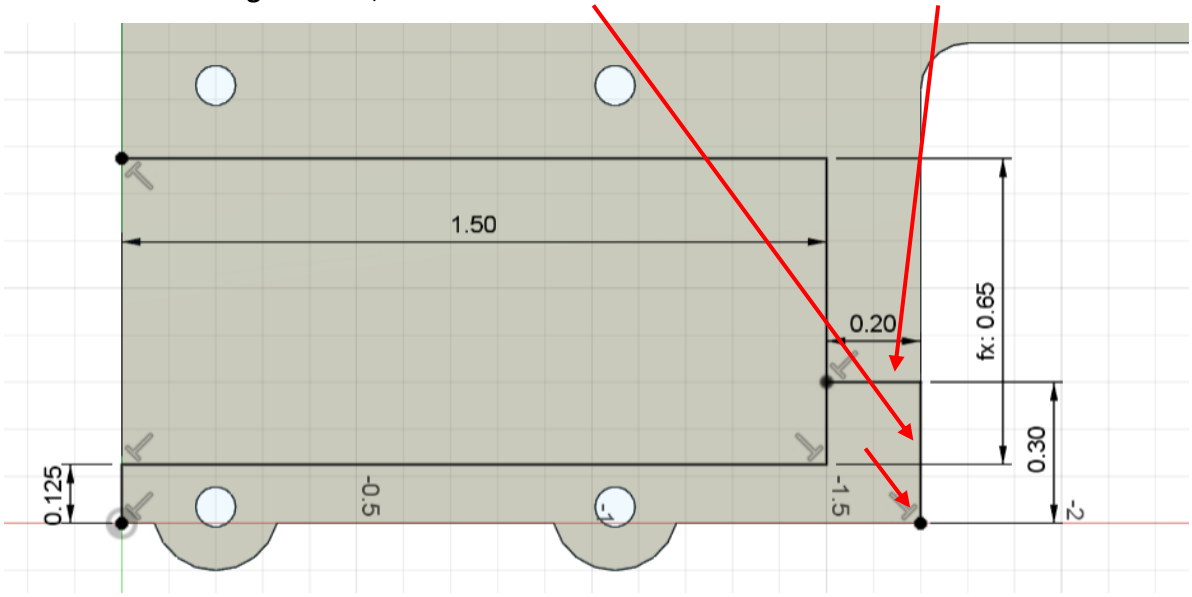
Note that the 0.125 line and some in the following steps, should be Construction Lines, however, these steps are being simplified and one doesn't have to worry about switching back to normal lines.



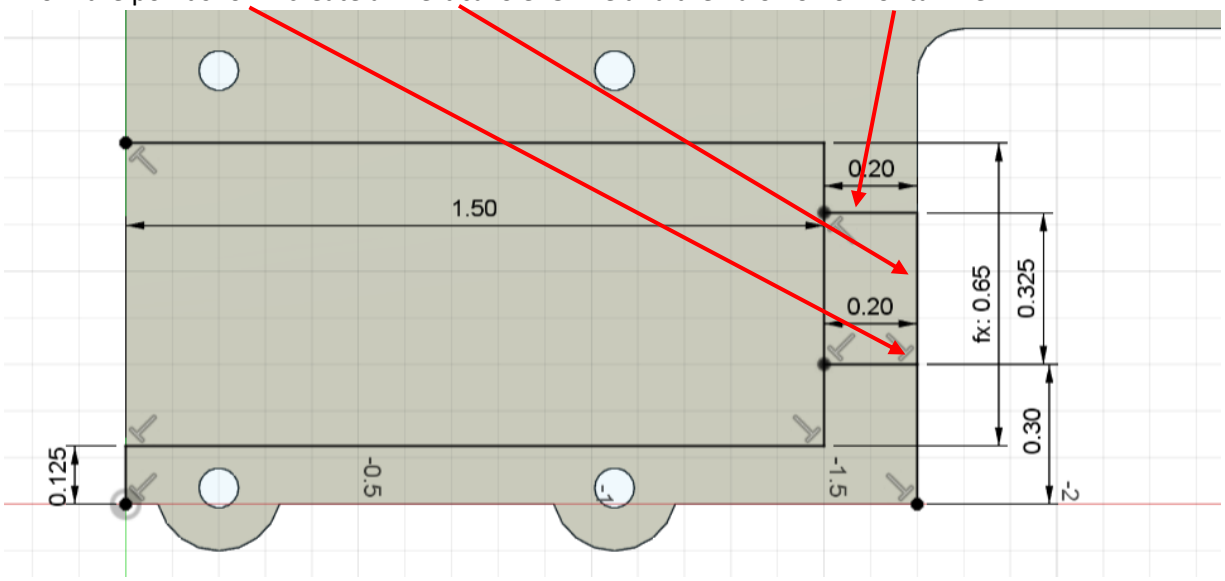
- create a horizontal line to the **left edge** of the body



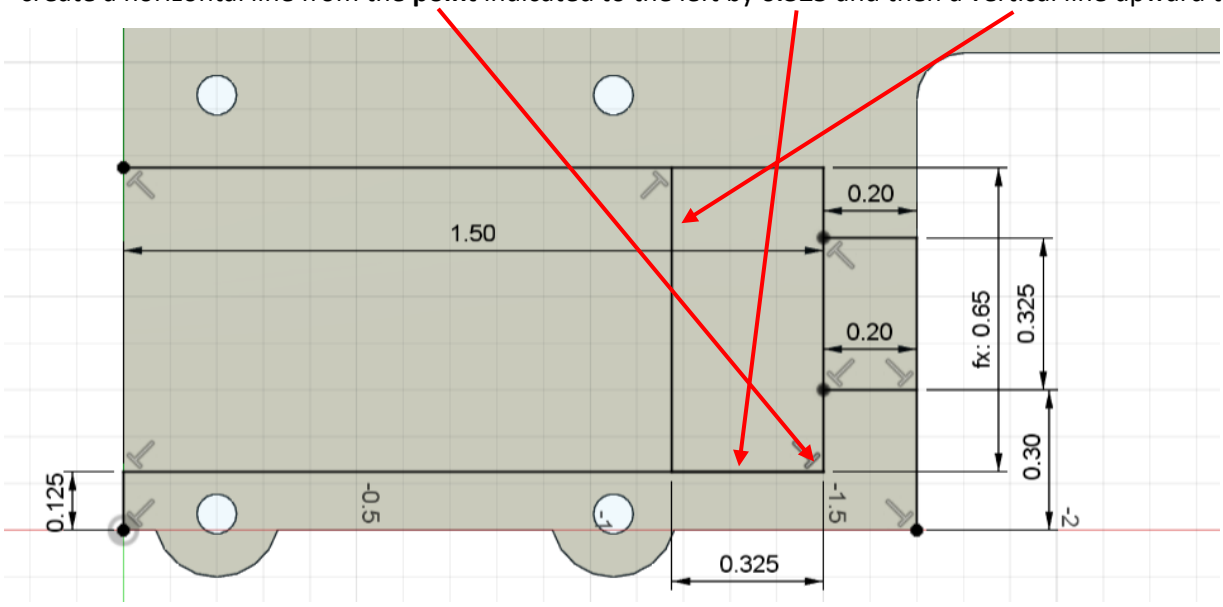
- from the **bottom right corner**, create a **0.30** vertical line and then a **0.20** horizontal line



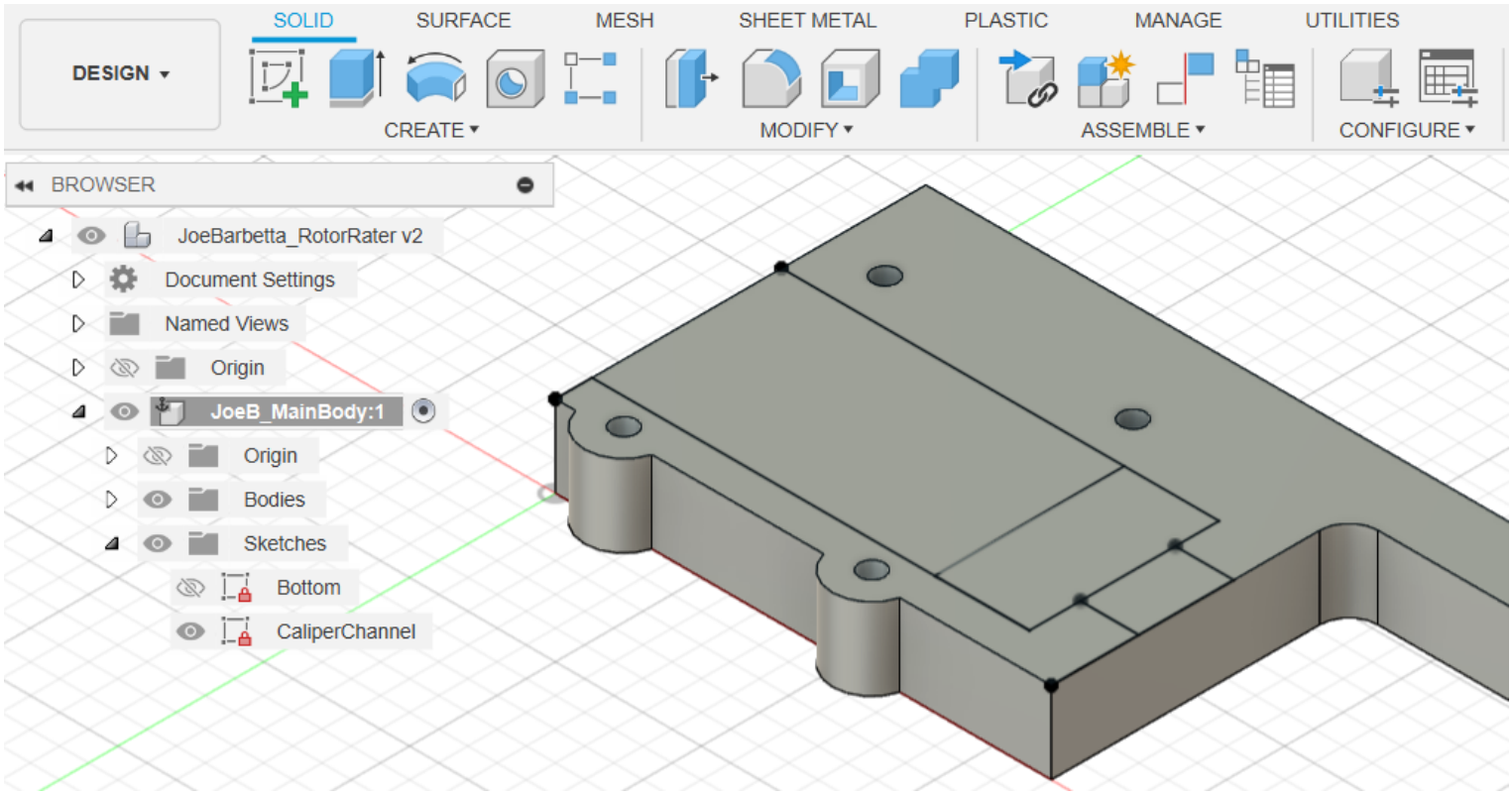
- from the point shown create an vertical **0.325** line and then a **0.20** horizontal line



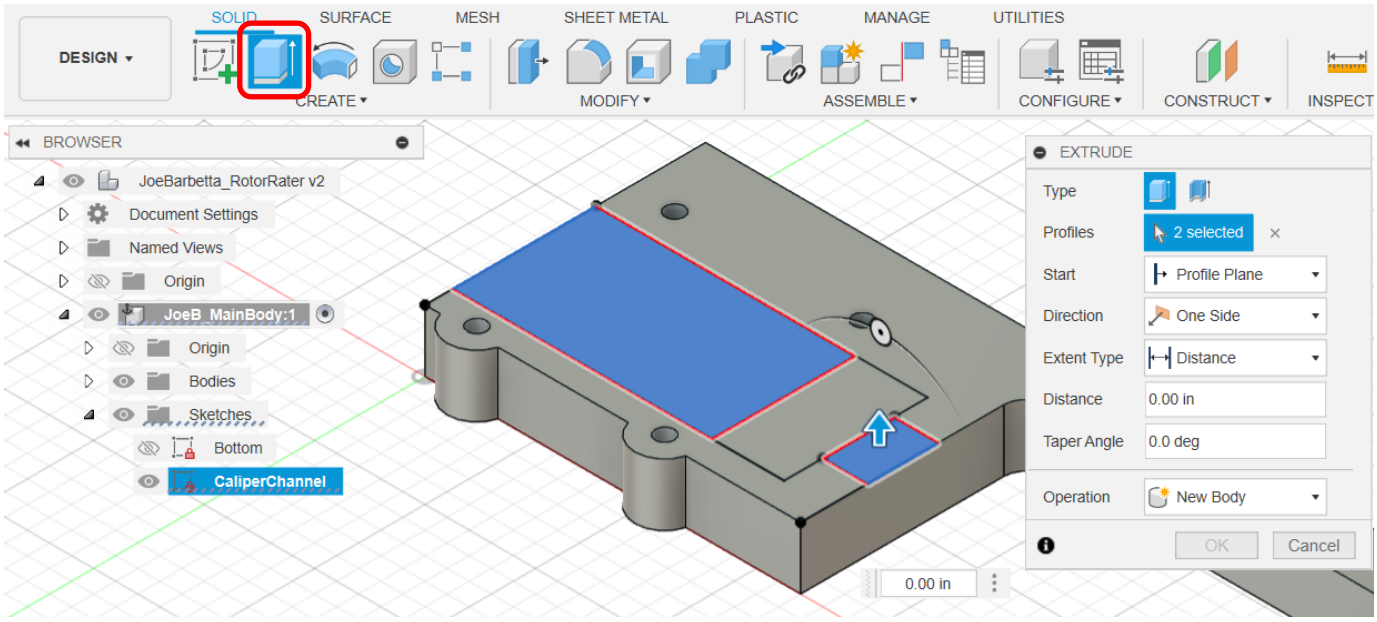
- create a horizontal line from the **point** indicated to the left by **0.325** and then a vertical line upward to the top line



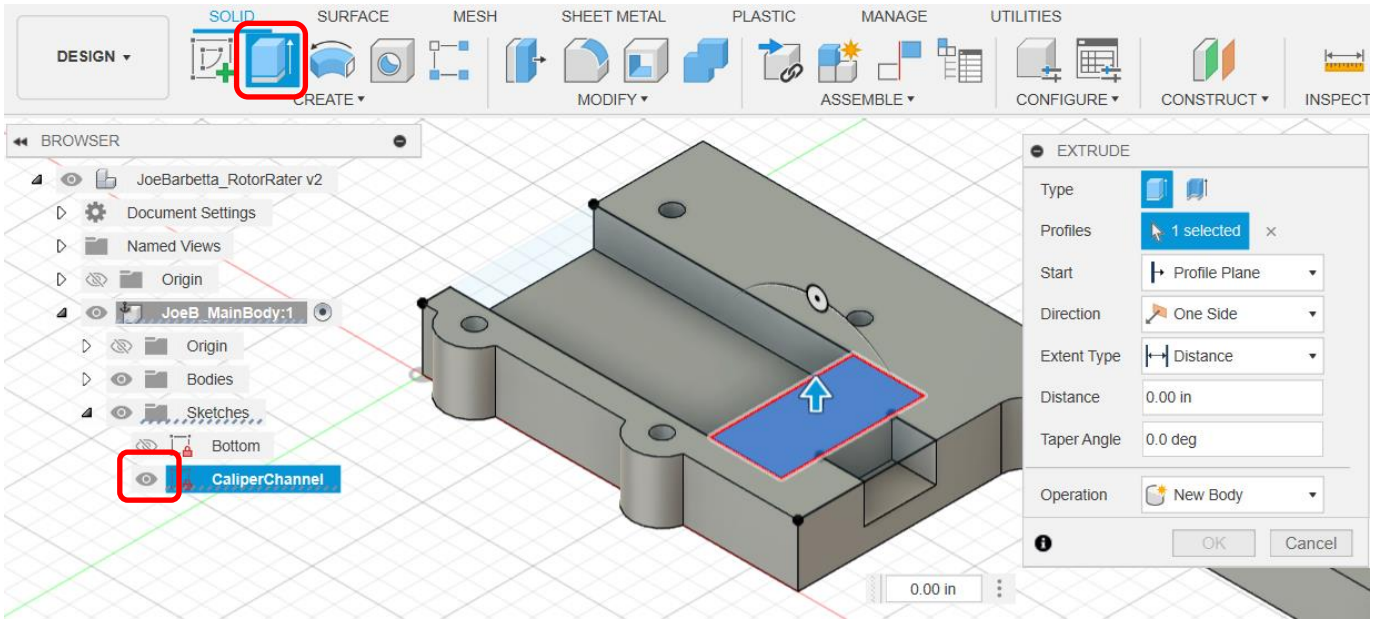
- click **Finish Sketch** and clikc on the **Home** icon at the **View Cube**
- **reopen the BROWSER**, which was closed previously and open the Sketches folder
- right-click on the 2nd Sketch and rename it to **CaliperChannel**



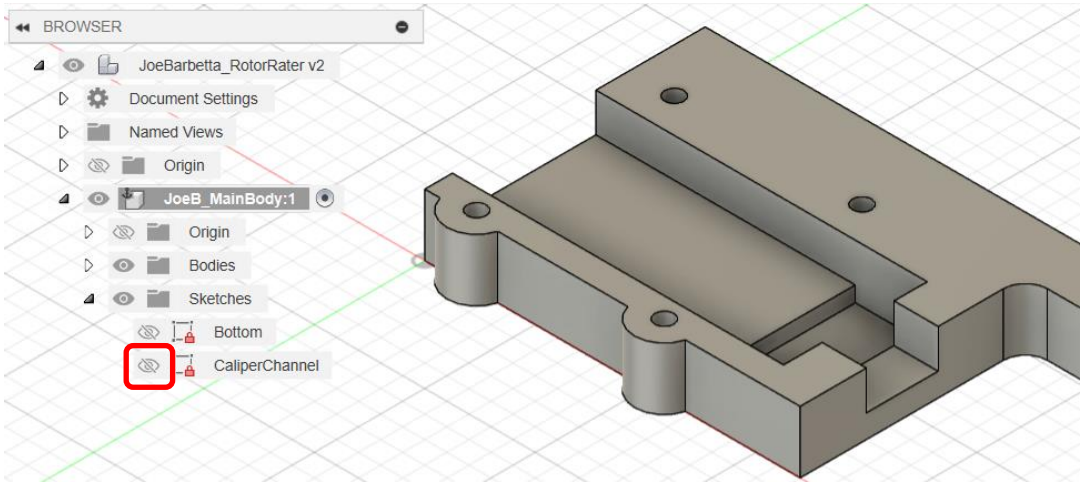
- select the **Extrude** tool and click on the two rectangular regions shown and extrude by **-0.174** (note the minus sign)



- click on the **eye** icon next to the **CaliperChannel** sketch to make the sketch visible again
- select the **Extrude** tool again and extrude the rectangluar region shown by -0.245 (note the minus sign)



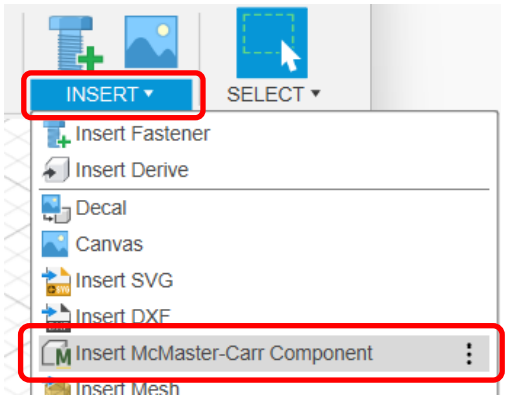
- click on the **eye** icon again to hide the Sketch



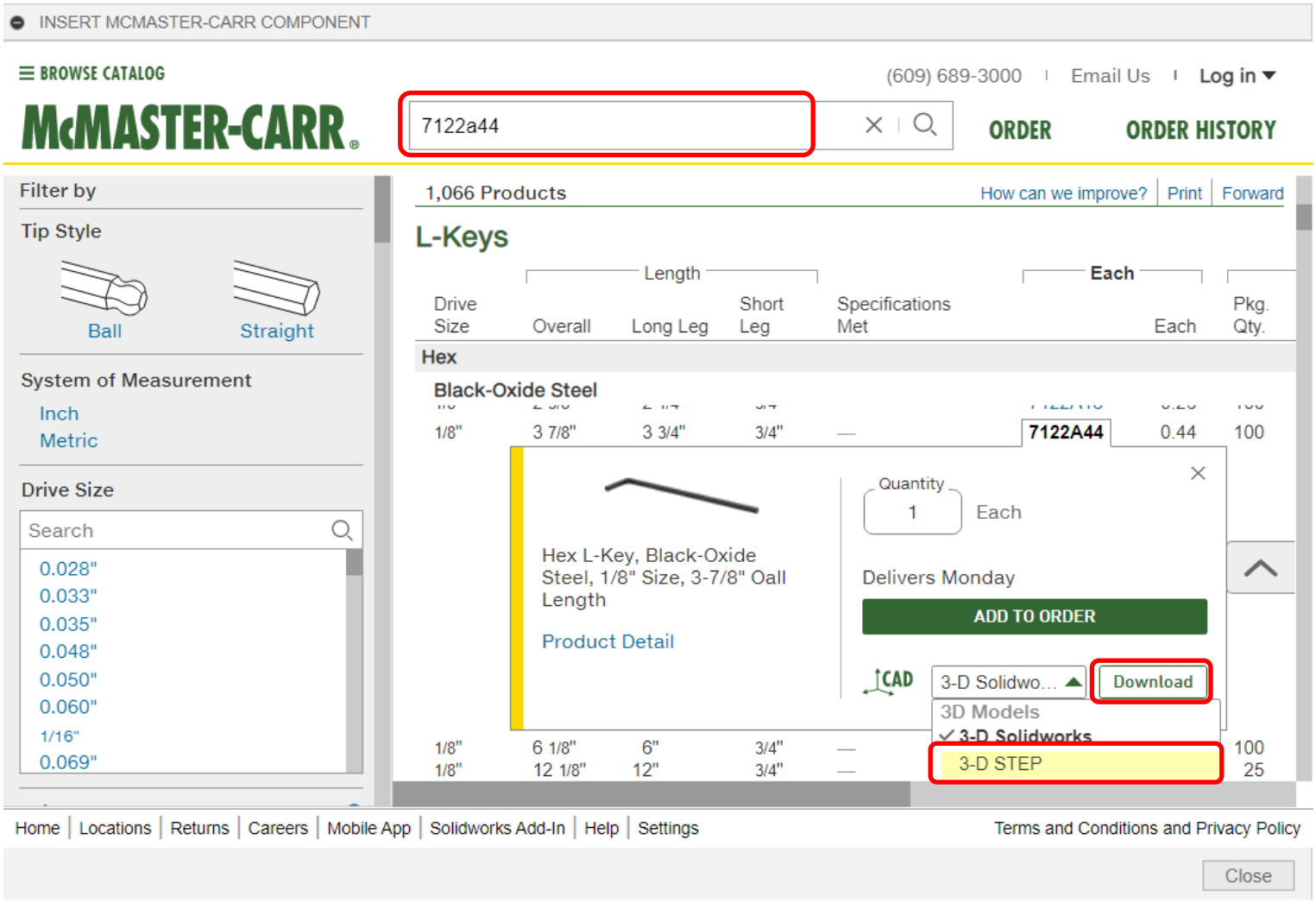
Importing a McMaster-Carr Component

We will be importing an L-shaped hex key into our design. Why do we need a hex key?
The hex key is being incorporated into the design add rigidity to the 3D printed part. Any L-shaped piece of metal could be used, but for this design a 1/8” hex key provides a good fit and is readily available.

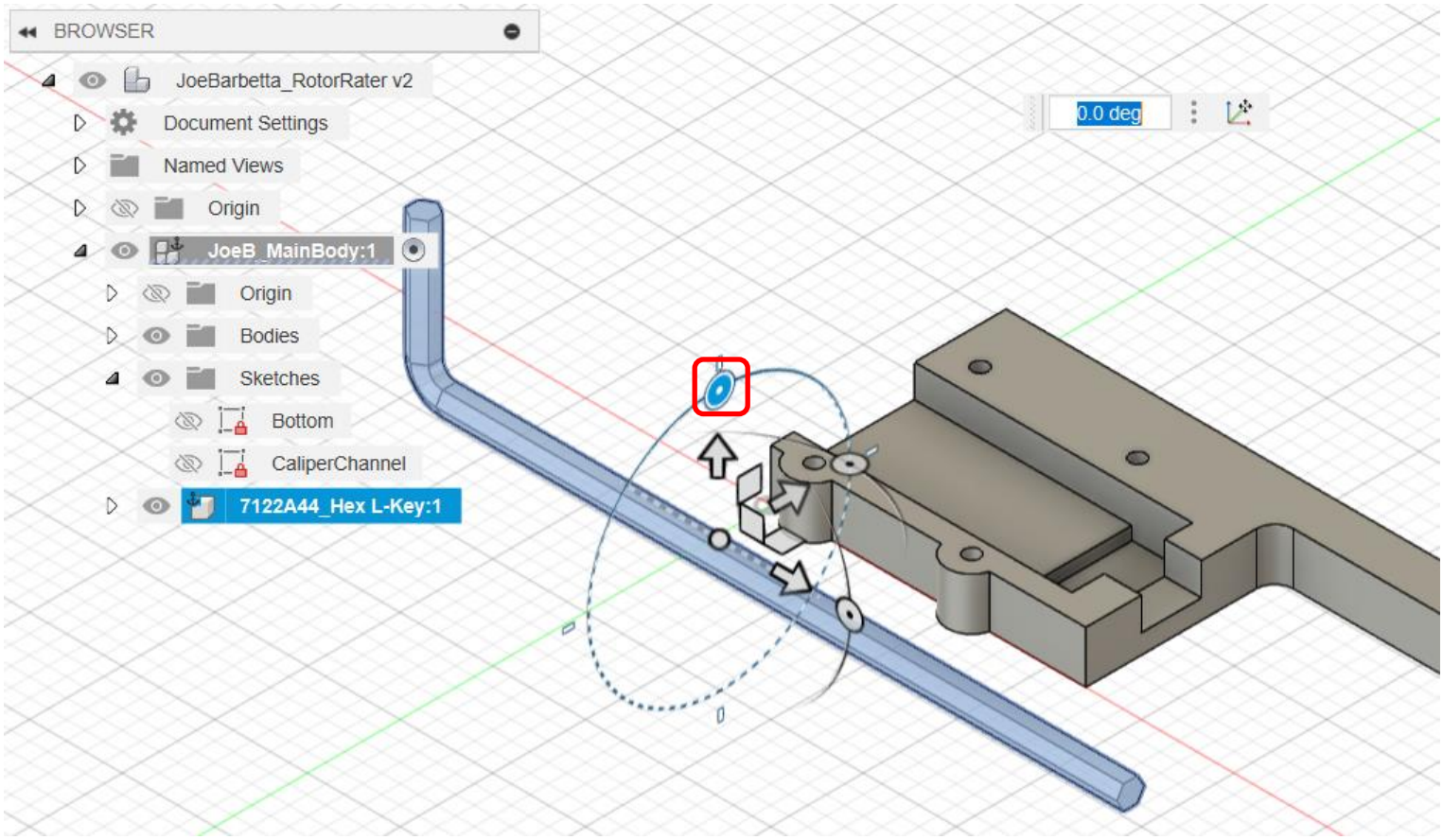
- from the top-right INSERT menu select **Insert McMaster-Carr Component**



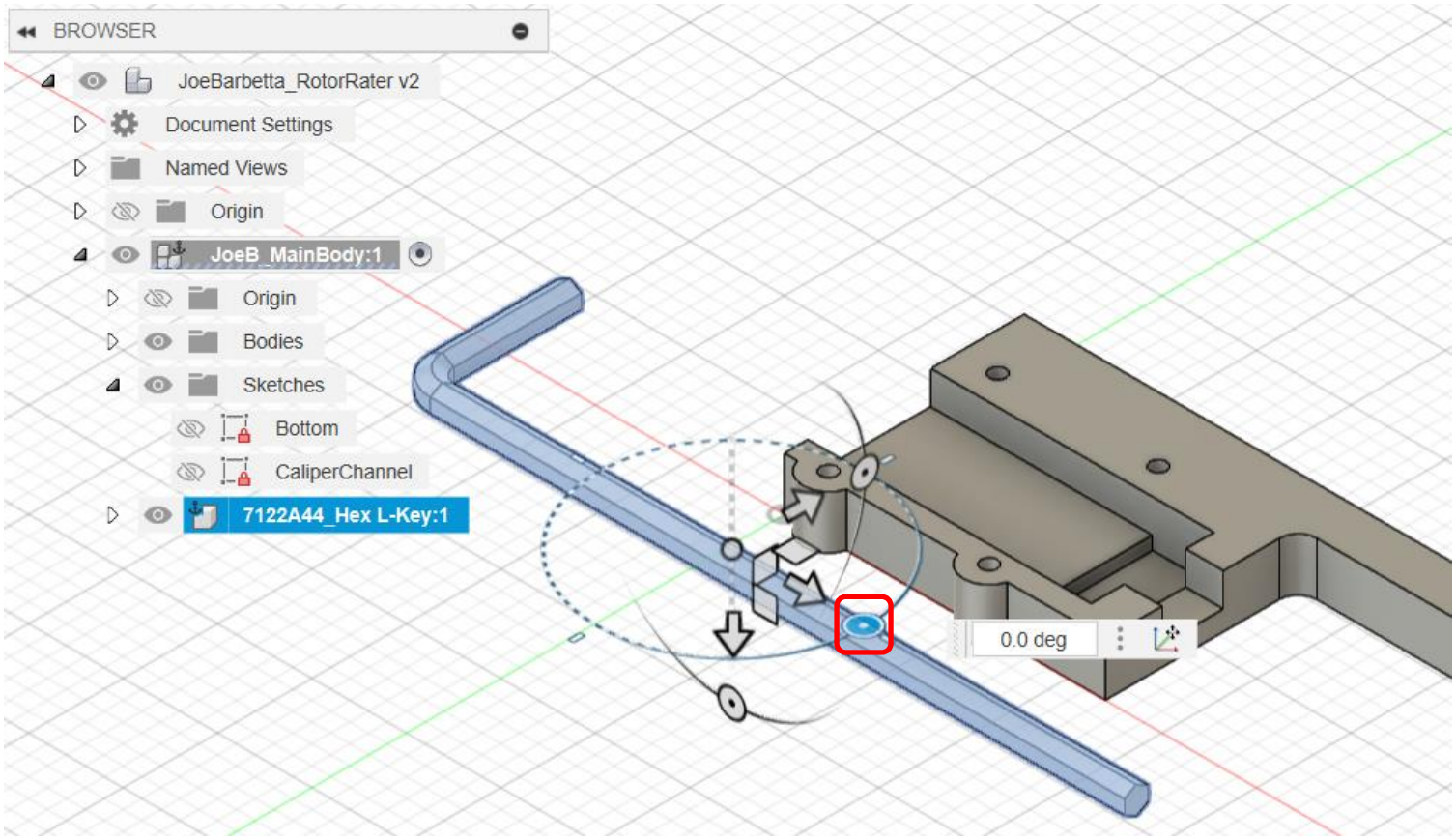
- enter the part number **7122a44** in the search box and press the **Enter key**
- expand the size of this window
- select **3-D STEP** from the CAD menu and then click **Download**



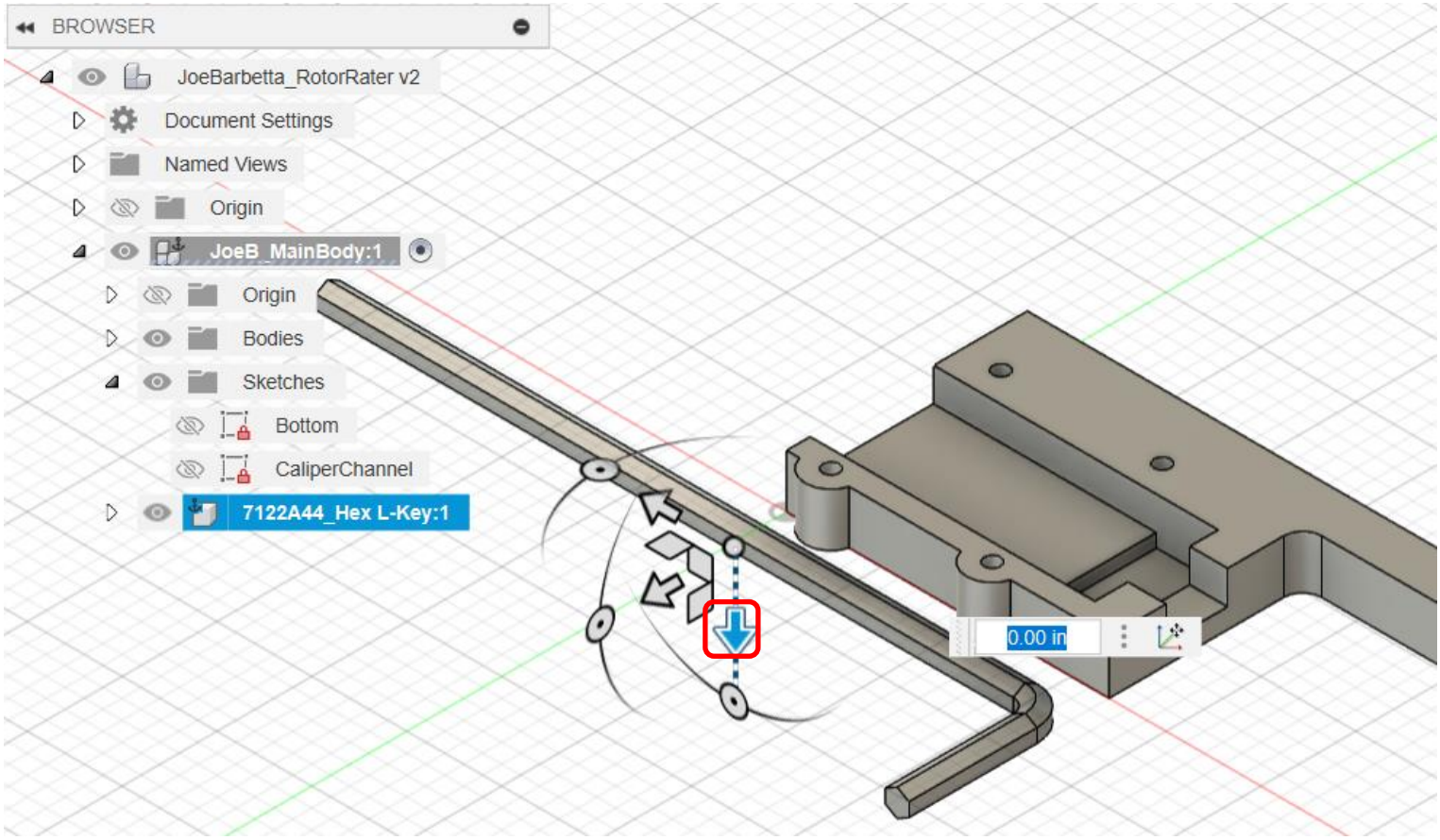
- drag the **Rotate** handle to rotate the hex key **90 degrees**



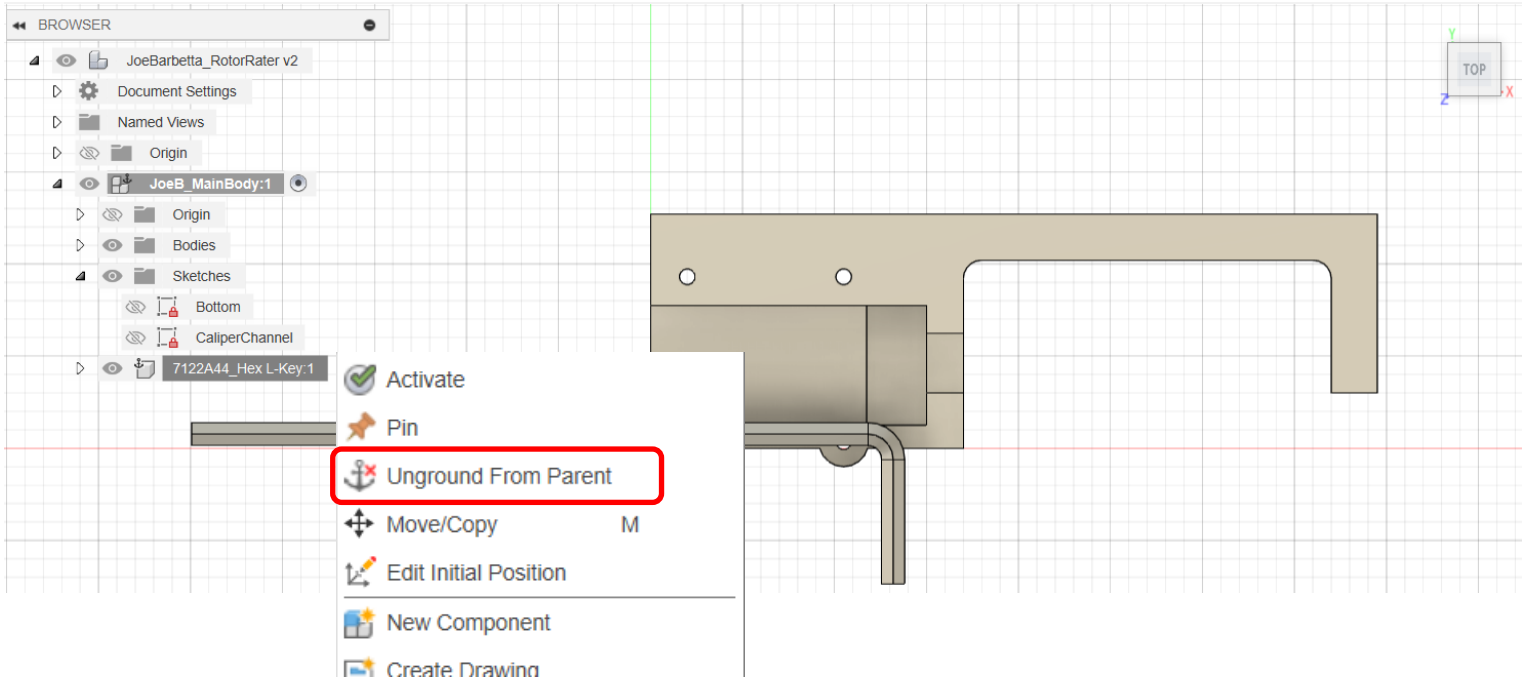
- drag the other **Rotate** handle to rotate the hex key **180 degrees**



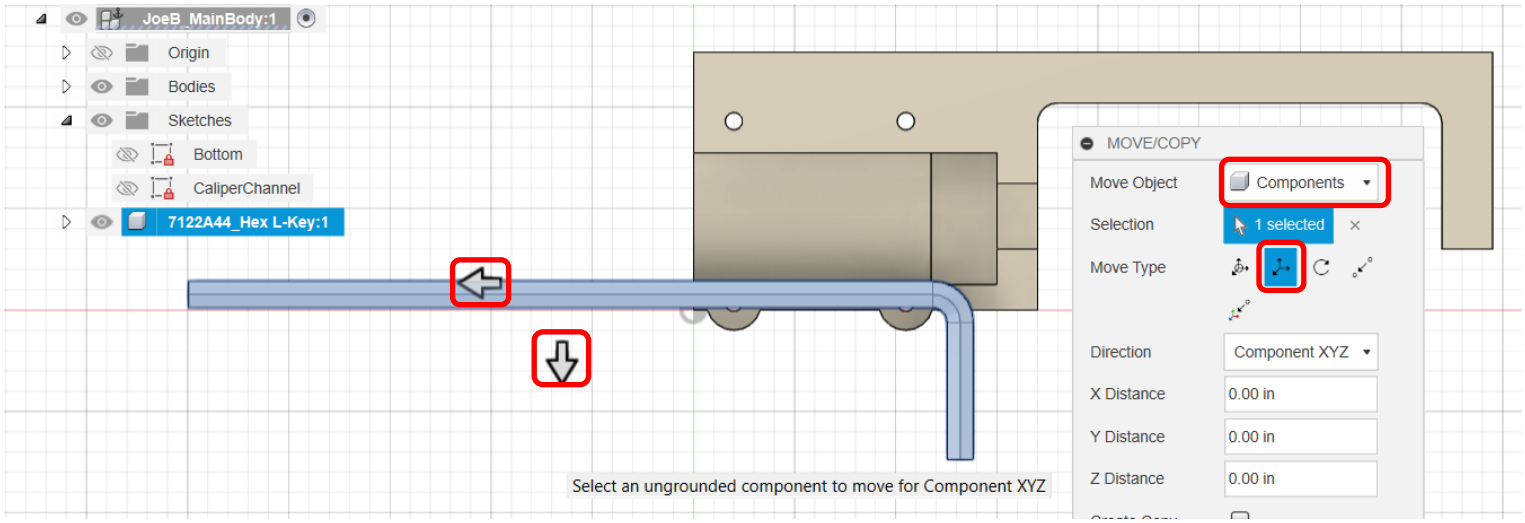
- drage the vertical move handle to move the hex key up by **1**



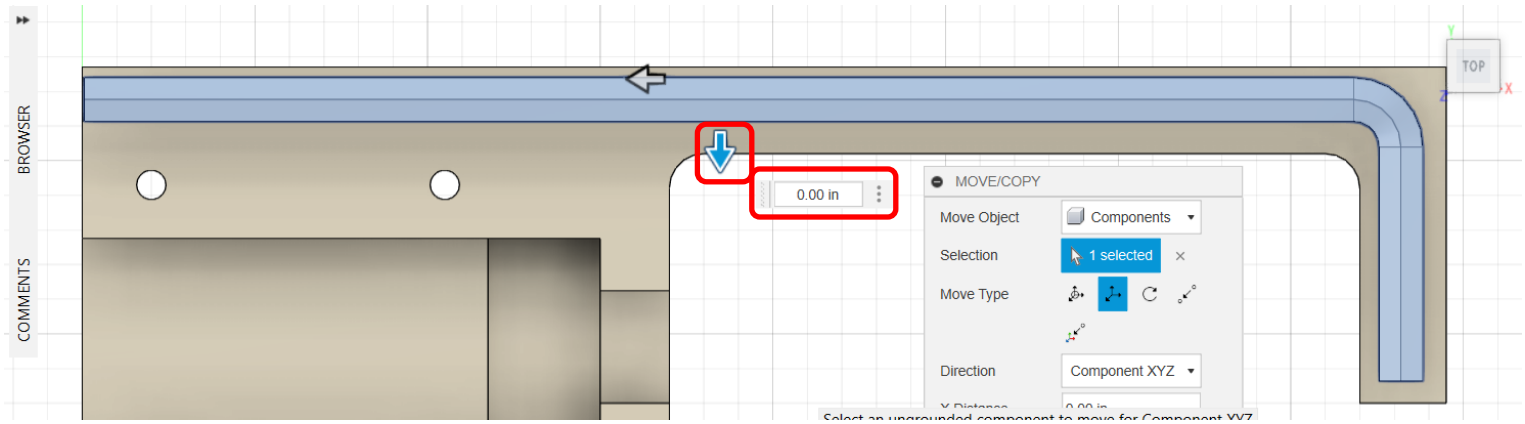
- click on the View Cube for a Top View
- right-click on the **Hex Key component** and select **Unground From Parent**



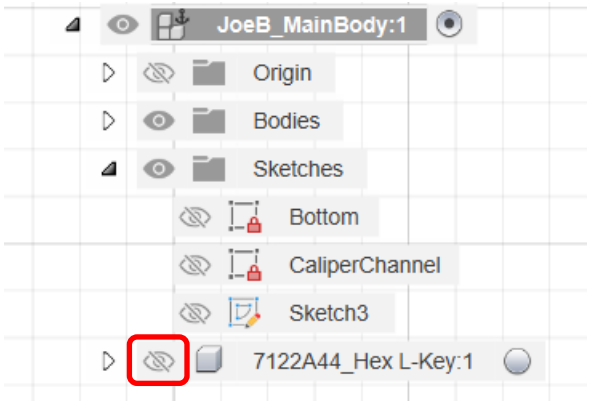
- select **Move/Copy** from the **MODIFY** menu
- ensure that **Components** is selected and the **Translate** icon is highlighted
- click on the hex key and use the horizontal and vertical move arrows to move it over the Main Body and click **OK**.



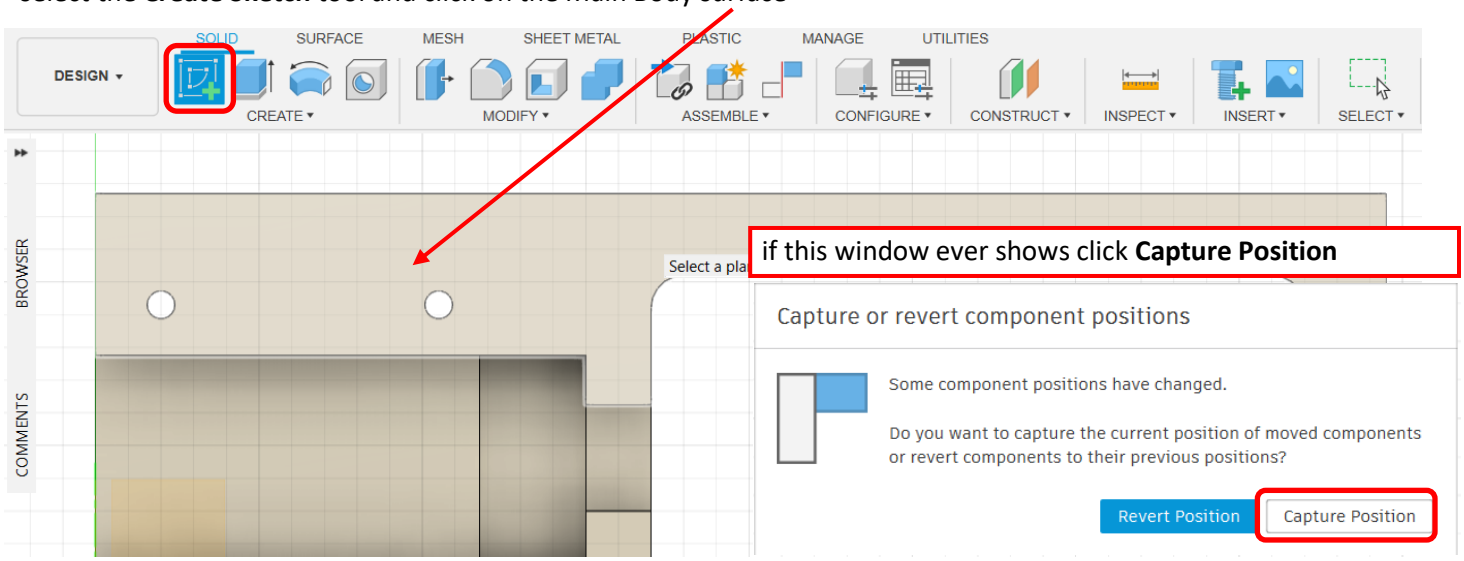
- close the BROWSER and zoom in as shown below
- select the **Move/Copy** tool again and use the **horizontal and vertical move arrows** to position the hex key more precisely. One can **select an arrow** and enter a value in the **value box** for fine movements. See the following picture.



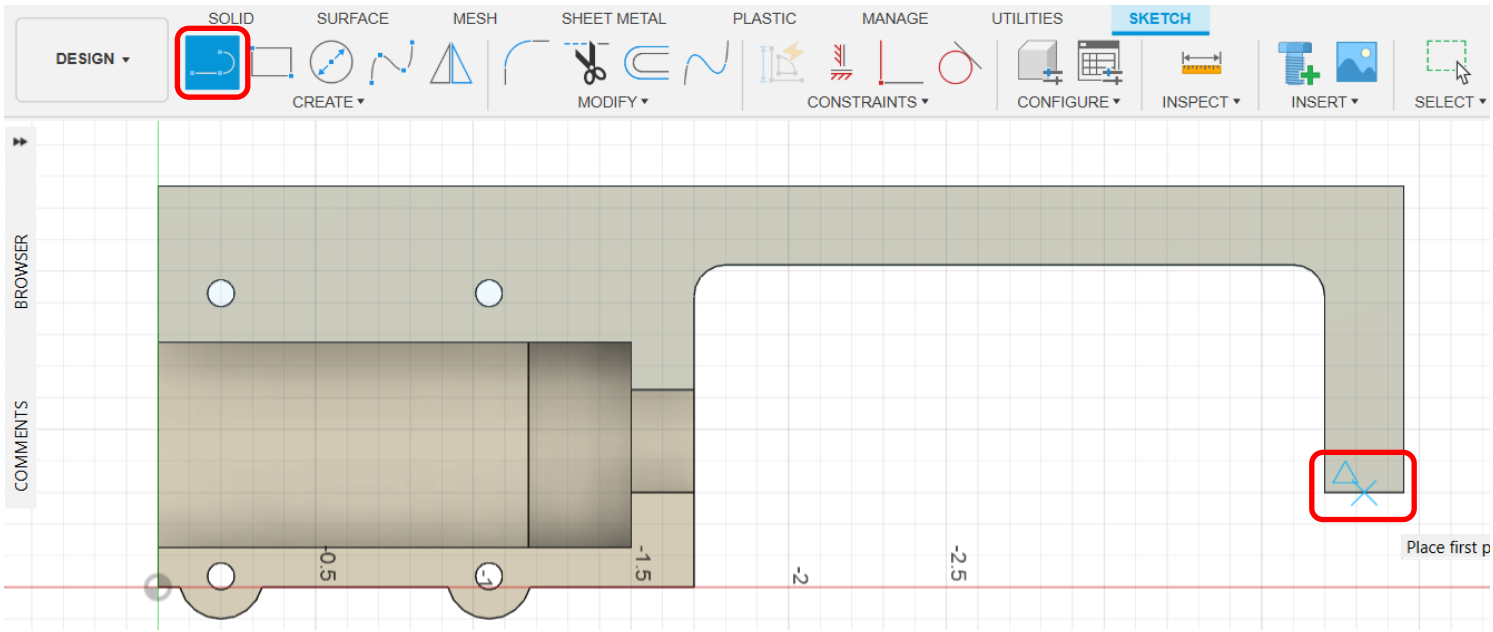
- open the BROWSER and click on the **eye** icon for the Hex Key to hide it



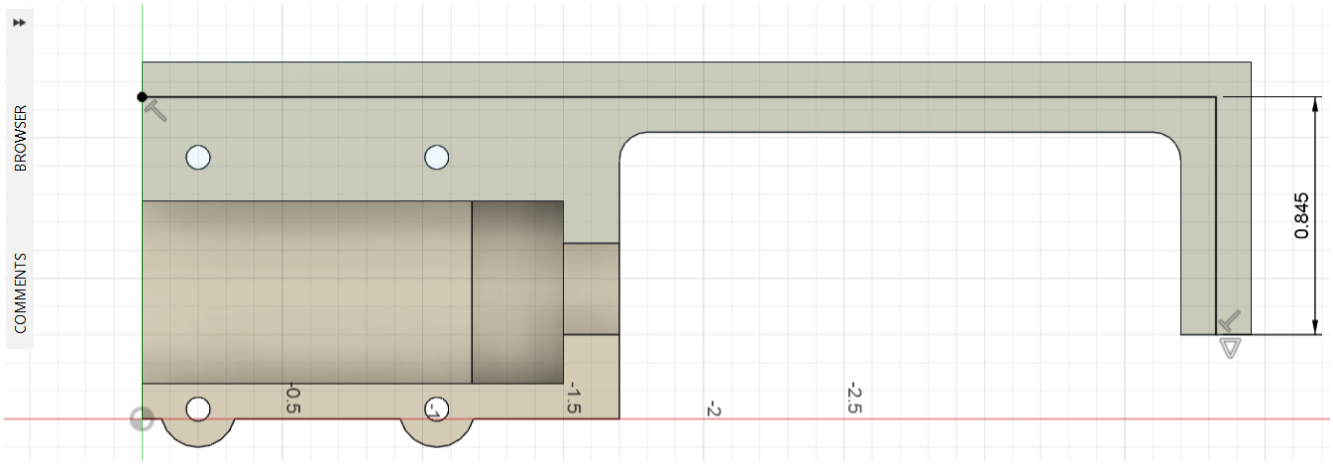
- select the **Create Sketch** tool and click on the Main Body surface



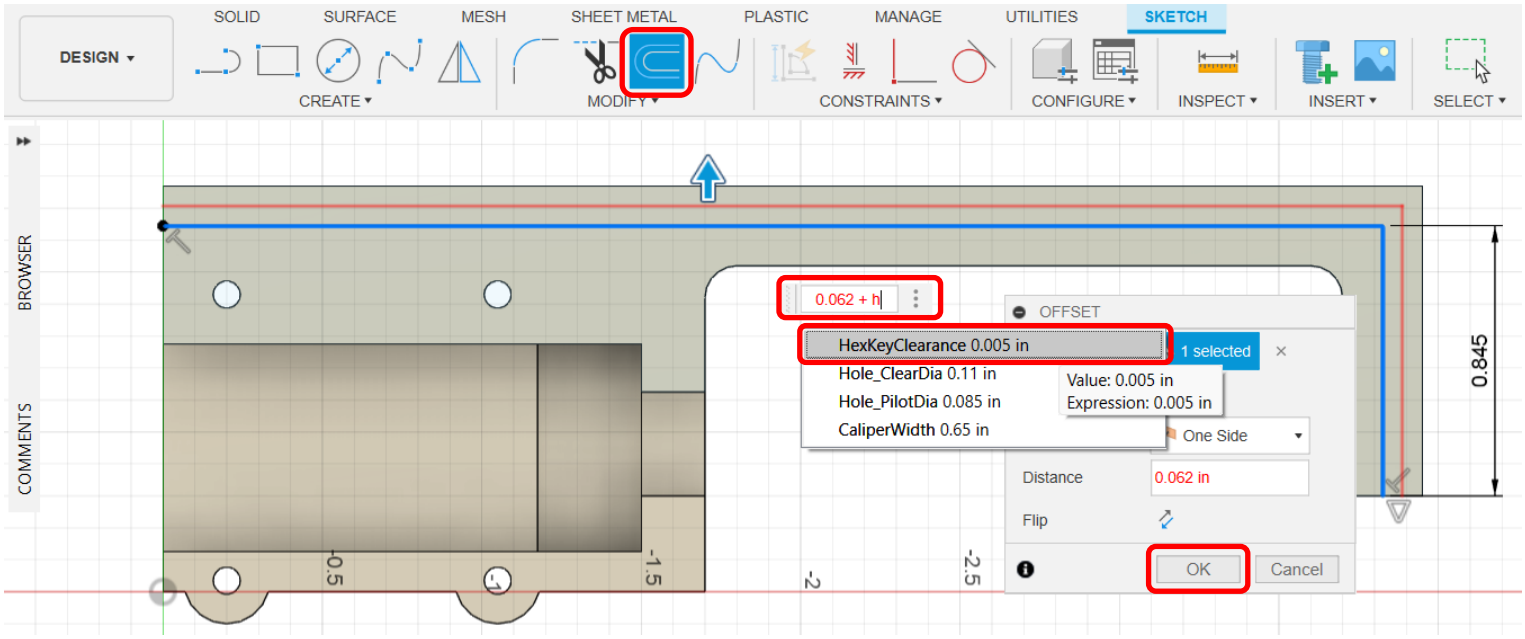
- select the **Line** tool and move the mouse over the lower end of the Main Body. When a **triangle** shows, which indicates the center of the edge, **click at that point** to start the line.



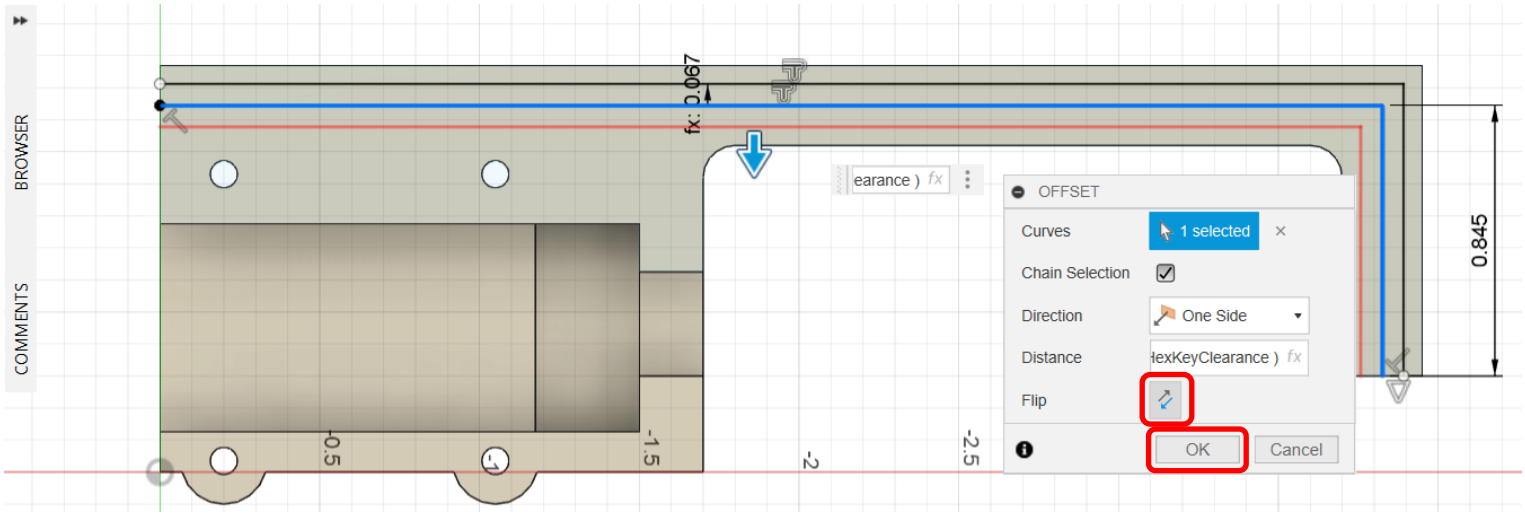
- extend the line upward by **0.845** and then create a horizontal line all the way over to the left edge



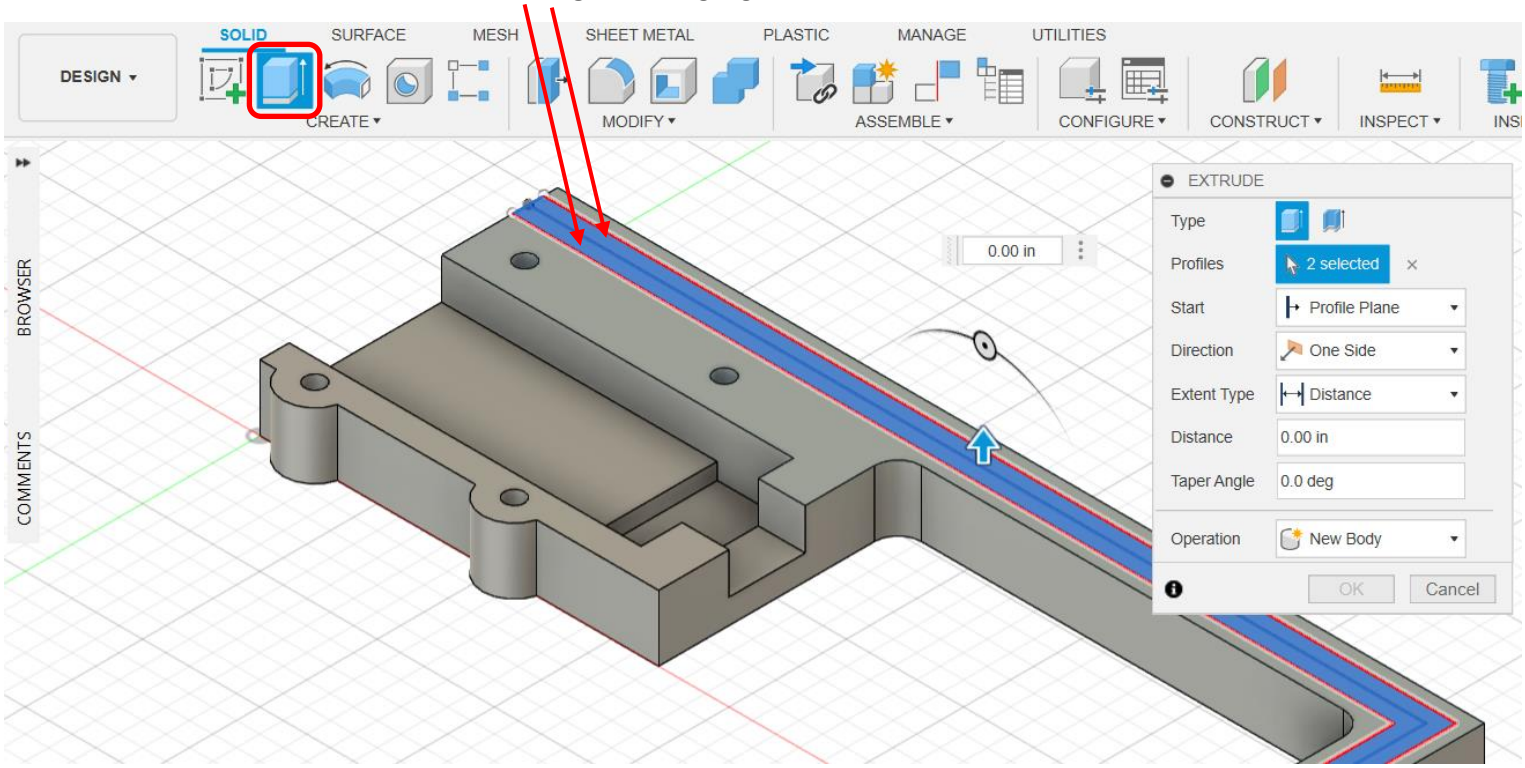
- select the **Offset** tool (if it is not visible find it in the MODIFY menu) and click on the horizontal line just created
- in the value box type **0.062 + h** and then select **HexKeyClearance** and click **OK**



- use the Offset tool again, type **0.062 + h** and then select **HexKeyClearance** as before and then click on the **Flip** icon, which should create a new line under the original line this time. Click **OK** and then click **Finish Sketch**.

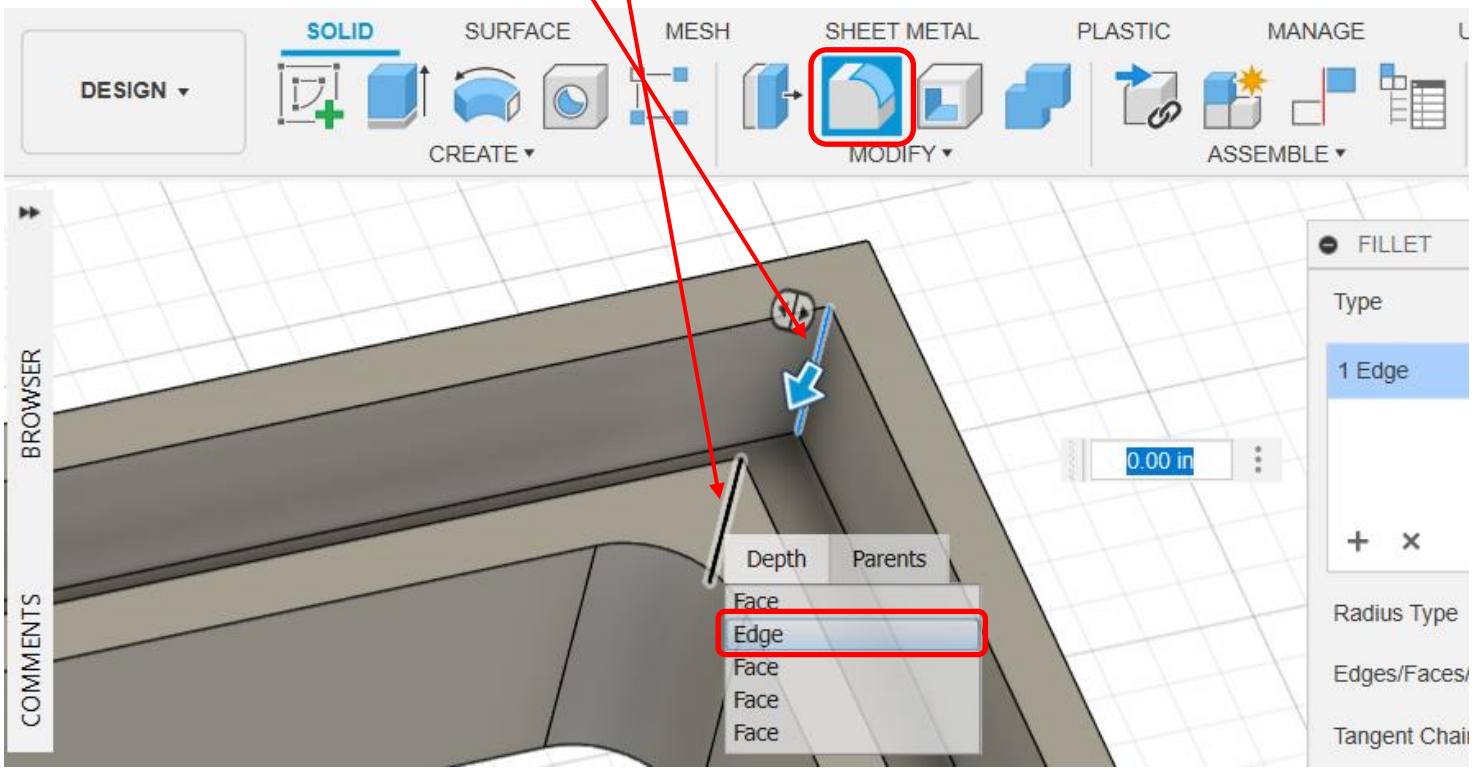


- select the **Extrude** tool and click on the **two regions** to highlight them in blue

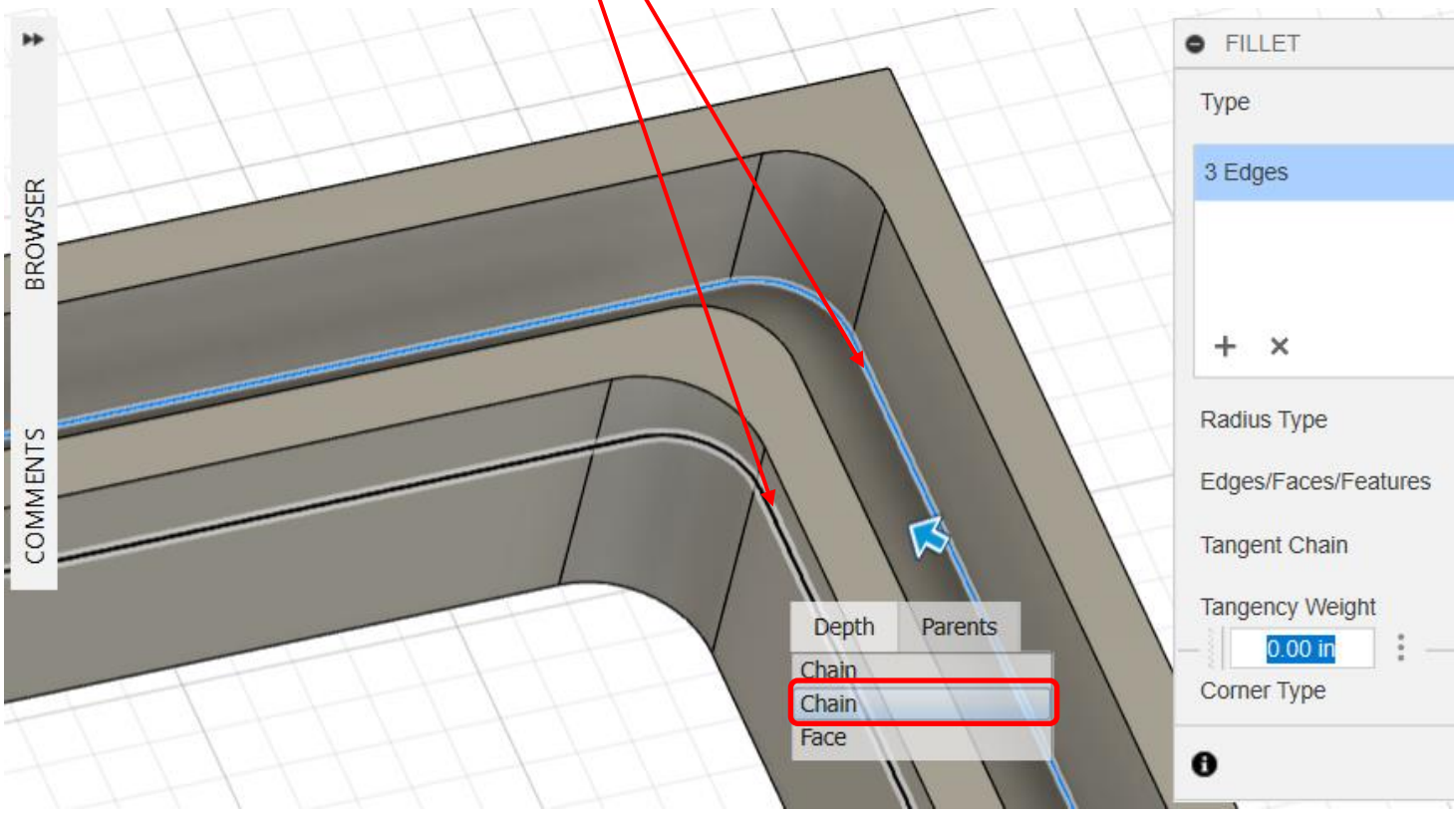


It is always a good idea to add fillets to inner edges to reduce stress concentrations, which can lead to cracking. Doing so also looks good.

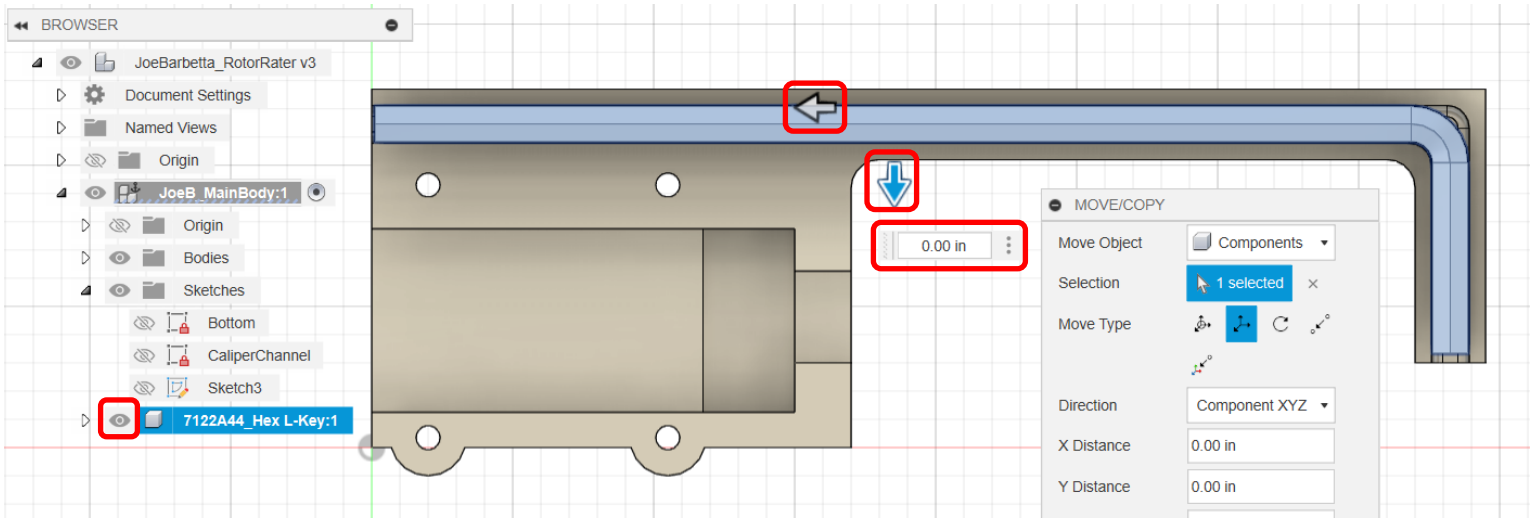
- zoom in to the corner of the channel just created
- select the **Fillet** tool and click on the **inner edge** to turn it blue
- hold the mouse button down at where the **opposite edge** is and select **Edge** from the list
- set the fillet value to **0.08** and click **OK**.



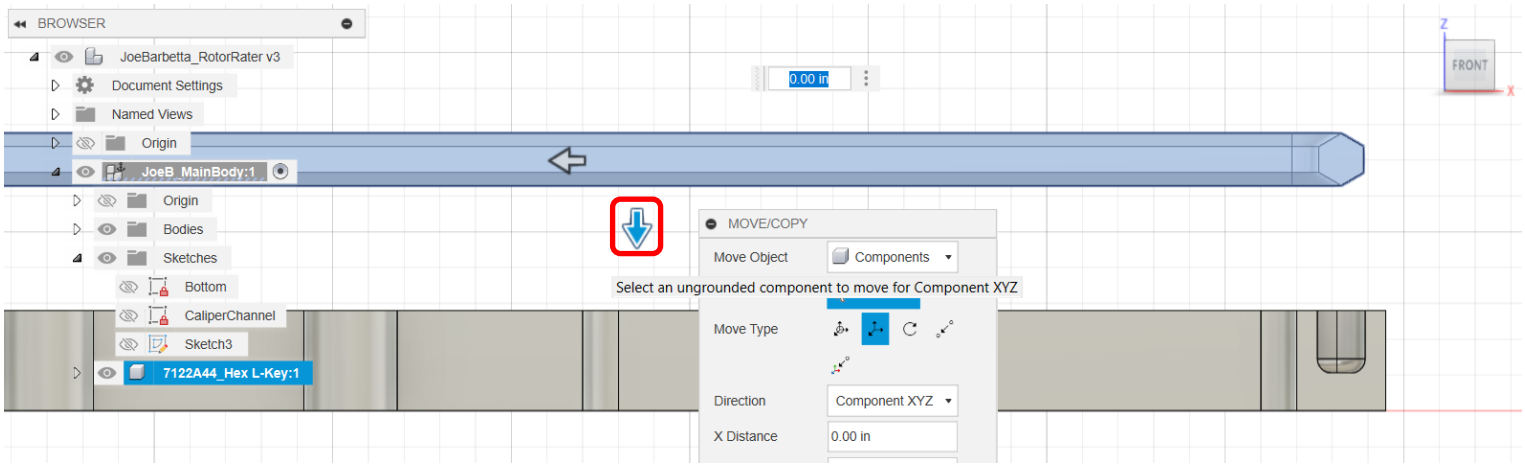
- select the **Fillet** tool again and click on the **inner edge** as shown
- hold the mouse button down at where the **opposite edge** is and select **Chain** from the list
- set the fillet value to **0.04** and click **OK**.



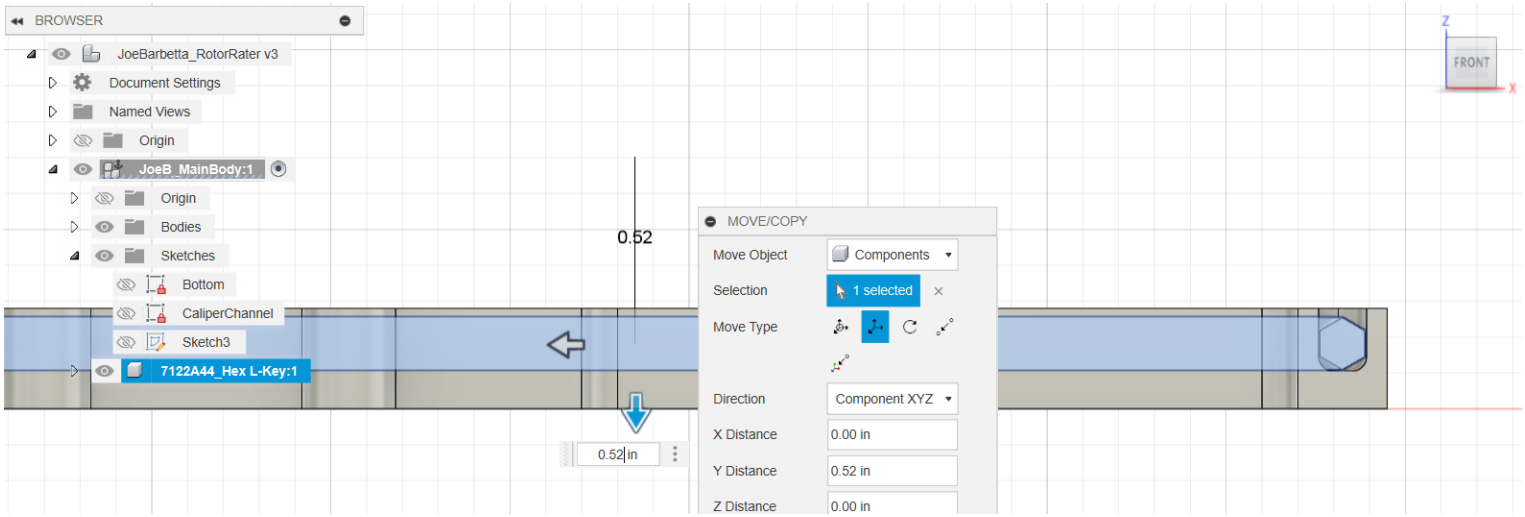
- click on the **View Cube** for a top view
- open the **Browser** and click on the **eye icon** for the Hex Key to make it visible
- select the **Move/Copy** tool and click on the horizontal and vertical arrows and enter values if needed to position the Hex Key over the slot.



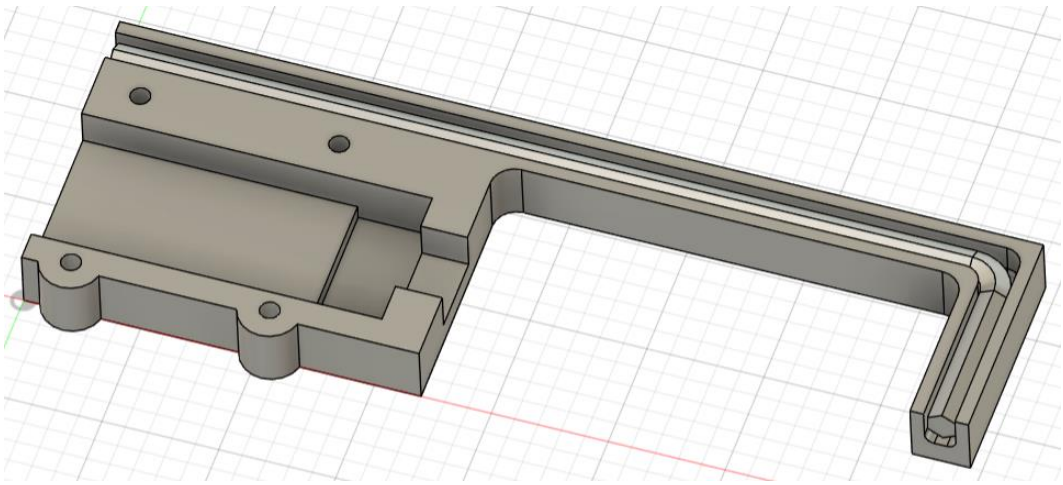
- click on the **Front** of the **View Cube**
- select the **Move/Copy** tool and click on the Hex Key
- use the vertical arrow to move the Hex Key down into the slot



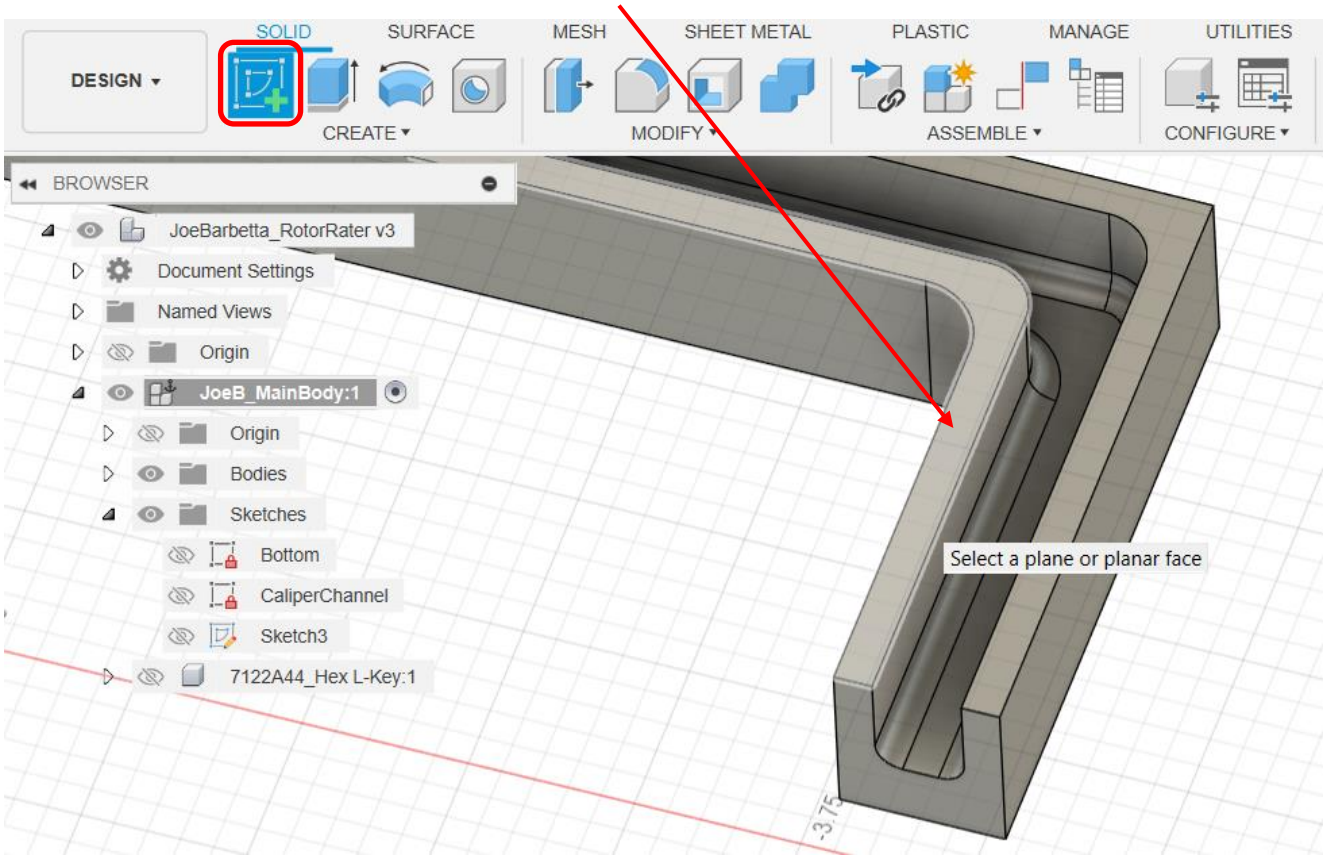
- after moving the Hex Key down and fine tuning the movement by typing **0.52** in the value box, allowed the Hex Key to lay in the bottom of the slot. This value may differ.



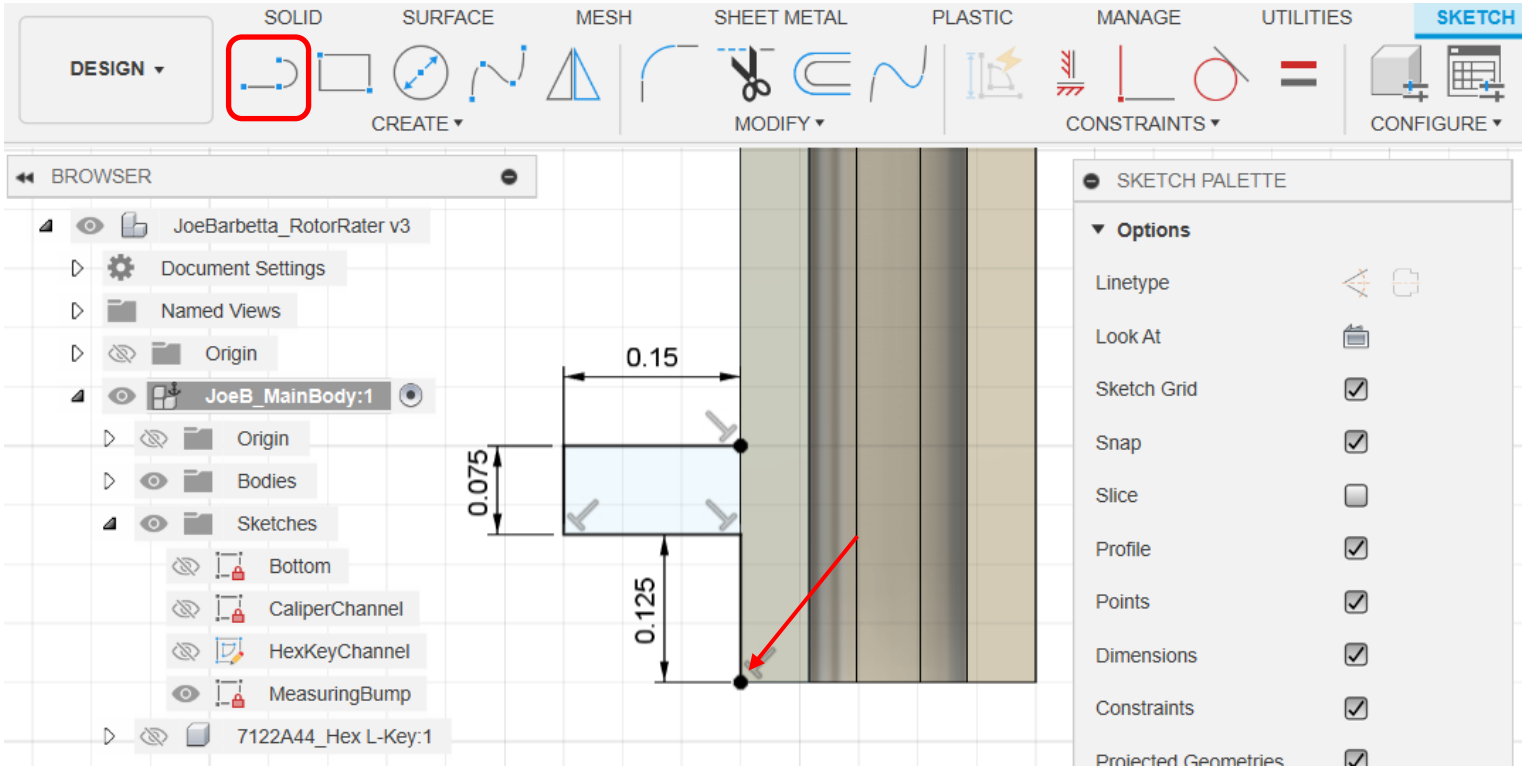
This positioning is not critical and is only to illustrate the placement of the Hex Key. This is why more advanced alignment features were not used.



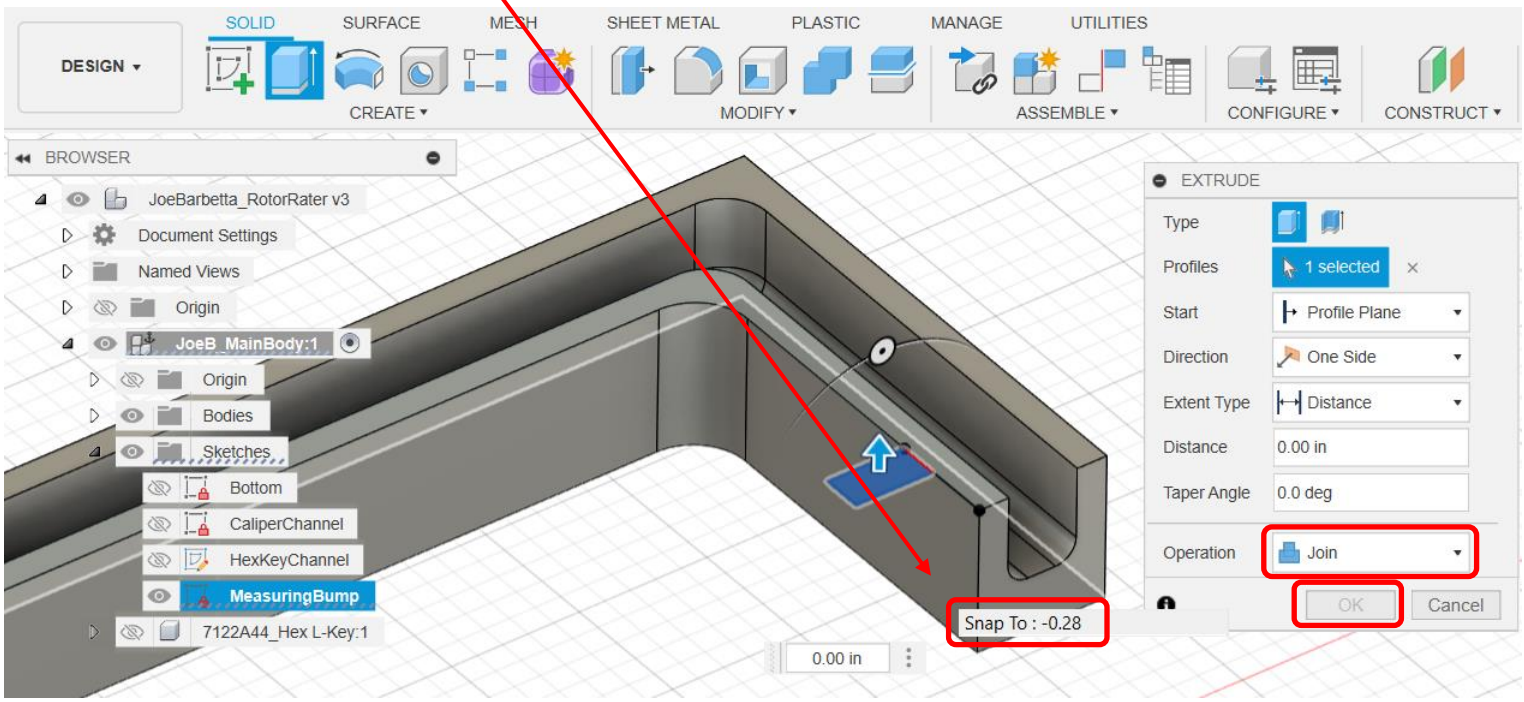
- using the eye icon in the BROWSER to hide the hex key
- select the **Create Sketch** tool and click on the **top surface**



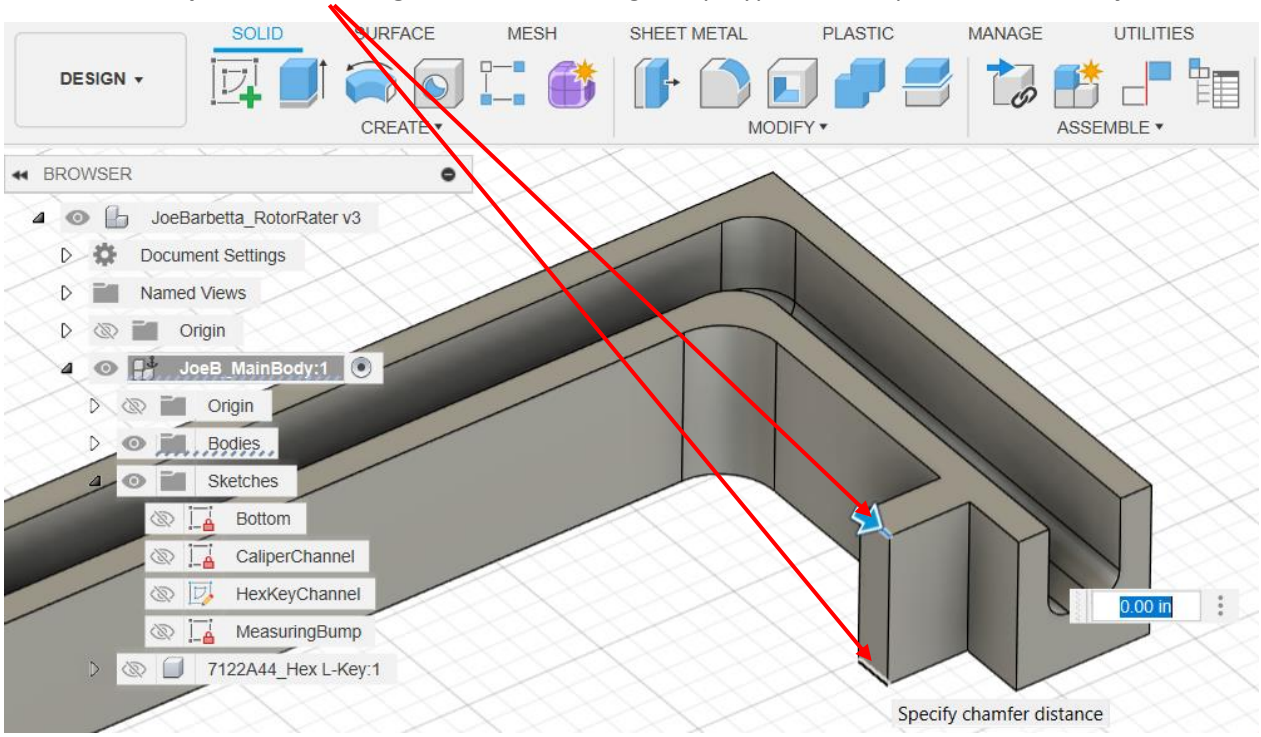
- select the **Line** tool and start the first line at the **bottom corner indicated by the red arrow**
- create the lines as shown below and click **Finish Sketch**



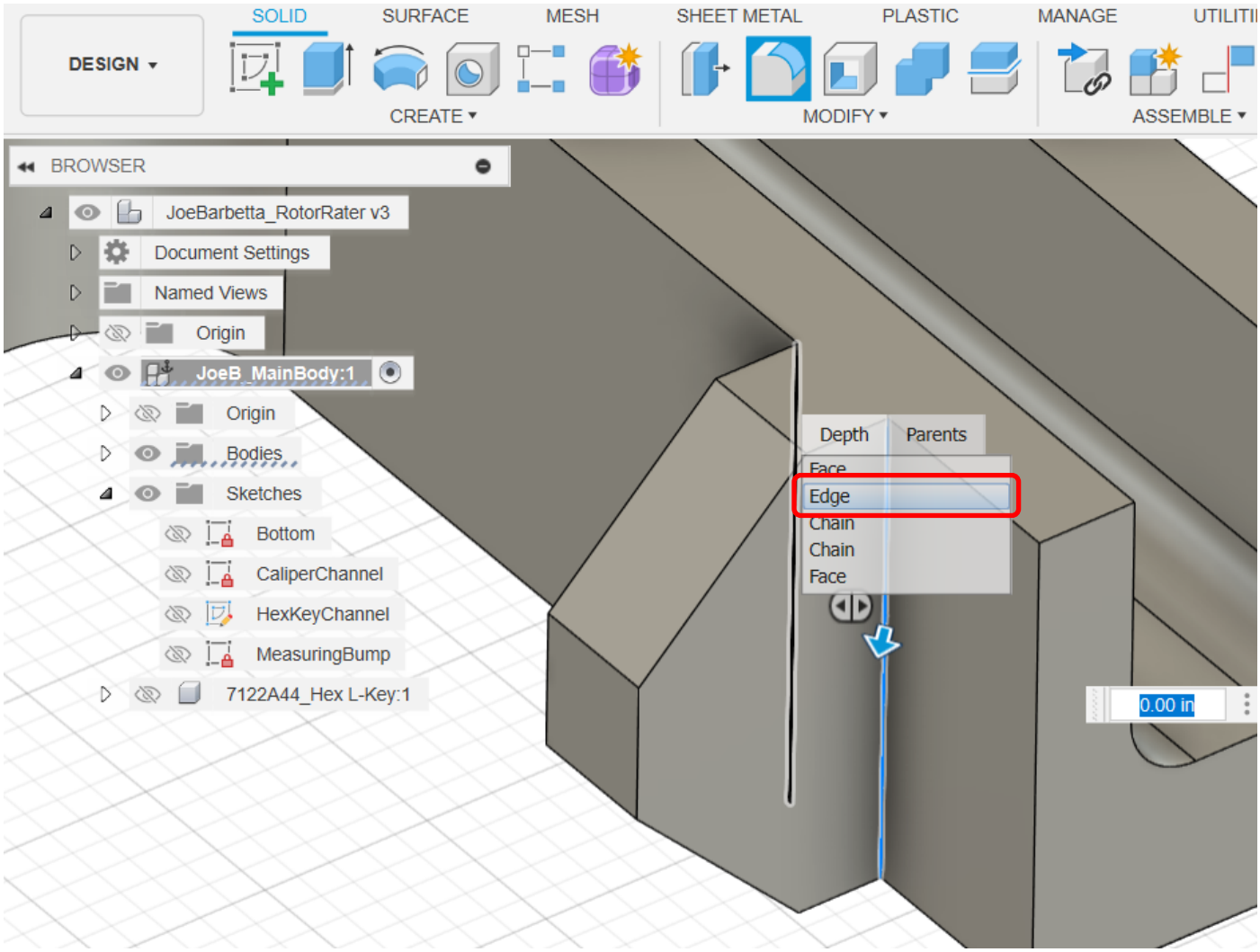
- adjust the view similar to that below
- select the **Extrude** tool, click on the **rectangle** just created, and ensure that the **Operation** is set to **Join**
- move the mouse to where the **red arrow** is pointing, which should cause a **Snap To: -0.28** box to show
- click at that point and then click **OK**



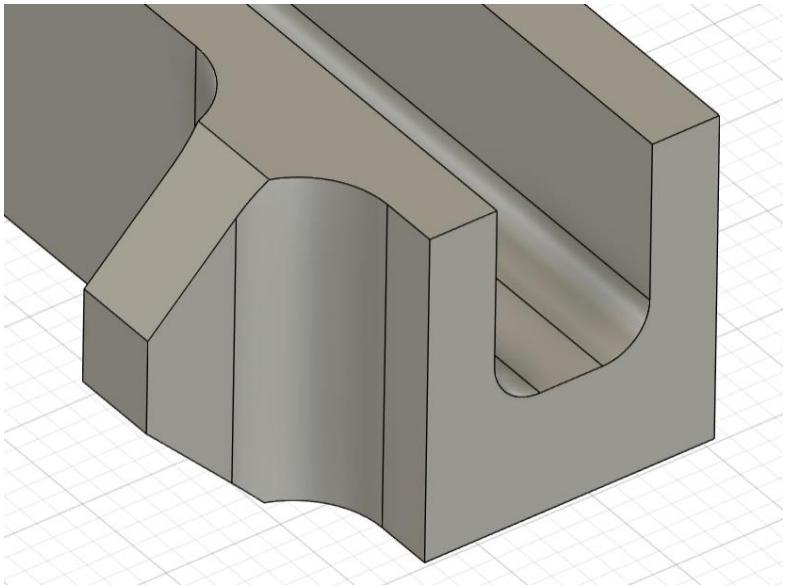
- from the MODIFY menu select the **Chamfer** tool
- click on the **top and bottom edges** of the measuring bump, type **0.1**, and press the **Enter key**



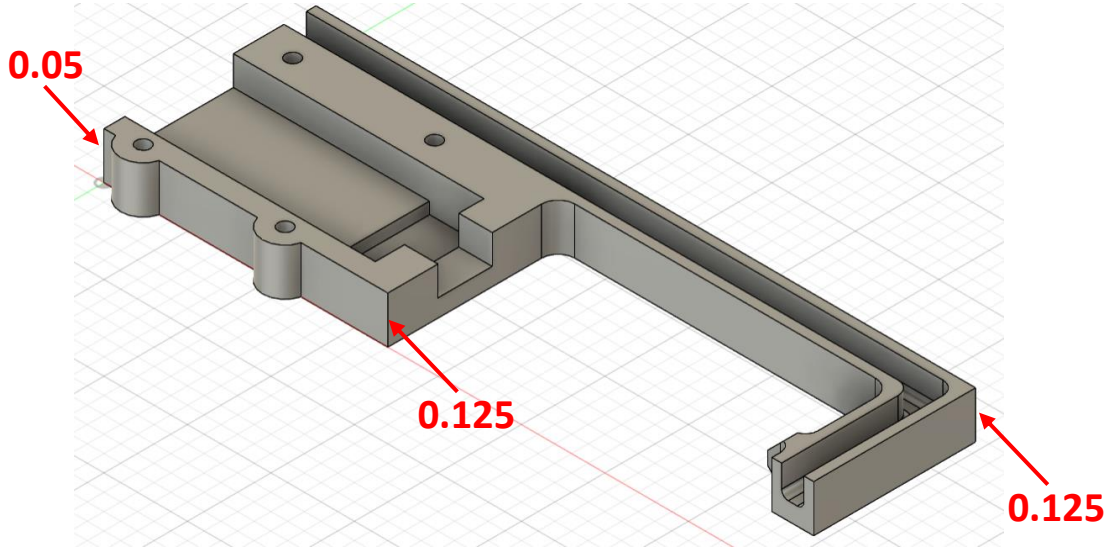
- select the **Fillet** tool and **select the edges** as shown. As done previously a hidden edge can be selected by holding the mouse button down where the edge should be and then selecting the item that highlights the edge.
- enter **0.075** for the fillet



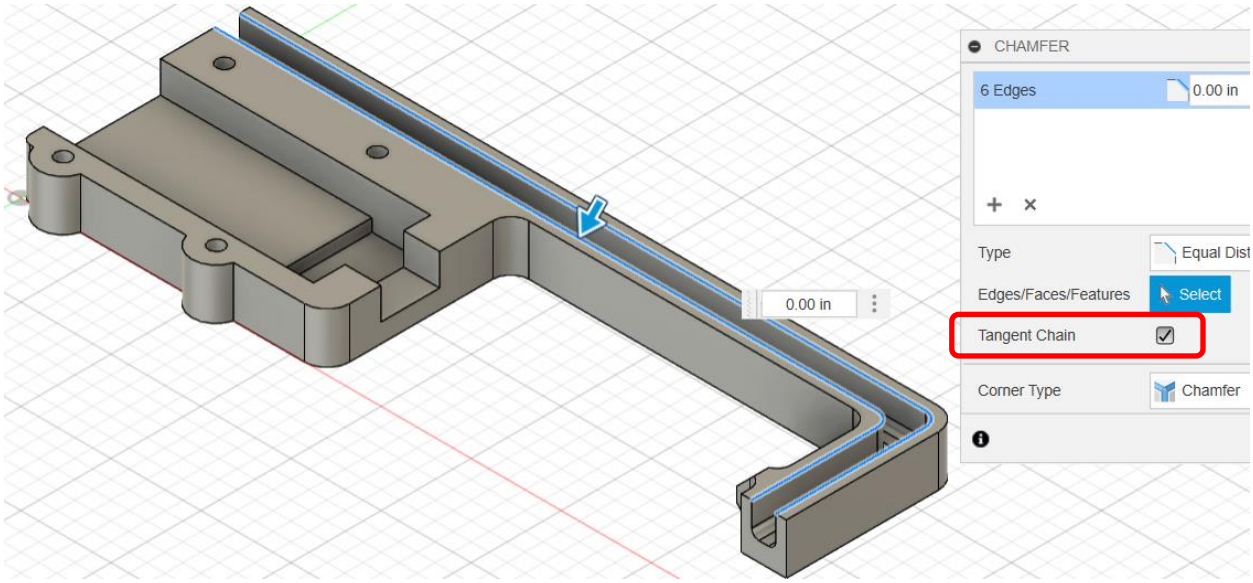
The measuring bump should look like that below.



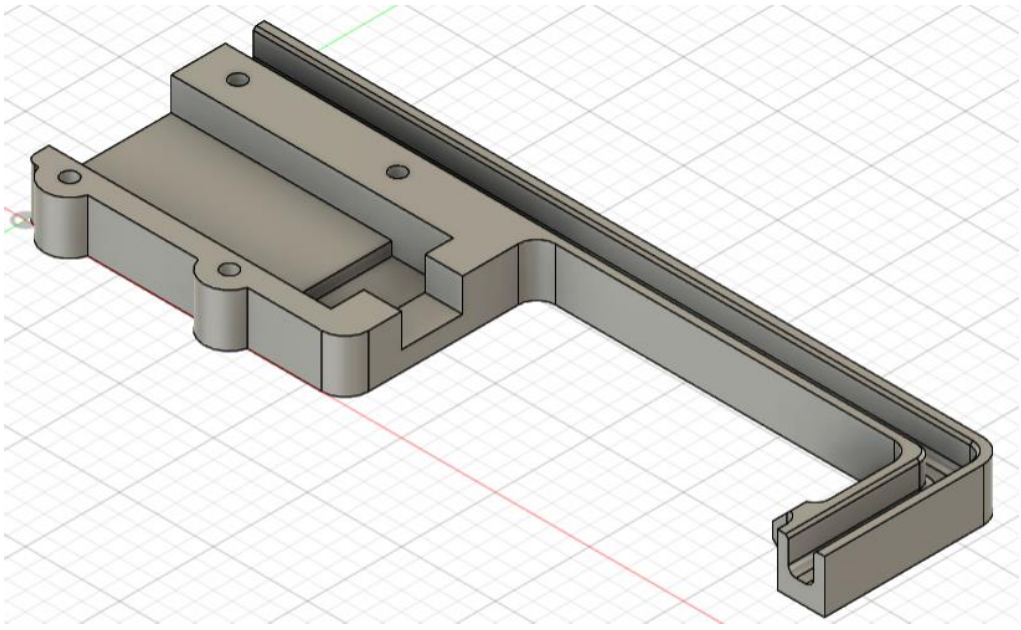
- return to the Home view and add **Fillets** to the three edges as indicated. This is mostly for aesthetics.



- add Chamfers of **0.015** to the edges highlighted in blue. When **Tangent Chain** is checked, which should be the default, clicking on an edge should highlight any arcs and other lines that are part of the “Chain”.

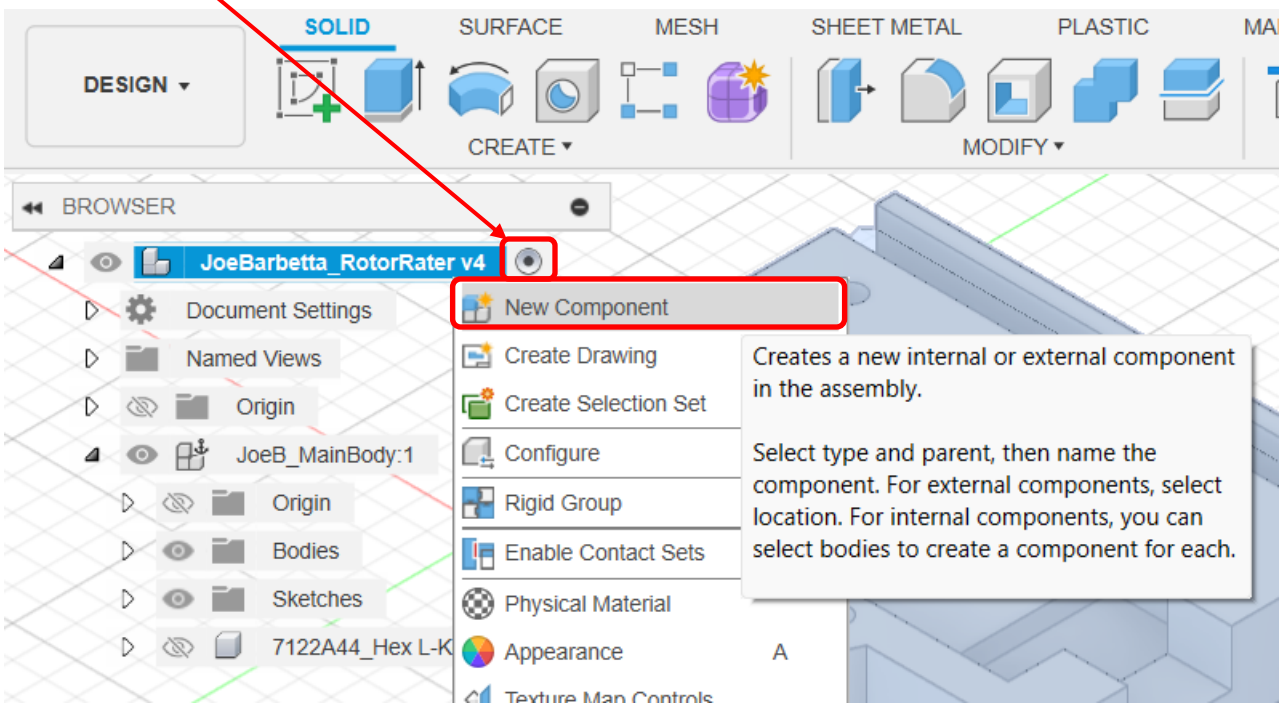


The Main Body should now look like that below.

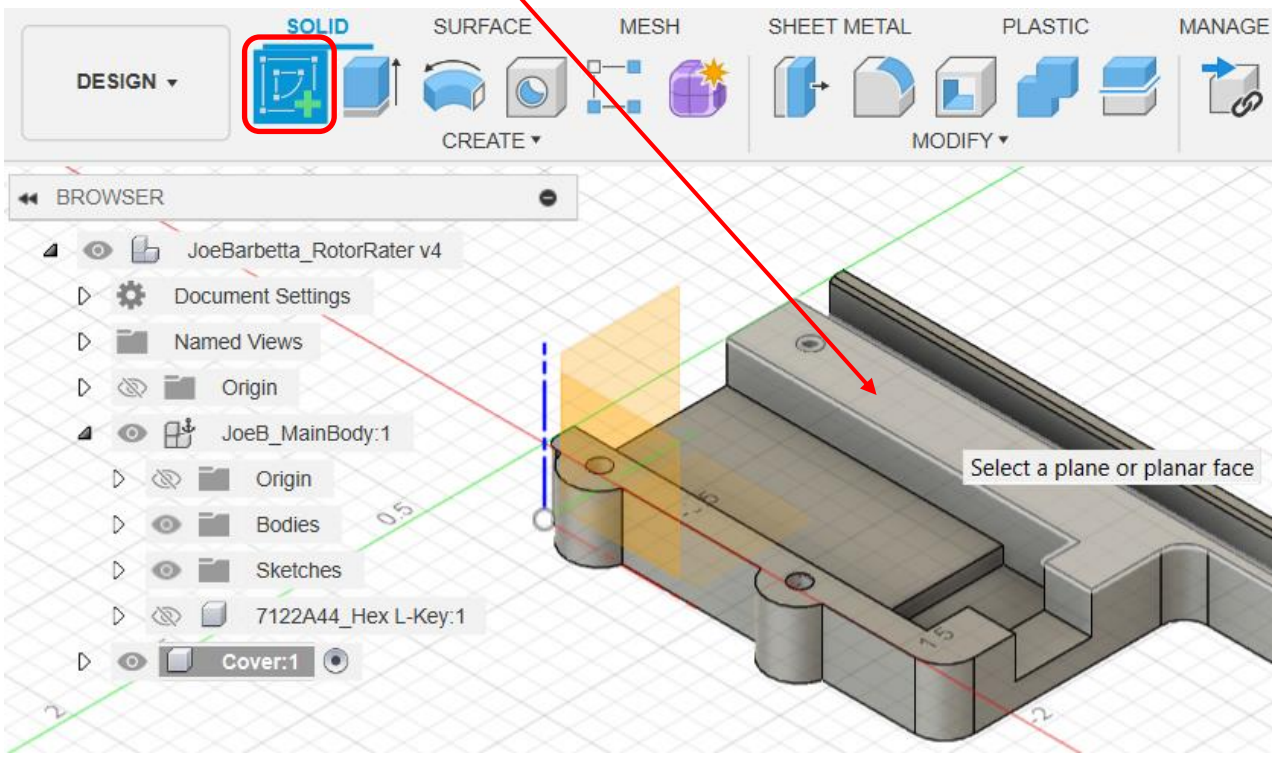


Creating the Top Cover

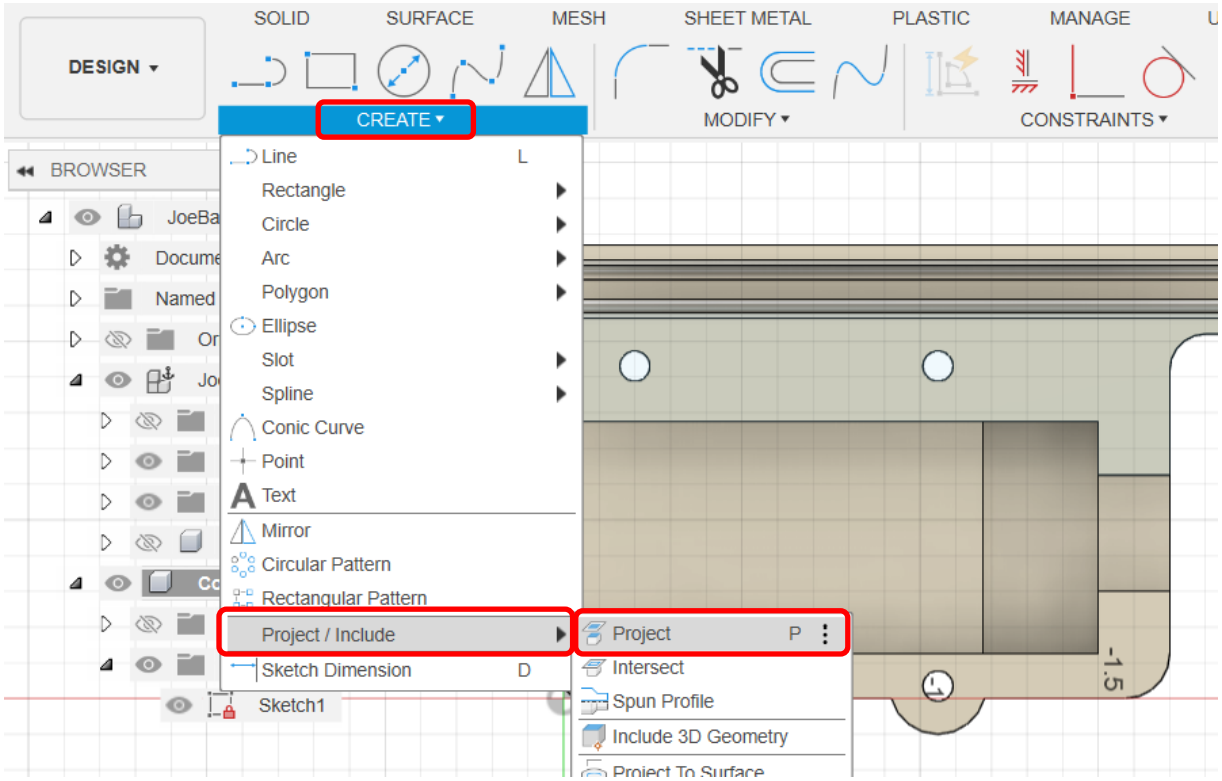
- click the **circle** next to the project name to activate it
- right-click on the name and select **New Component** and name it **Cover**



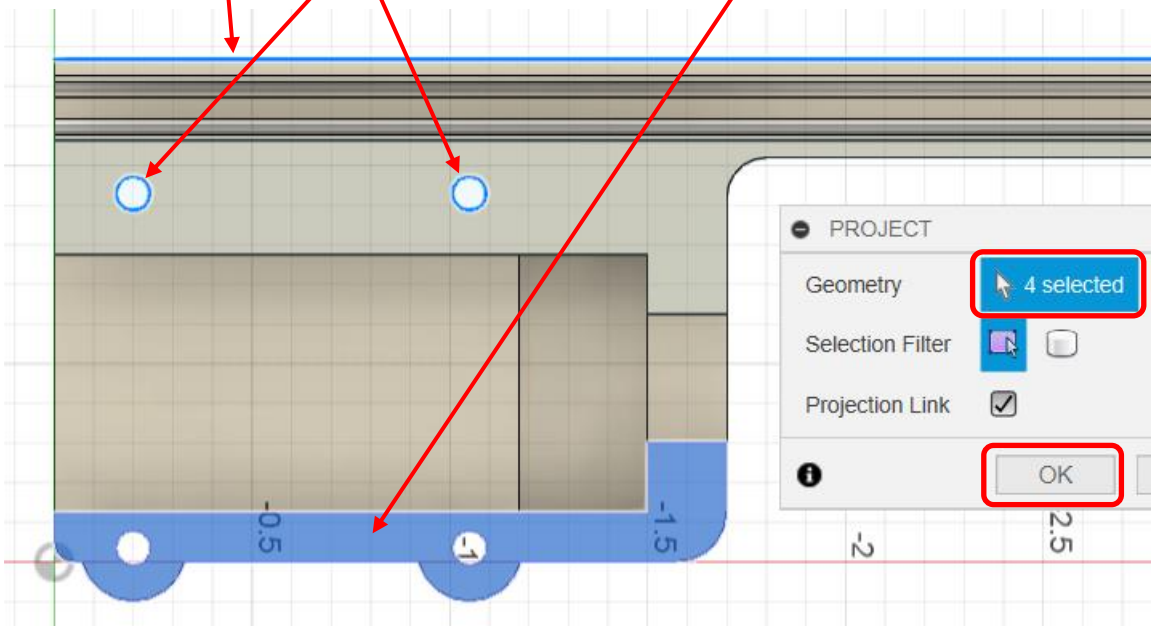
- note that the new component **Cover** now appears in the BROWSER and it is **activated**.
- select **Create Sketch** and click on the **surface** as indicated



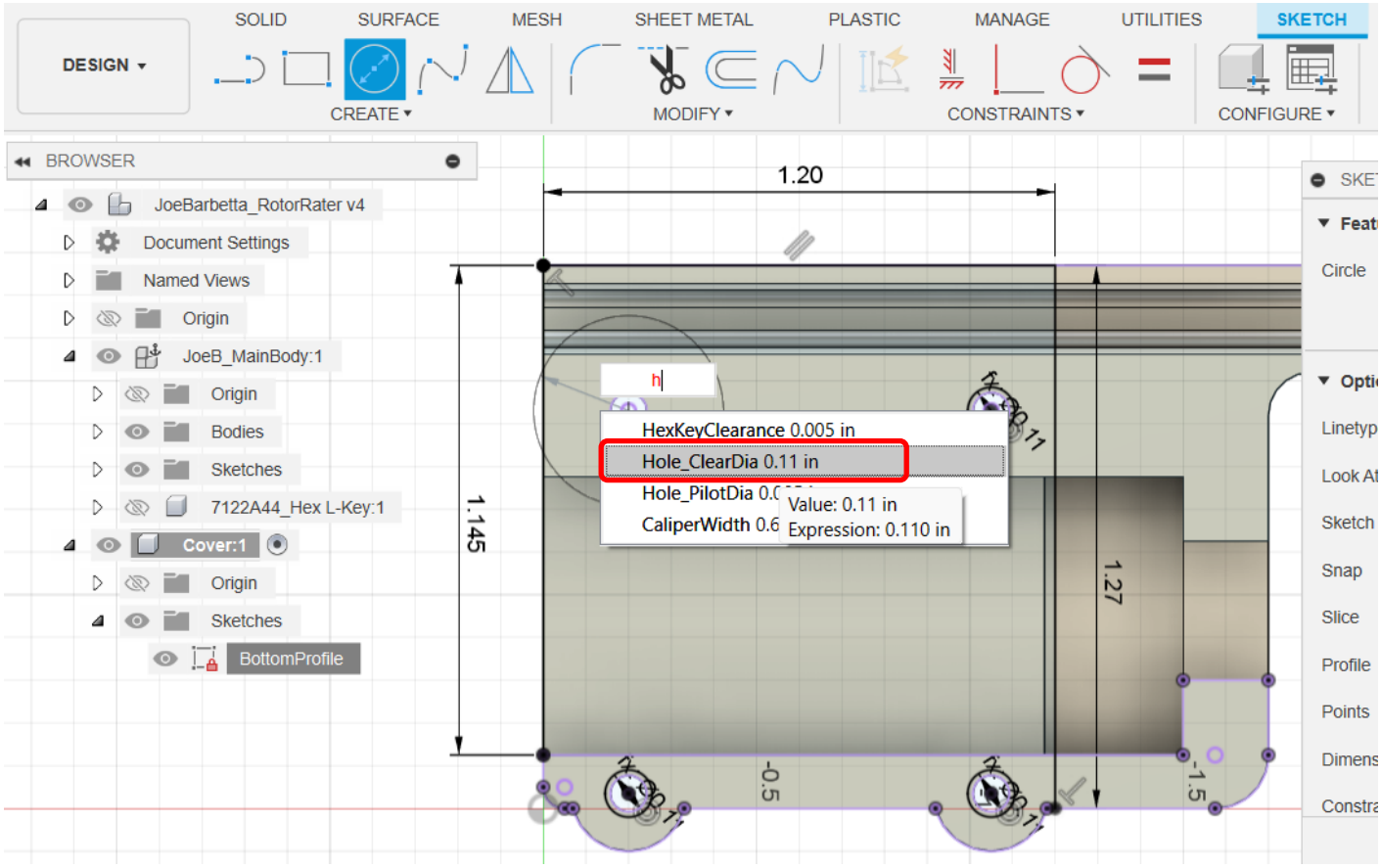
- from the **CREATE** menu select **Project/Include** and **Project**



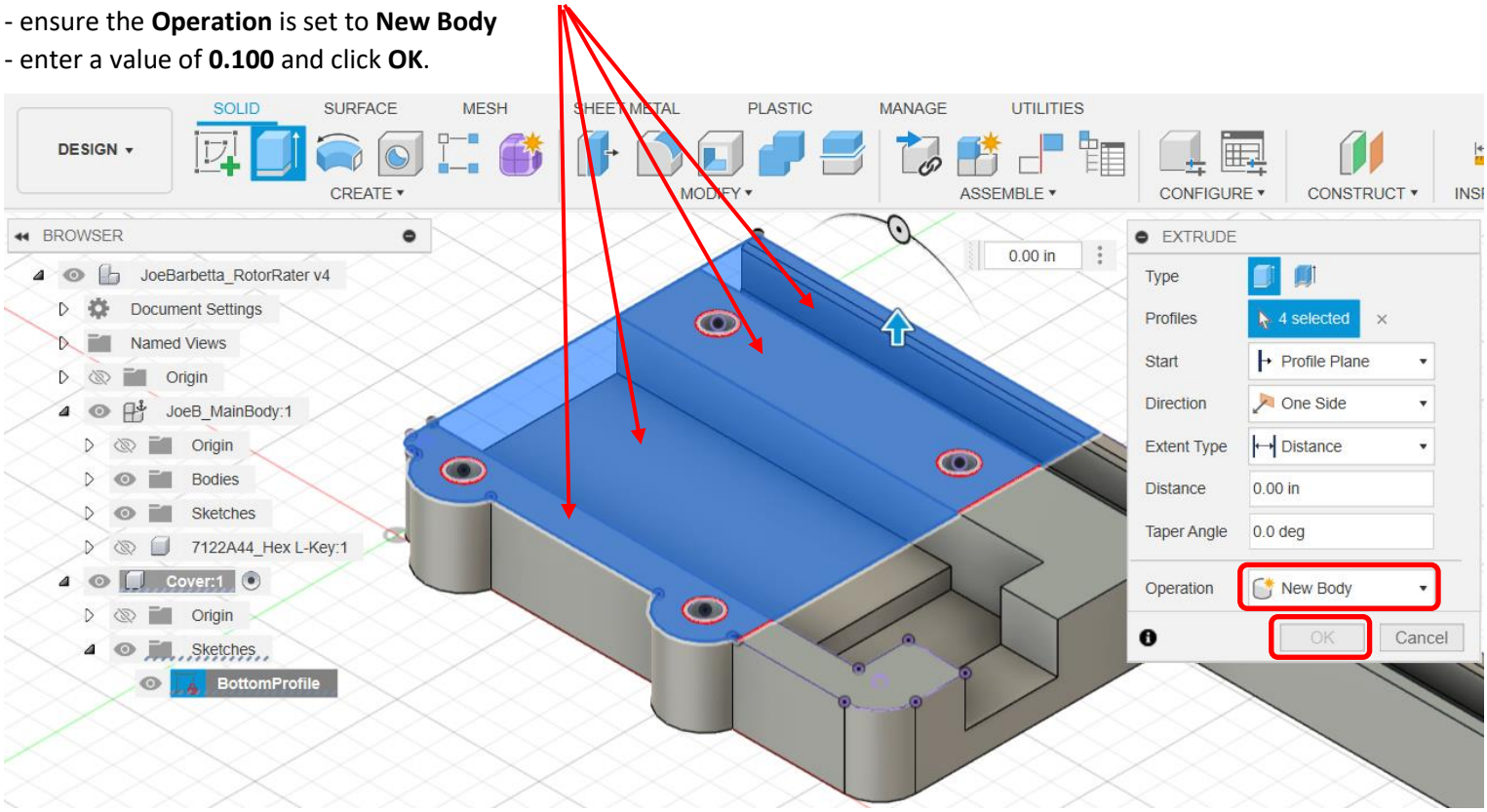
- click on the **top edge**, the **edges of the two holes**, and the **lower surface**, which should highlight them blue. Note that the PROJECT window should indicate **4 selected**. Click **OK**.



- select the **Line** tool and starting from the **top-left corner** create a **line to the right** (**1.20**) and then a line down.
- from the **top-left corner** also create a **line downward**
- select the **Circle** tool, click on the **center point** of a circle, extend it out, type **h**, and select **Hole_ClearDia**, and press the **Enter**.
- repeat the above for the other 3 circles. Each violet circle should now have a slightly larger black circle around it.
- click **Finish Sketch**

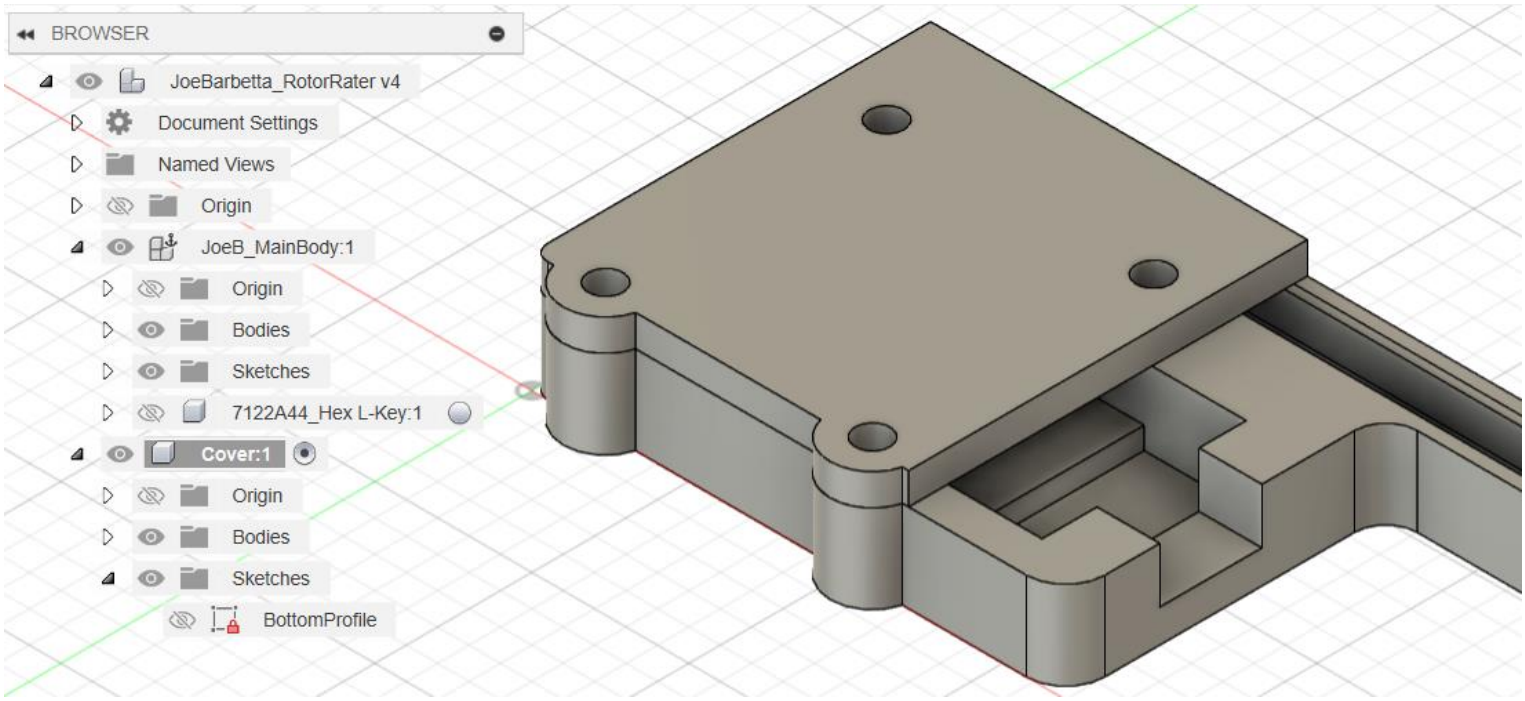


- select the **Extrude** tool and **click on the four regions**, which should result in the blue region.
- ensure the **Operation** is set to **New Body**
- enter a value of **0.100** and click **OK**.

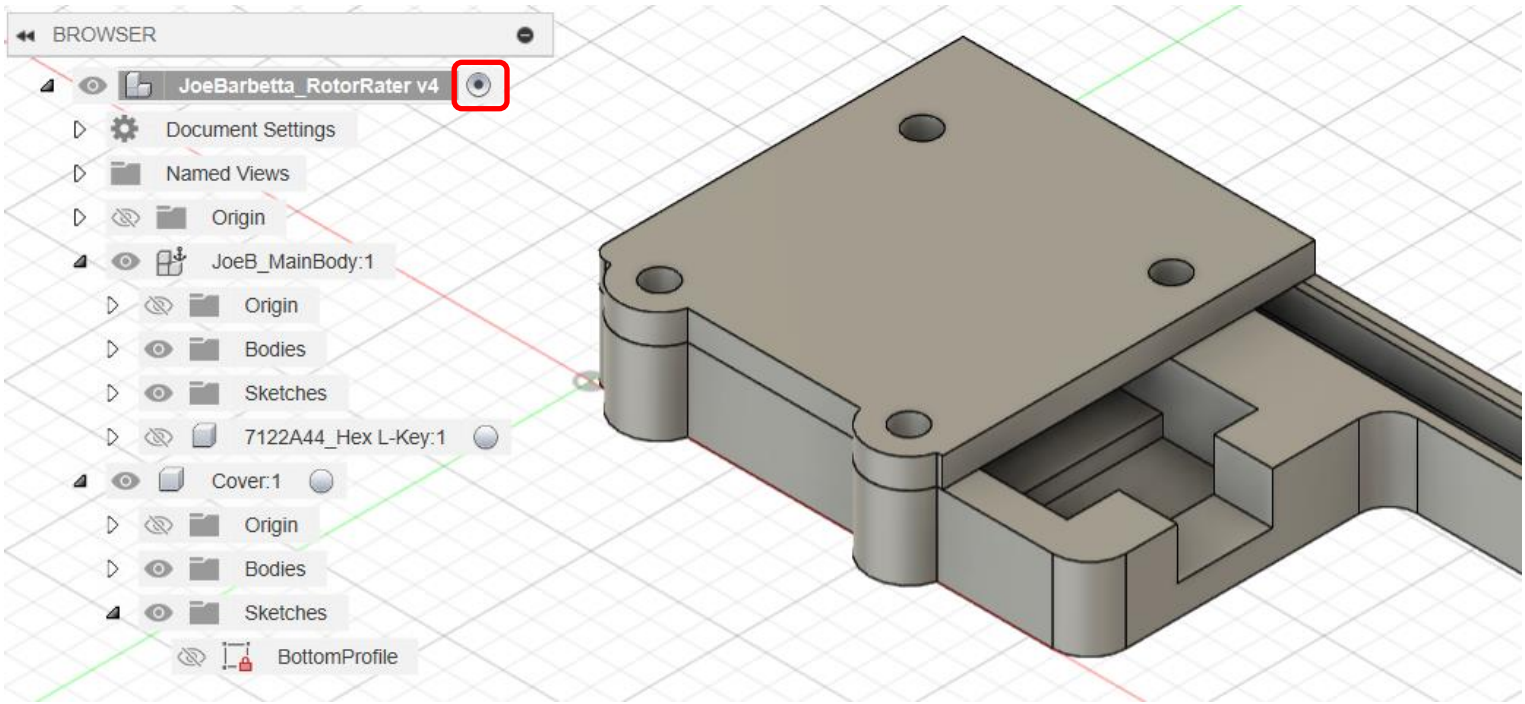


At this point the Cover should look like that below.

- ensure that the last Sketch is renamed to **BottomProfile**

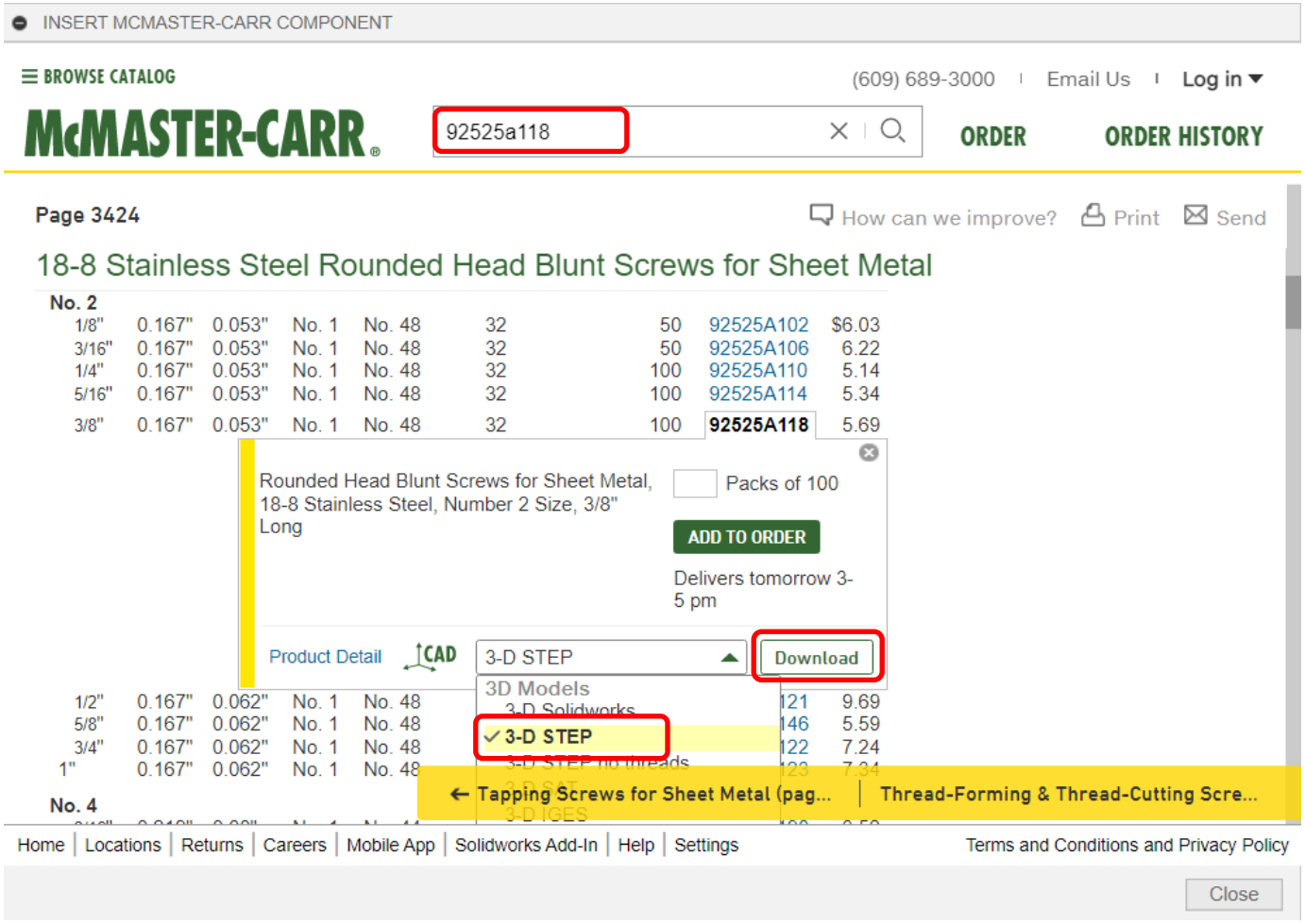


- reactivate the project name

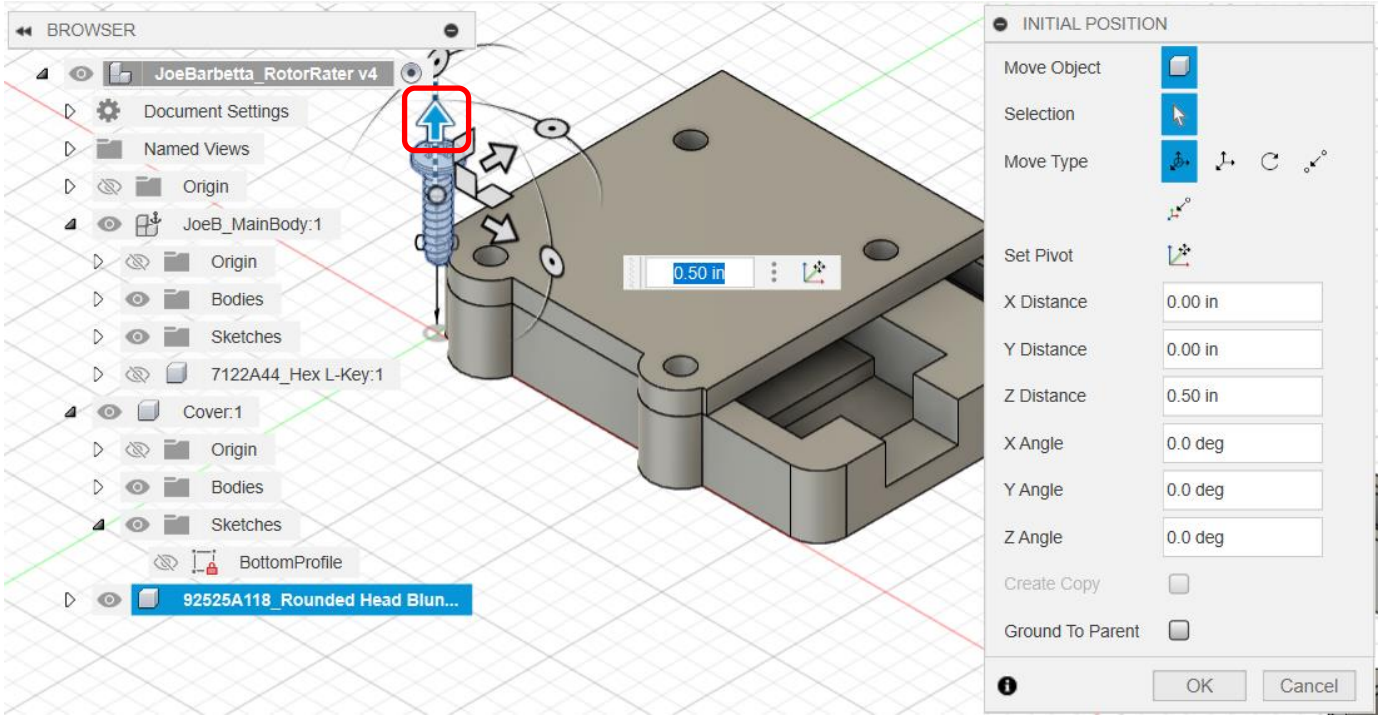


Adding Screws from McMaster-Carr

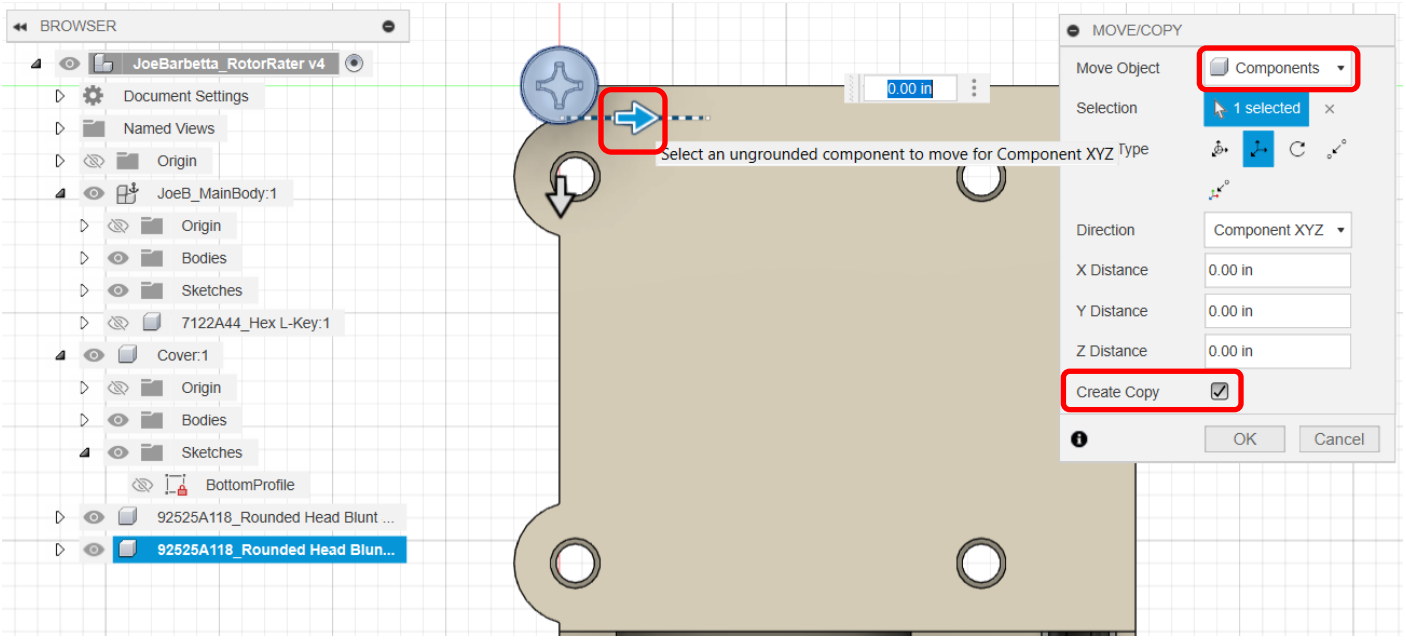
- from the top-right INSERT menu select **Insert McMaster-Carr Component**
- enter **92525A118** in the search box, select **3-D STEP**, and click **Download**



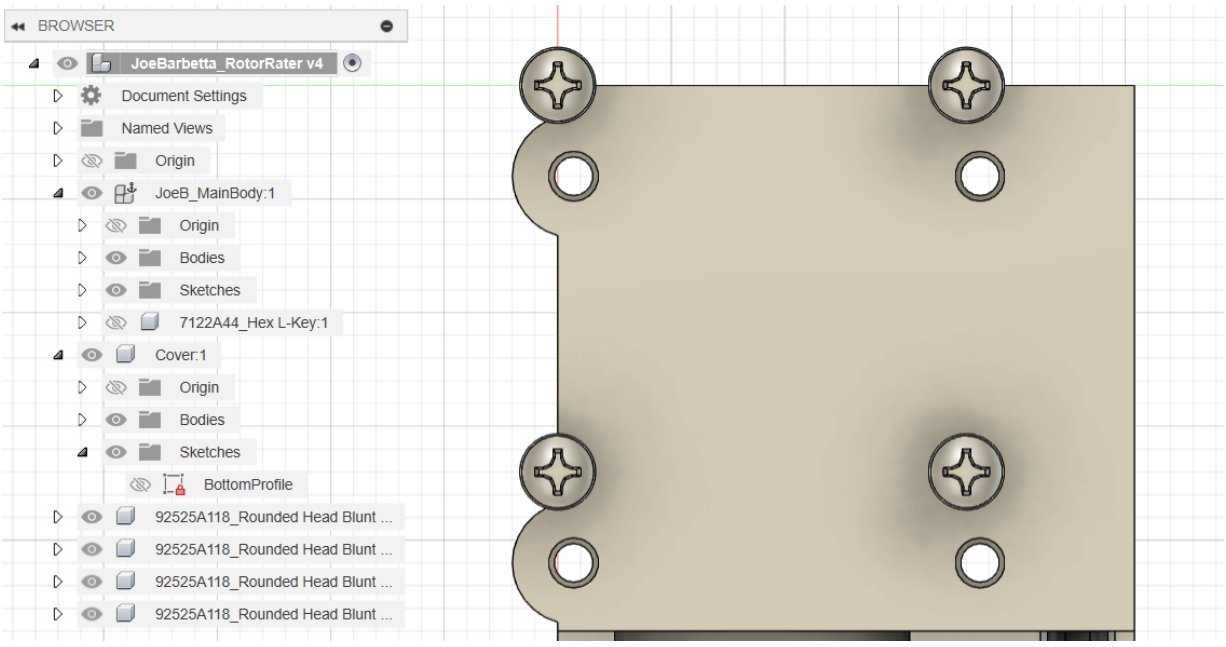
- use the up translation **arrow** to move the screw up by at least **0.5**.



- from the **MODIFY** menu select the **Move/Copy** tool
- ensure the Move Object is set to **Components** and that **Create Copy** is checked
- drag the **blue arrow** to move the screw near the hole to the right and click **OK**

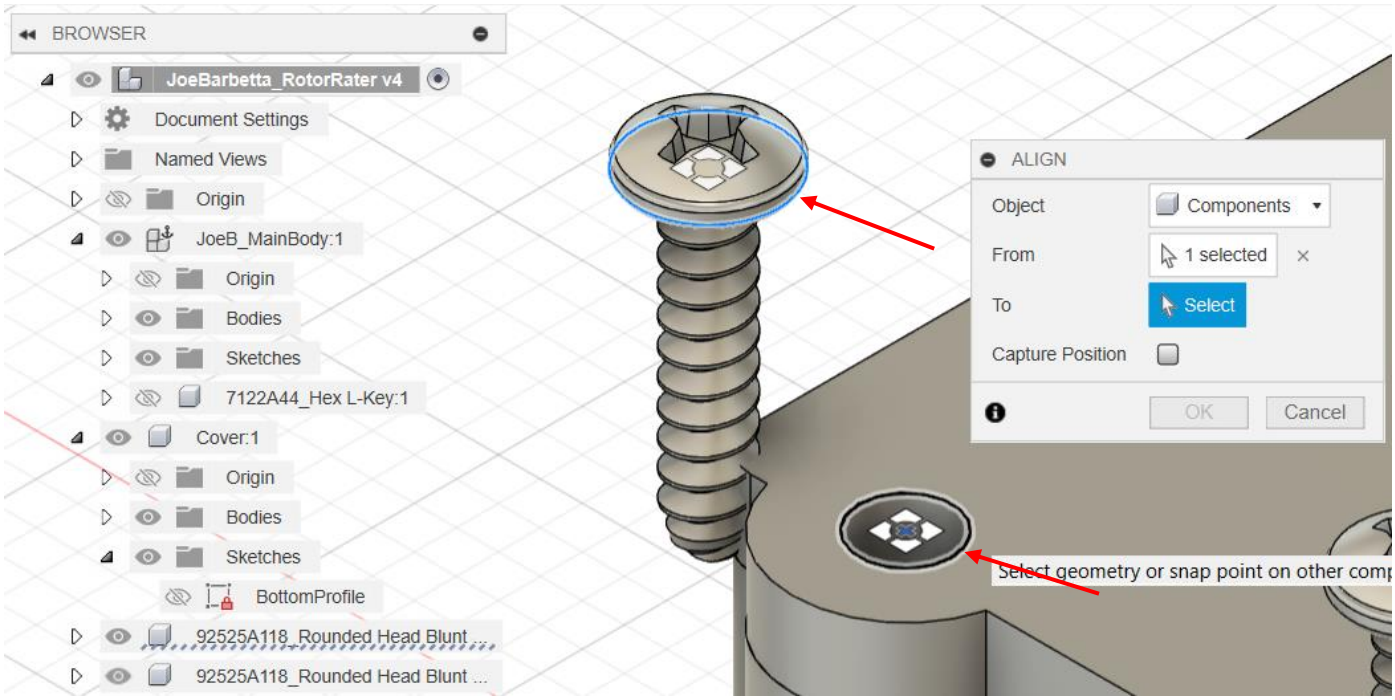


- perform this operation two more times to have a screw near each hole

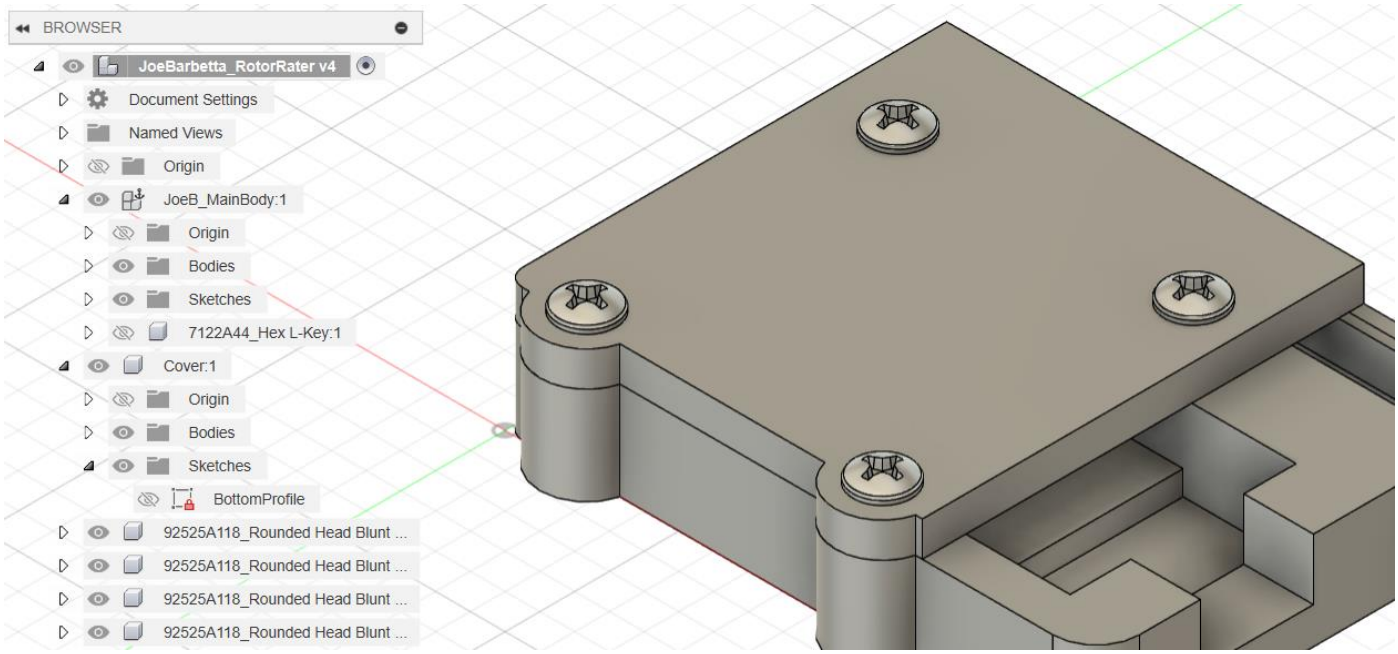


Using the Align tool

- return to the **Home** view and zoom in to one screw and hole
- from the **MODIFY** menu select the **Align** tool
- select the **bottom edge** of the screw head and then the **edge** of the hole.

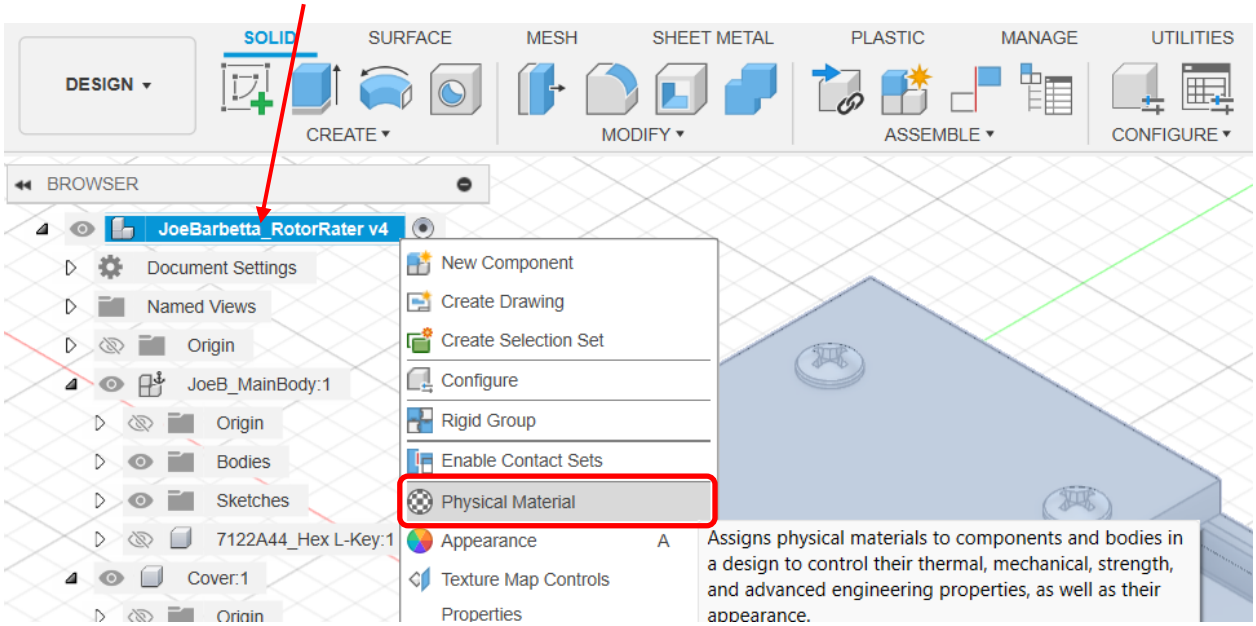


- perform the same Align operation for the other screws



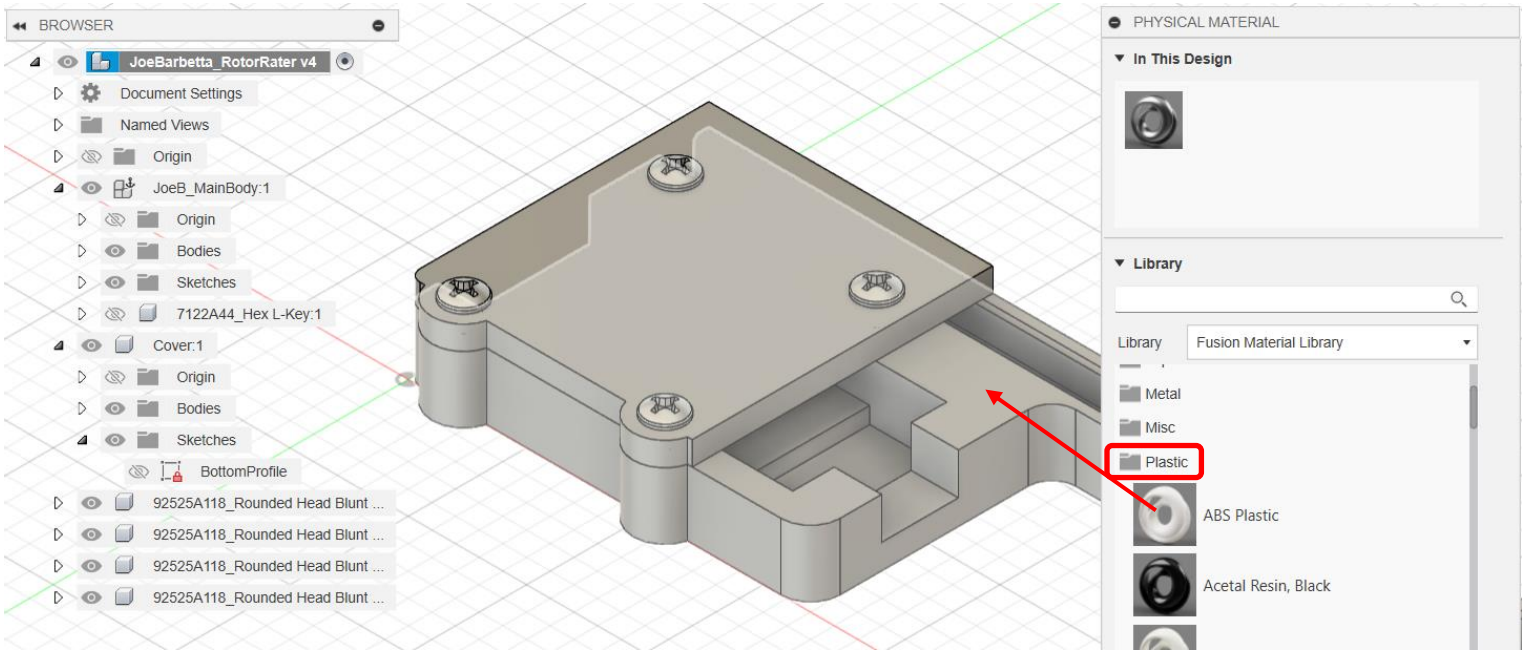
Adding Color

- right-click on the deisgn name and select **Physical Material**

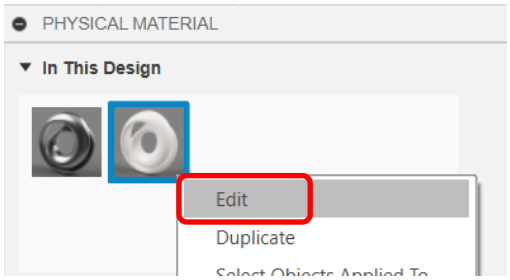


- scroll down to the **Plastic** folder and click on it
- drag the **ABS Plastic** icon onto the main body

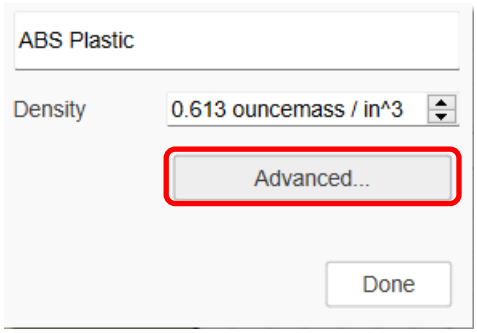
The rotor rater will likely be printed using PLA and the material and its color depends on the filament in the printer. Here we are assigning a material just to add some color to our design. Fusion does use the actual Material properties when Fusion displays the mass of an object and when the Fusion Simulation mode is used.



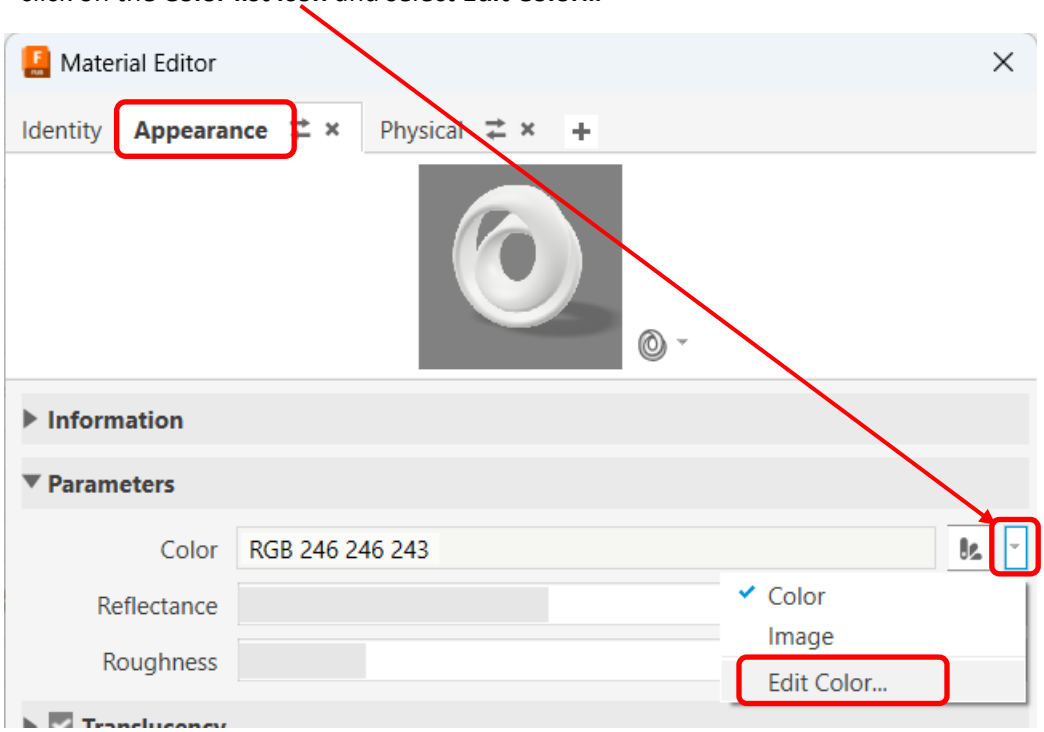
- right-click on the plastic material icon and select **Edit**
The icon on the left is steel, which Fusion uses as a default material.



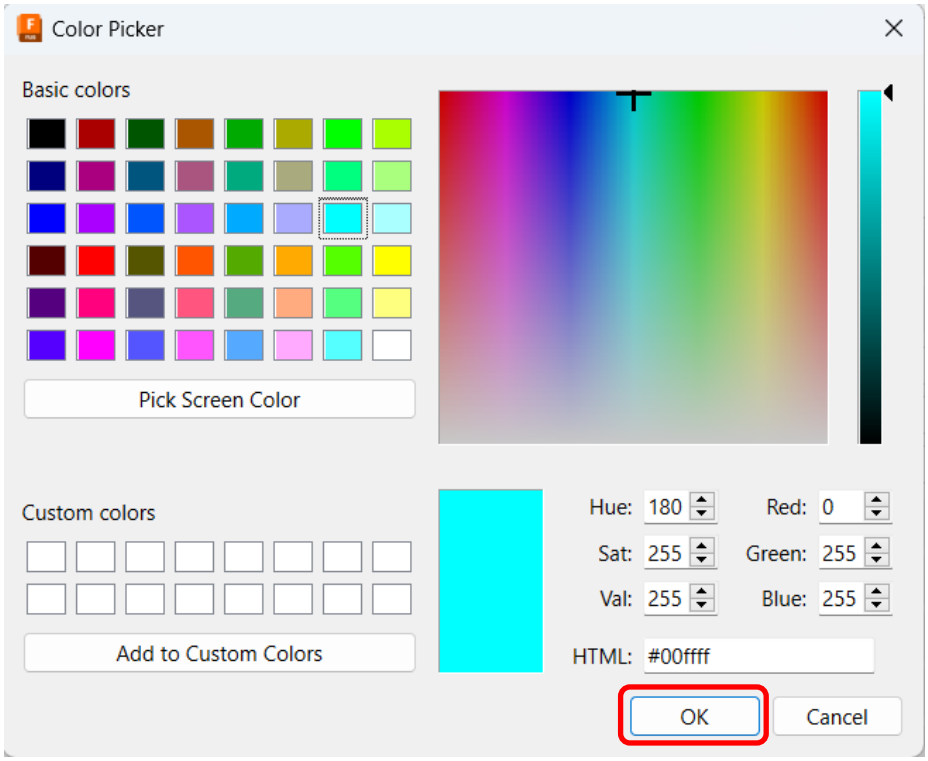
- click on **Advanced**



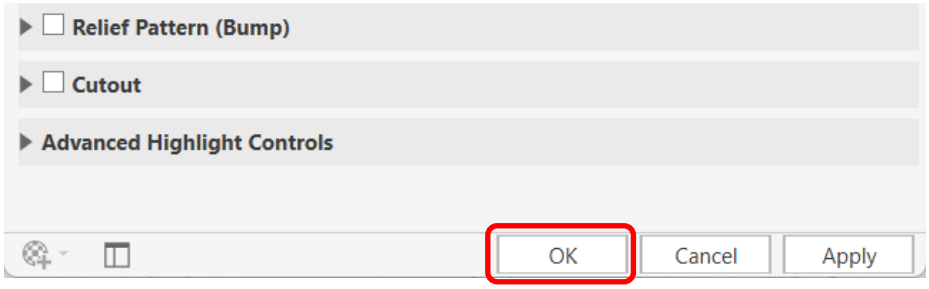
- select the **Appearance** tab
- click on the **Color list icon** and select **Edit Color…**



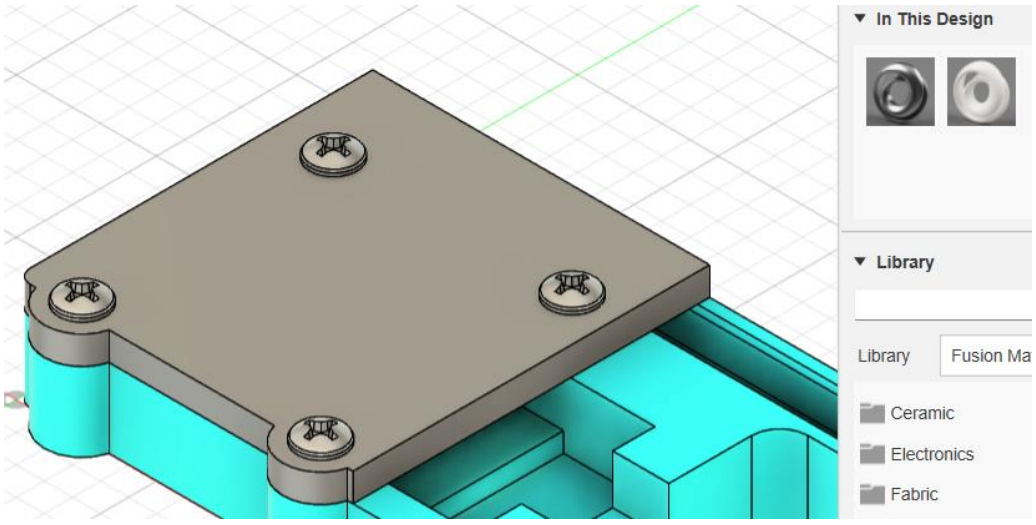
- click on a color and click **OK**



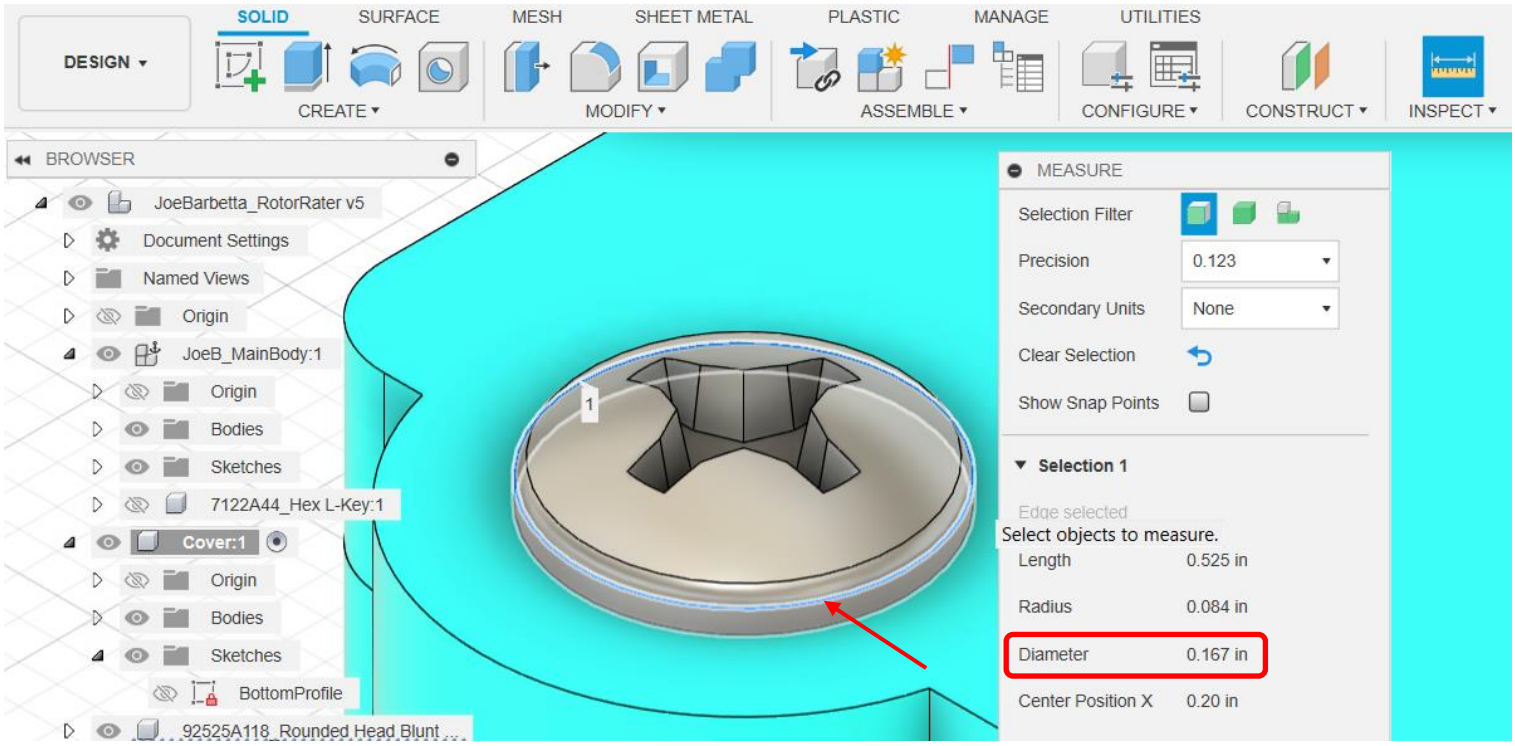
- click **OK** at the bottom of the Material Editor window



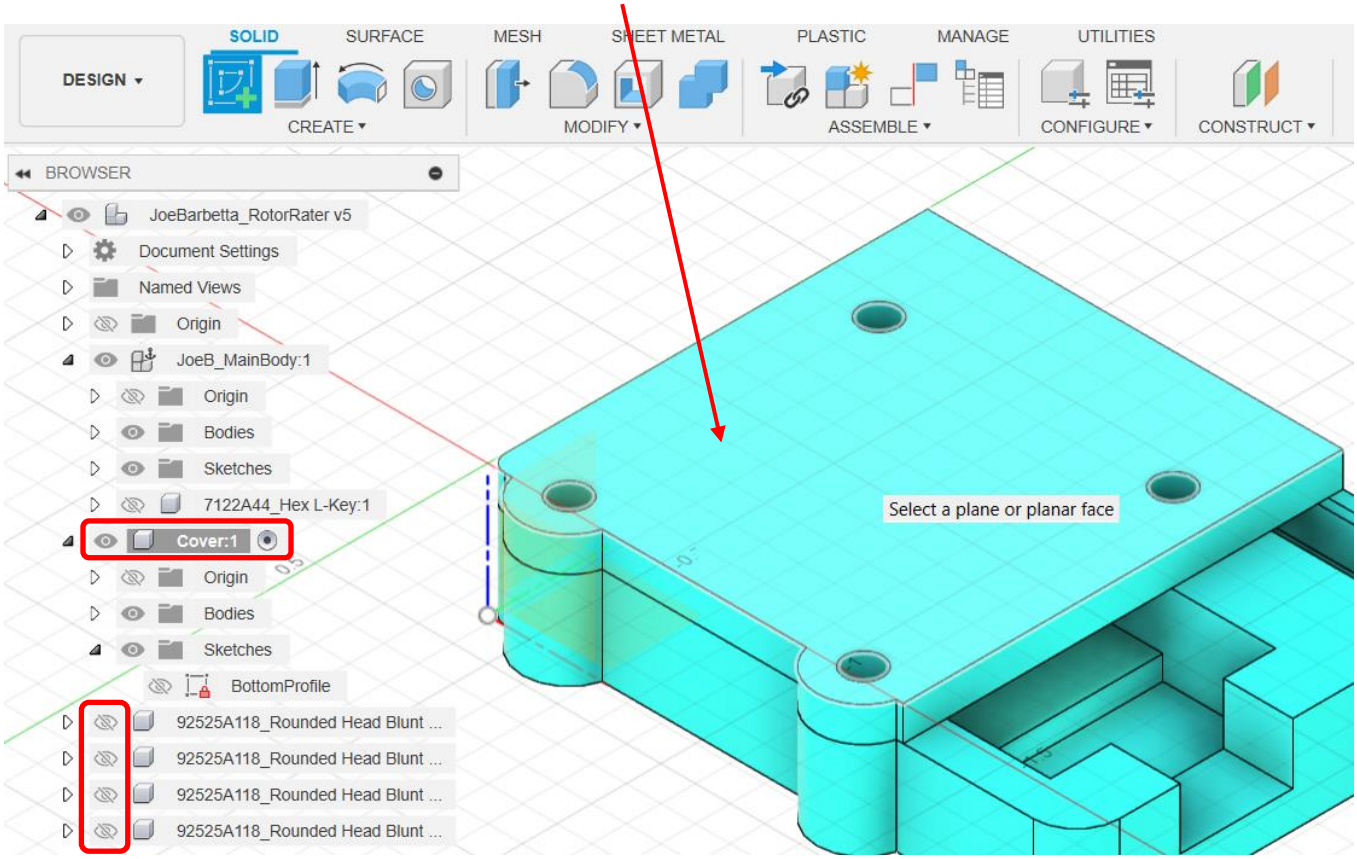
The color of the body should change. The color of the icon may not change.
The icon can also be dragged onto the cover as well.



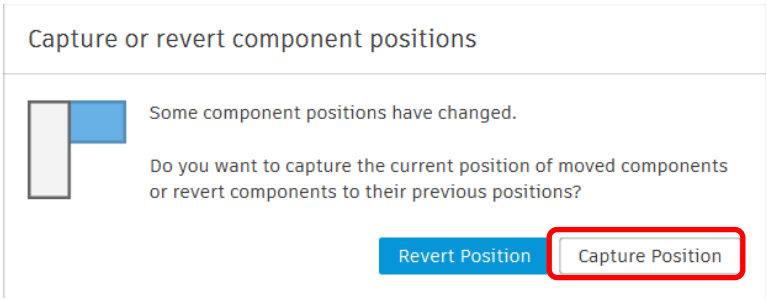
- zoom into one of the screws
- select the **Inspect Measure** tool and click on the circle for the largest diameter of the screw head, which should result in a **Diameter** of **0.167** in, then click **Close** on the Measure window.



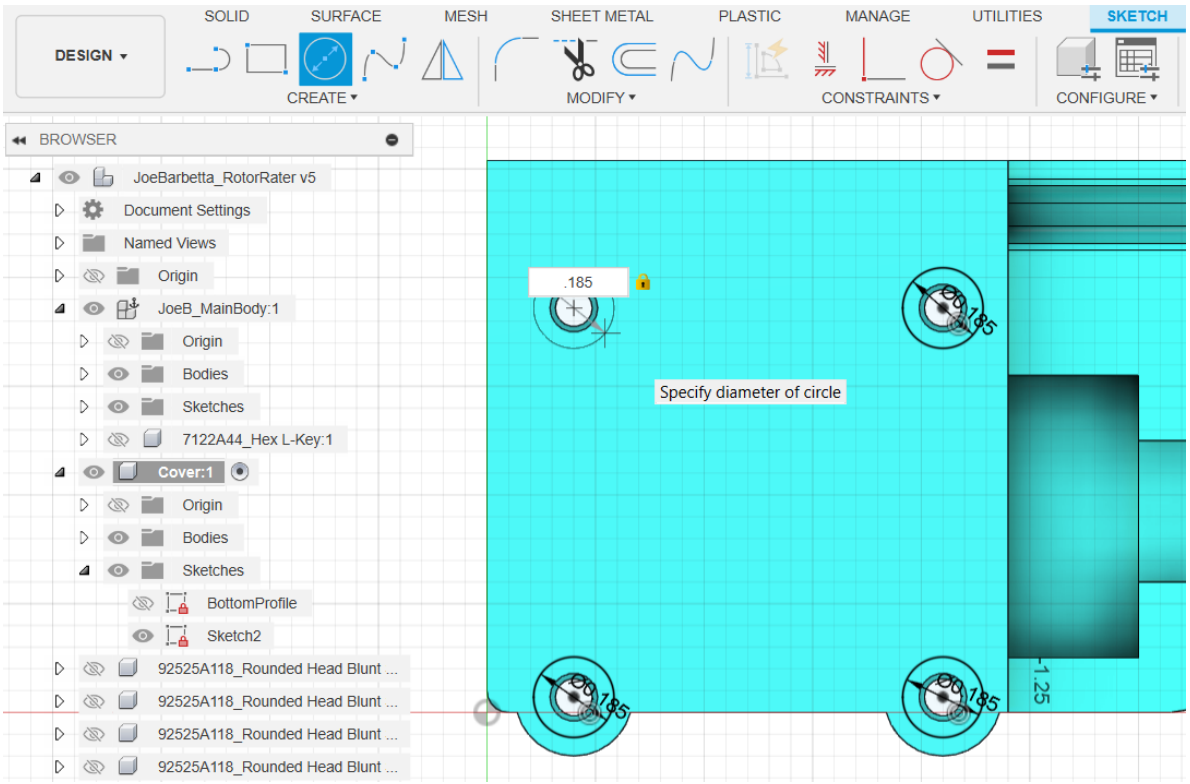
- click on the **eye** icons for each screw to hide them
- click on the **circle** next to **Cover** to activate it
- select the **Create Sketch** tool and click on the **top surface** of the cover



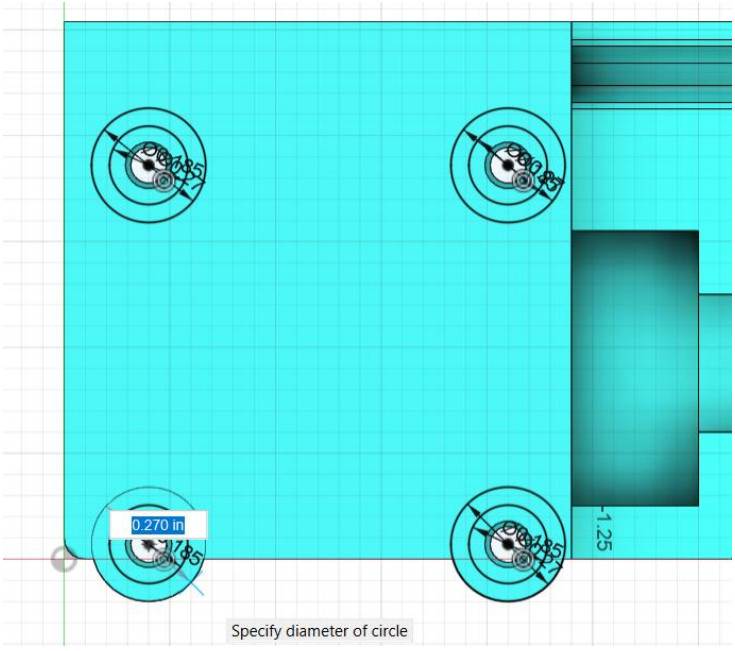
- if this message window ever shows, click on **Capture Position**



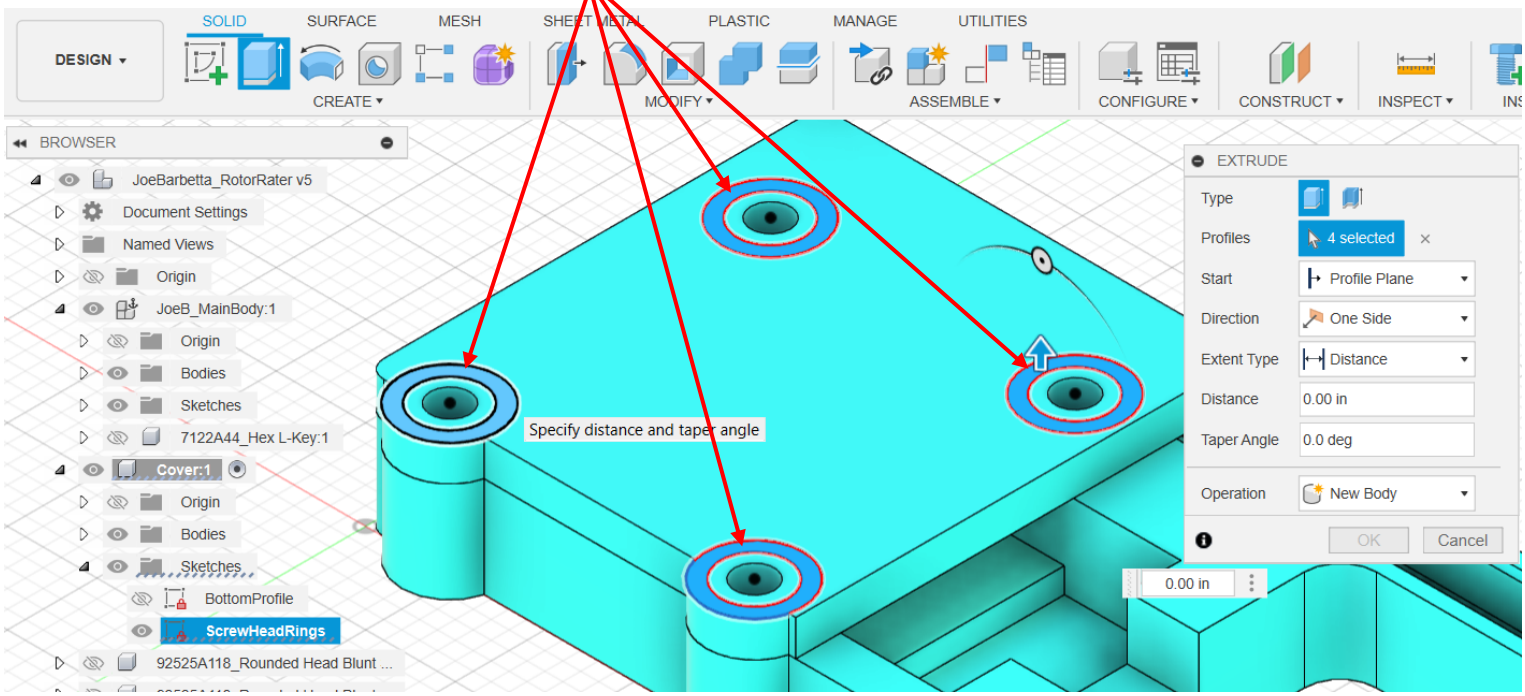
- use the **Circle** tool to create a circle of diameter **0.185** around each hole. This value was used because it is a little larger than the diameter of the screw head of **0.167**.



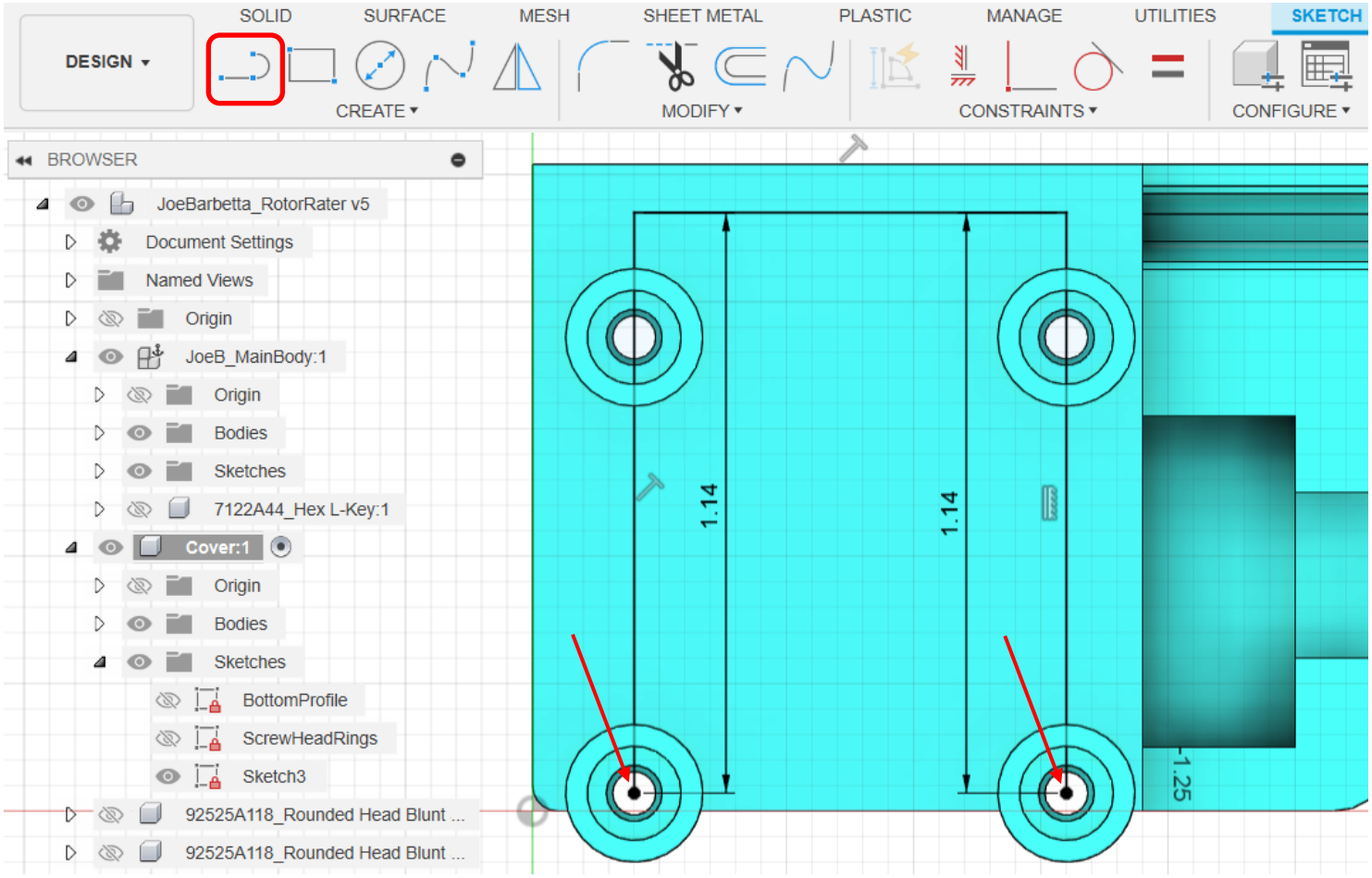
- create 4 more circles with a diameter of **0.270** around each hole. Click **Finish Sketch**.



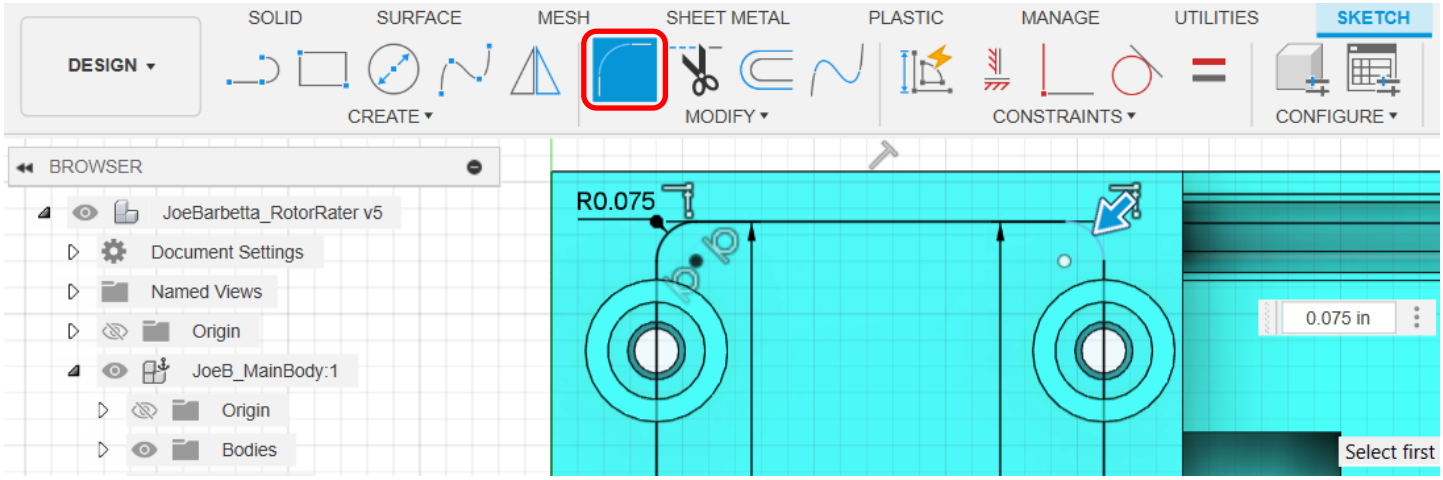
- return to the **Home view** and zoom into the top of the cover
- select the **Extrude** tool and click on the **outer ring region** around each hole and enter a value of **0.05**.



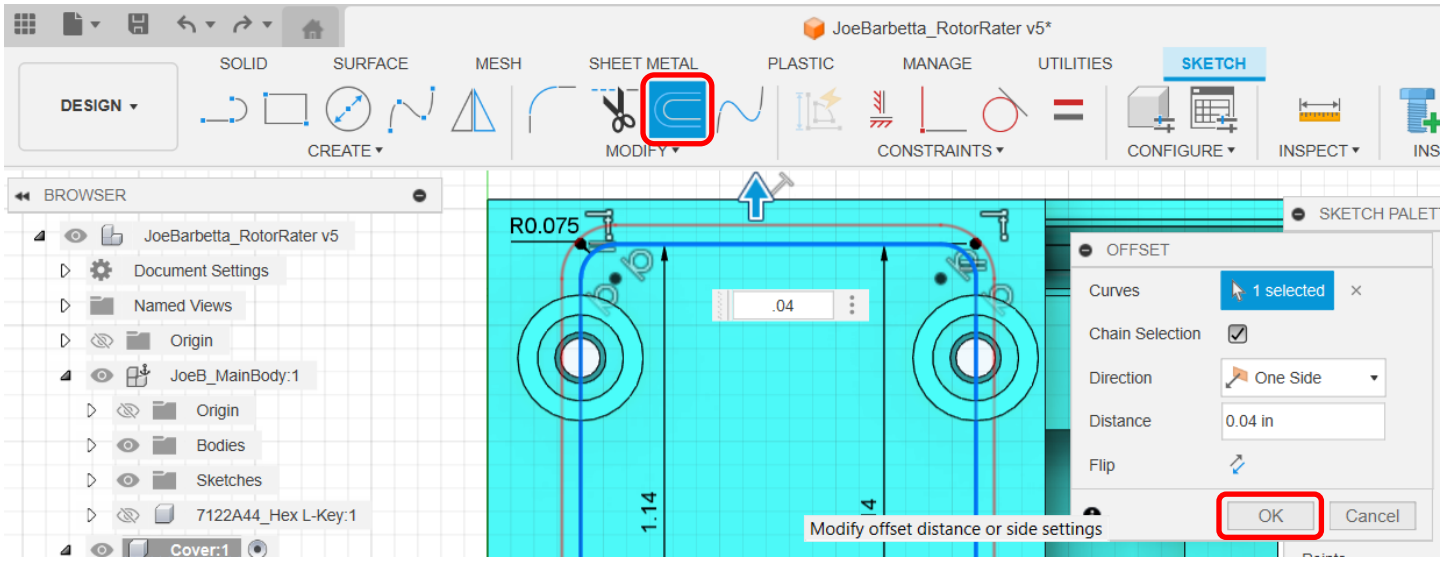
- start a new **Sketch** on the **top surface of the Cover**
- select the **Line** tool and create two lines starting at the bottom holes and upward by 1.14
- create a 3rd line connecting the tops of the two lines



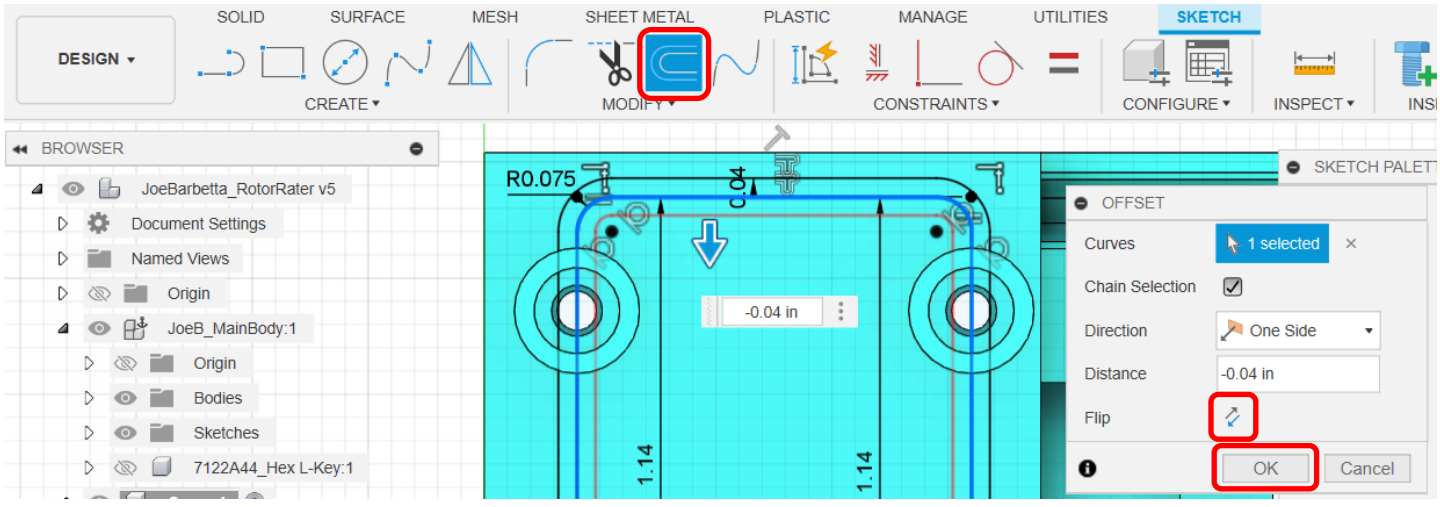
- create **fillets** of **0.075** at the top two corners of the lines just drawn



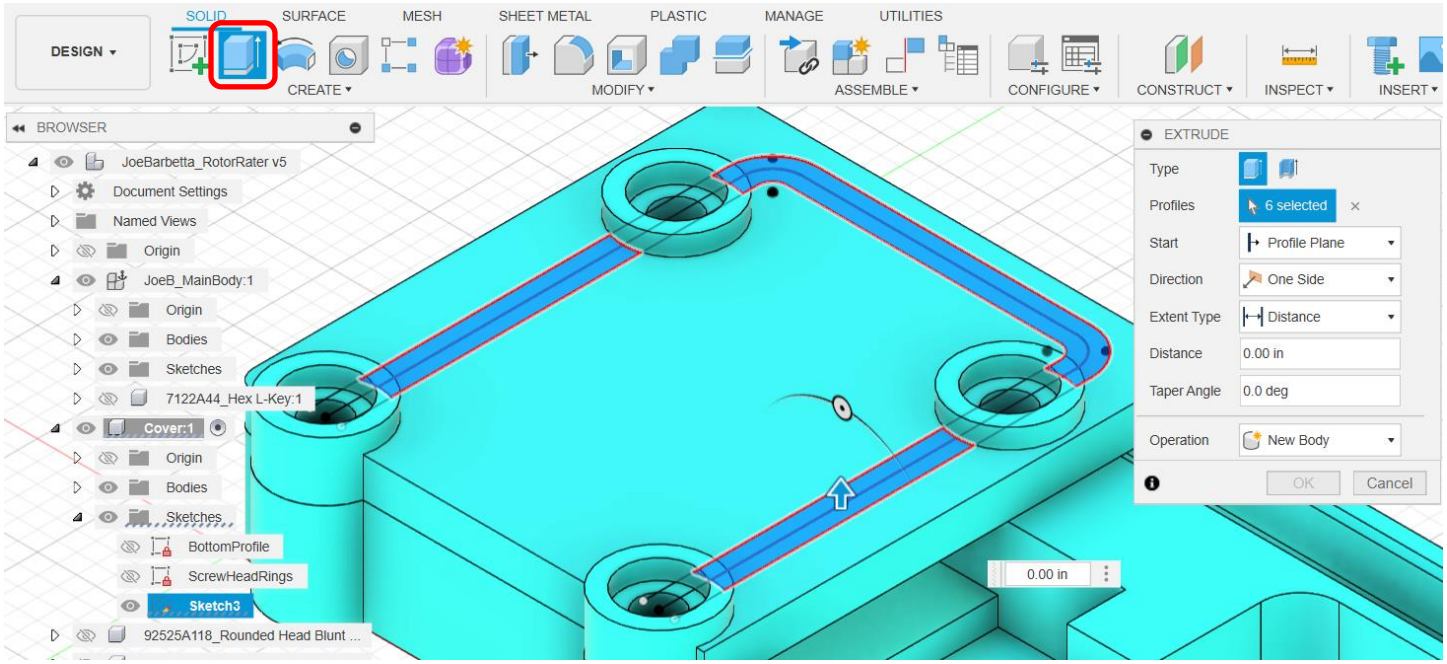
- select the **Offset** tool (if it is not visible find it in the MODIFY menu)
- click on the **top line**, type **0.04**, and click **OK**



- select the **Offset** tool (if it is not visible find it in the MODIFY menu)
- click on the **top line**, type **0.04**, and click **OK**

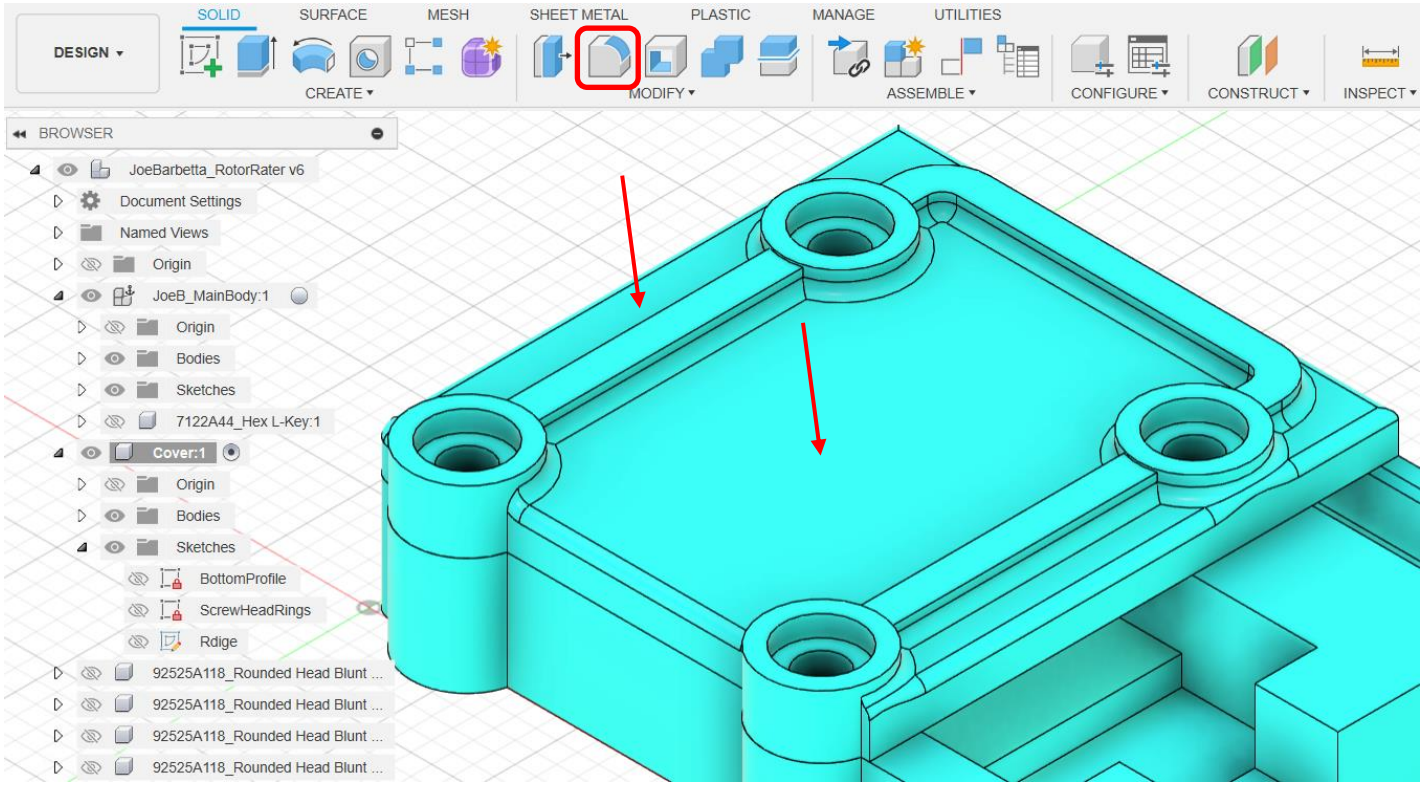


- click **Finish Sketch**
- return to the **Home** view and zoom into the **Cover**
- select the **Extrude** tool and click on the **6 regions** as shown
- enter a value of **0.045** and click **OK**



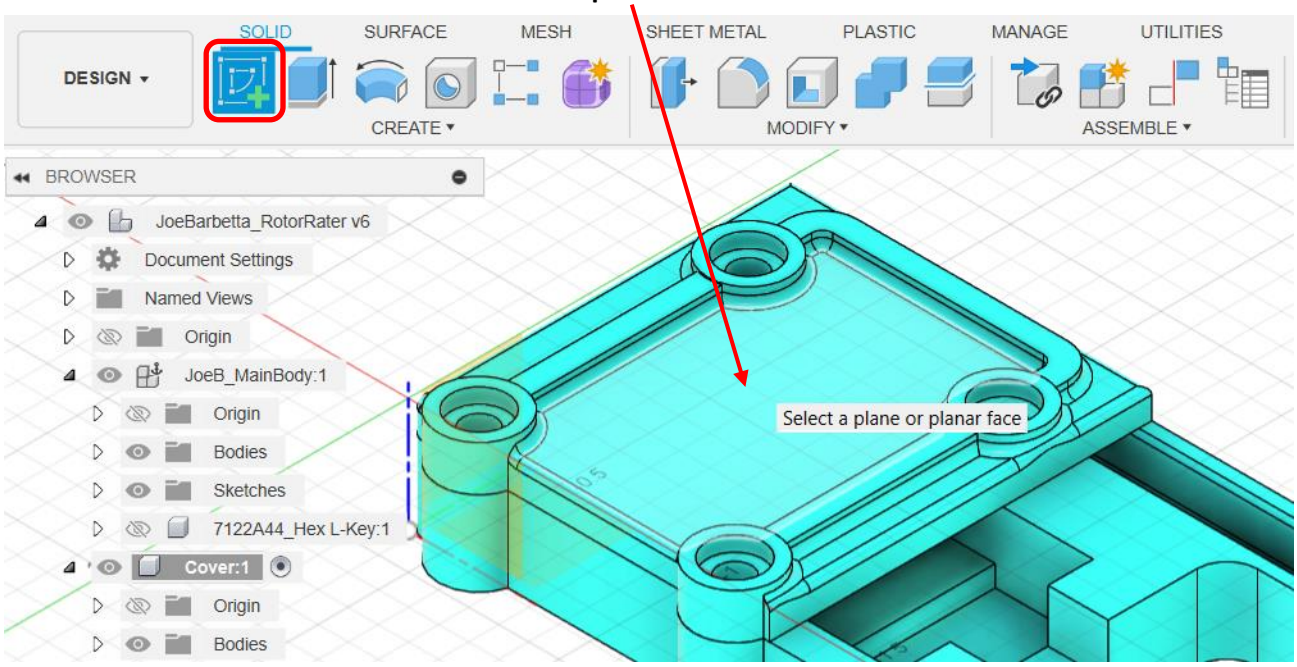
- select the **Fillet** tool and click on the two surface regions, one inside the ridge and one outside, and use a value of **0.025**. A few clicks just made the top look better.

The result should look like this. Won't this look better than just 4 screw heads on the surface. The ridge just added also adds some structural stretch by reducing upward deflection.

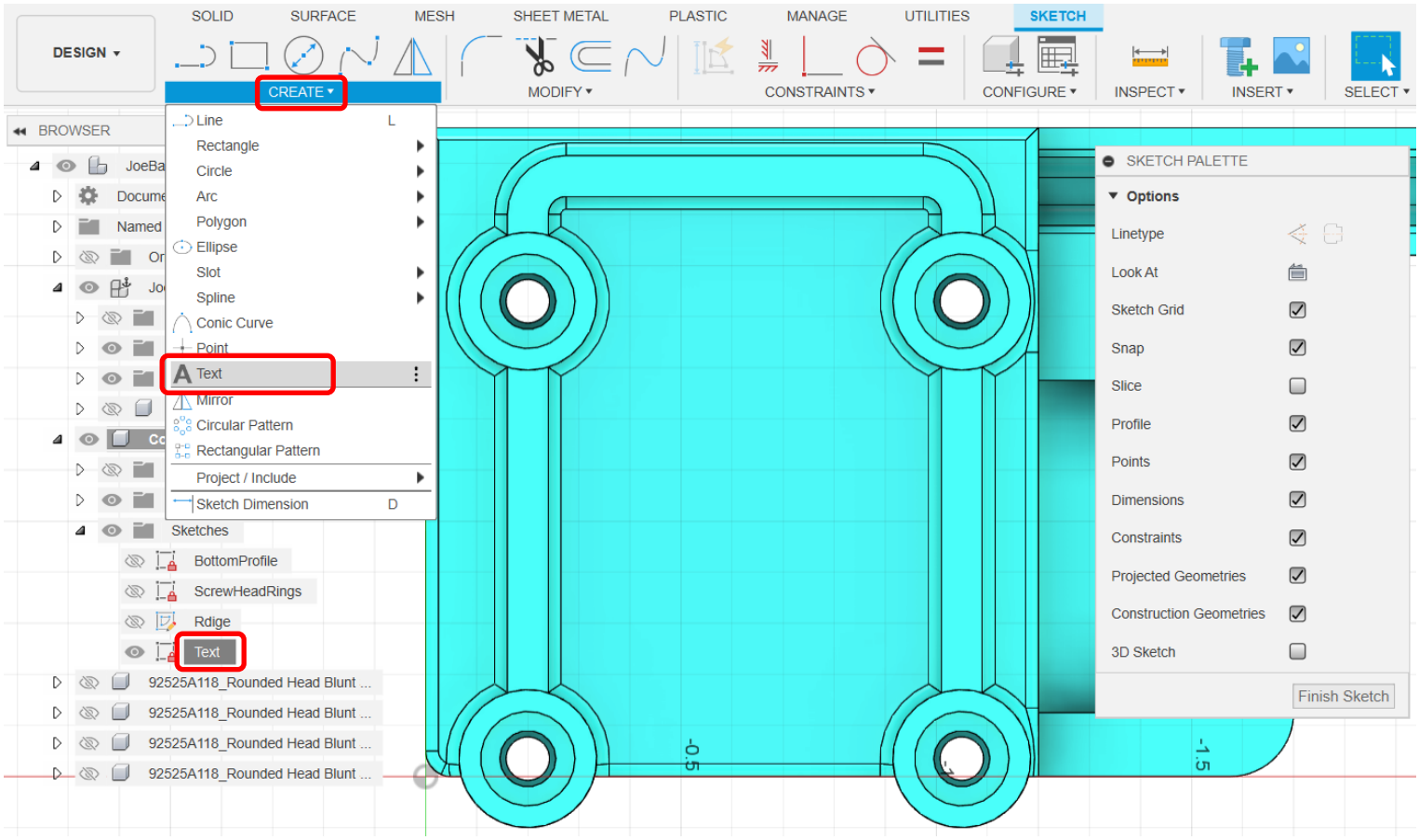


Adding Text

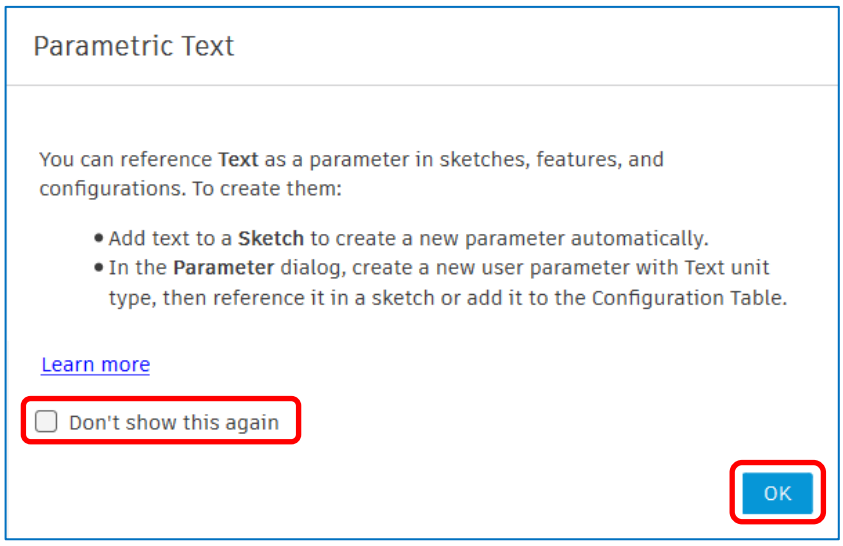
- select the **Create Sketch** tool and click on the **top surface** of the **Cover**



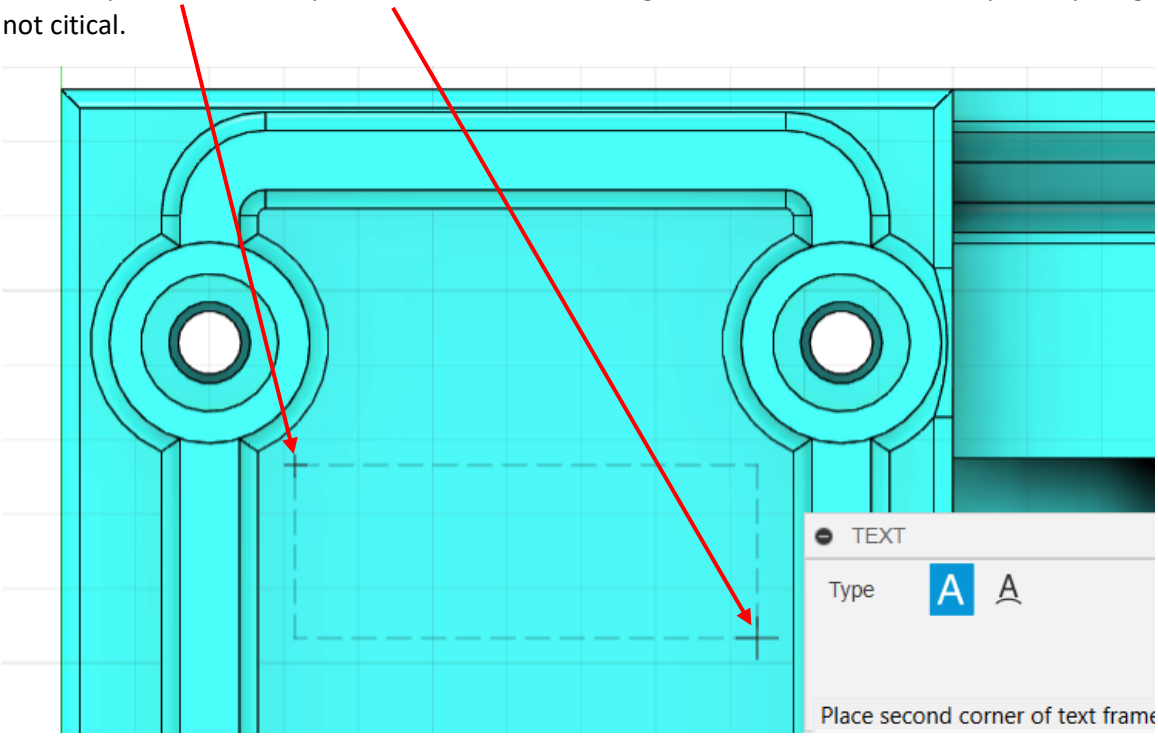
- zoom in to the top as shown
- from the **CREATE** pull-down menu select the **Text** tool. If a window pops up about **Parametric Text**, click its **OK** button.
- rename the Sketch to **Text**



- if this window like this pops up **check the Don't show this again** check box and then click **OK**. It essentially announces a new feature where a Parameter can be used to specify the text. It's a feature that we will Not be using.

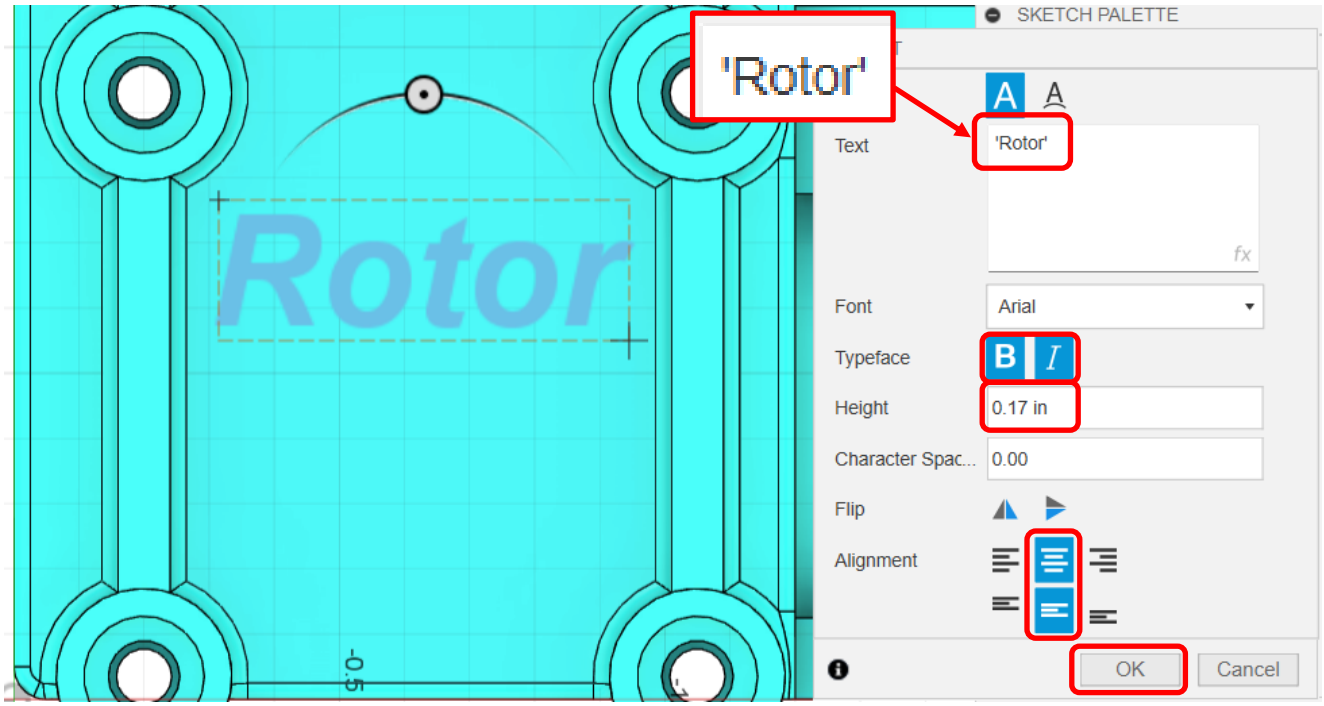


- click at point 1 and then point 2 to define the text region. The mouse will not snap to anything and the exact positions are not citical.

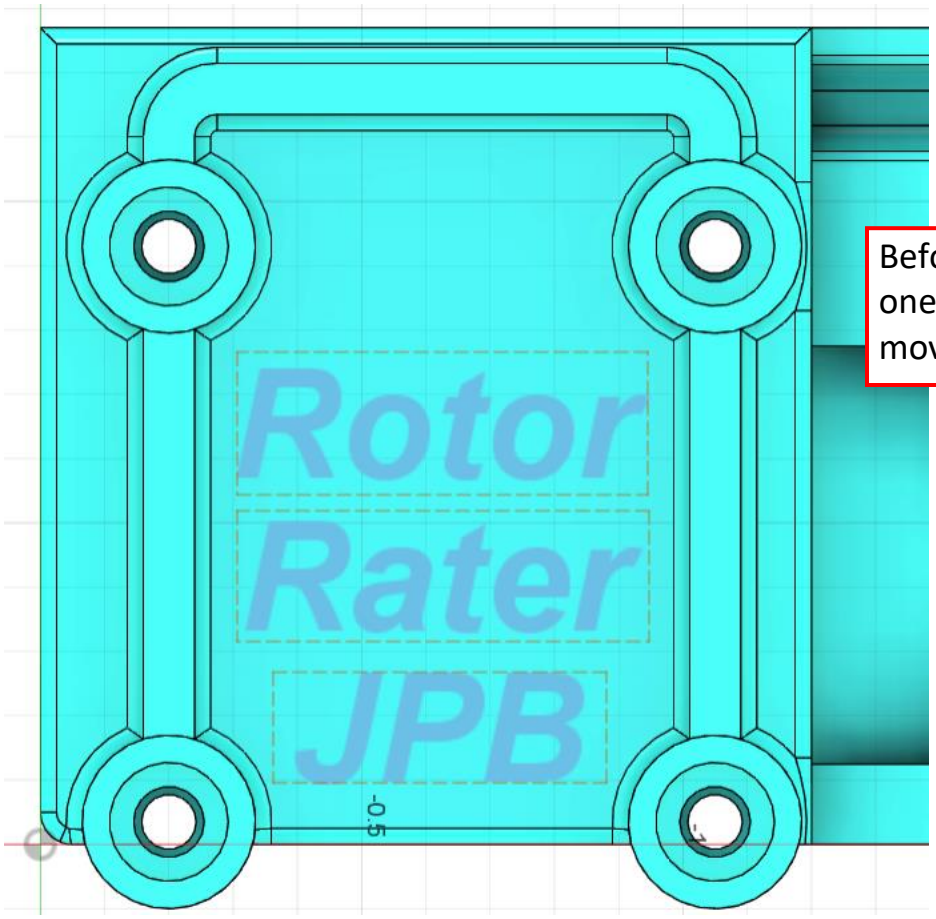


- enter the text **Rotor** in the **Text box**
- set the **Height to 0.17**
- click on the **Bold** and **Italic** options
- click on the **two middle Alignment options**
- click **OK**

As the magnified view of the Text entry shows on the right, the **text must be surrounded by single quotes, ‘, and Not double quotes, “.**

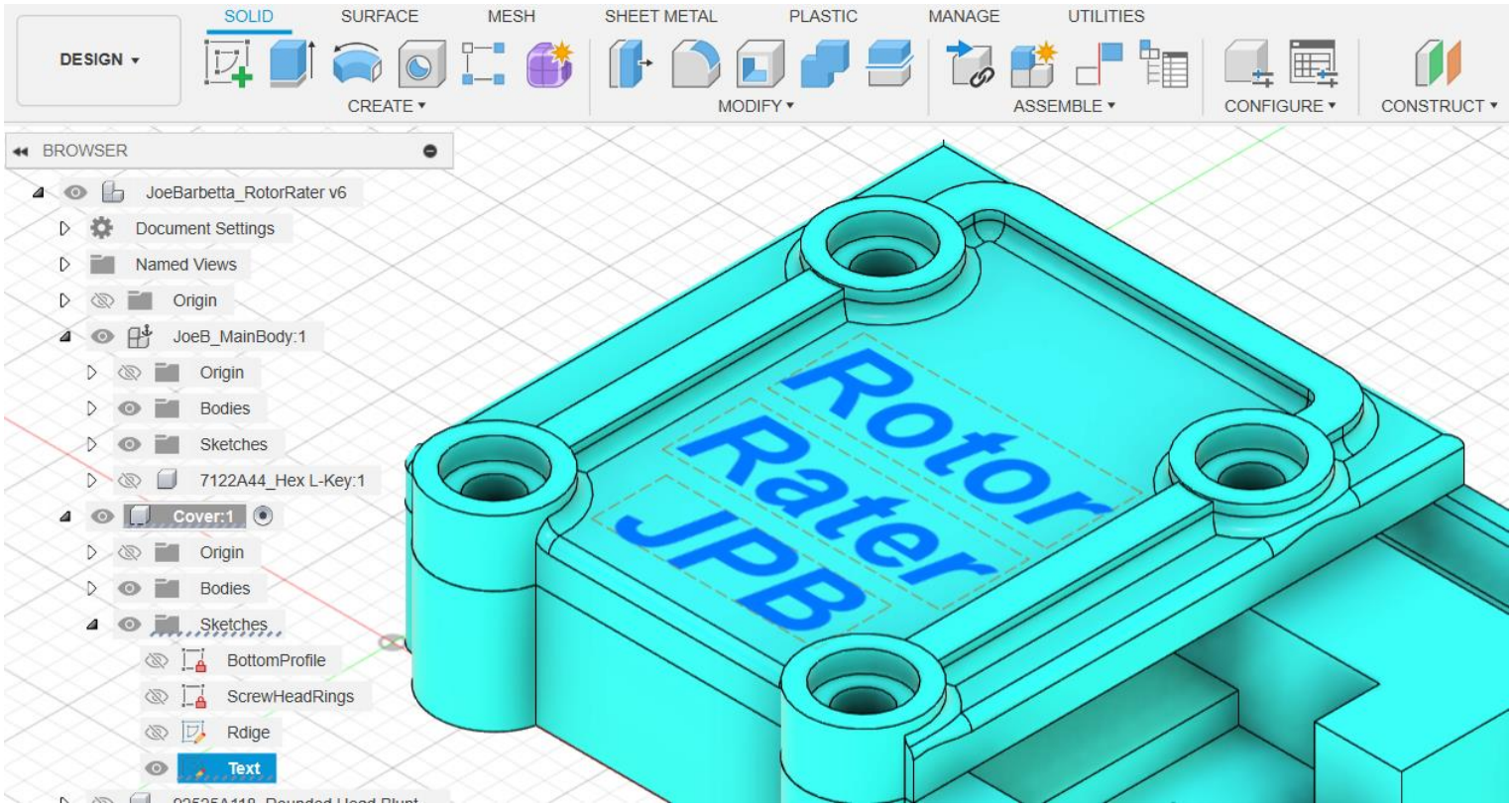


- perform the same operation to add the text **Rater** and your **initials** and click **Finish Sketch**

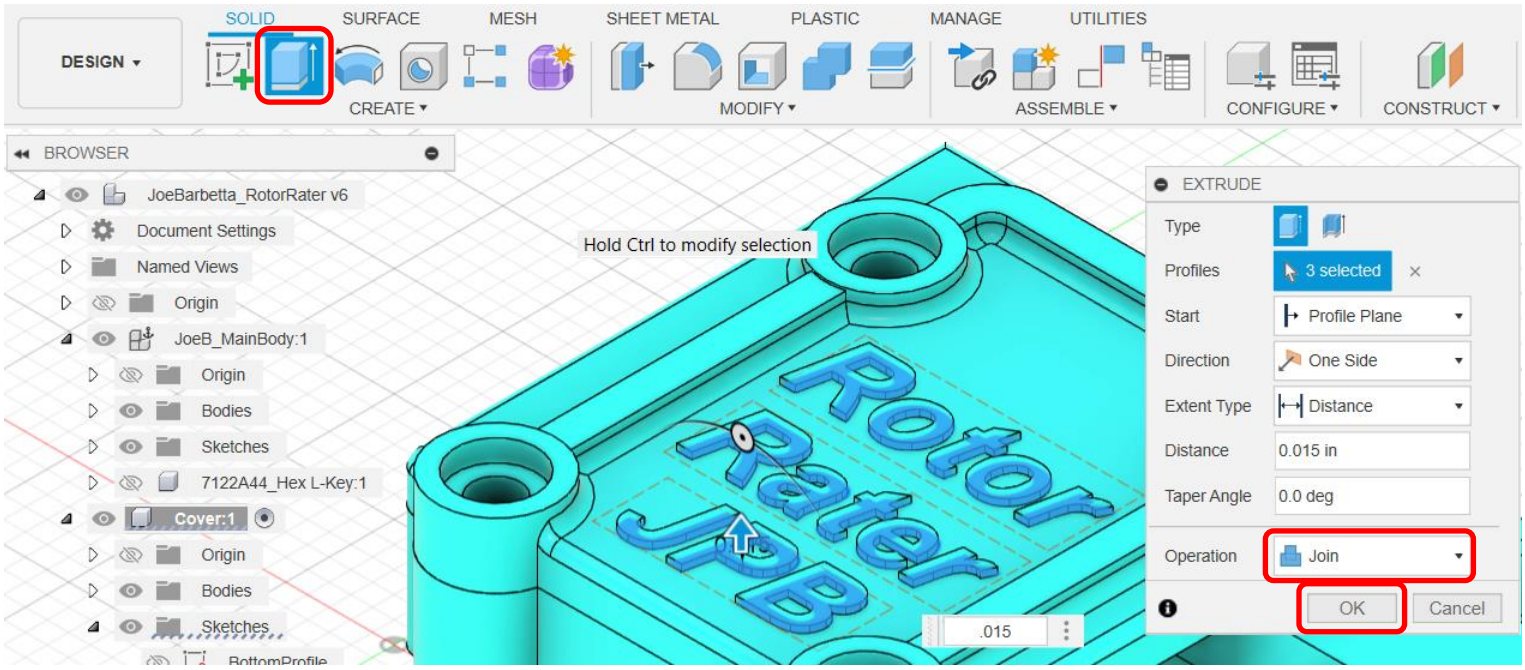


Before clicking Finish Sketch, one can click on a letter to move a word.

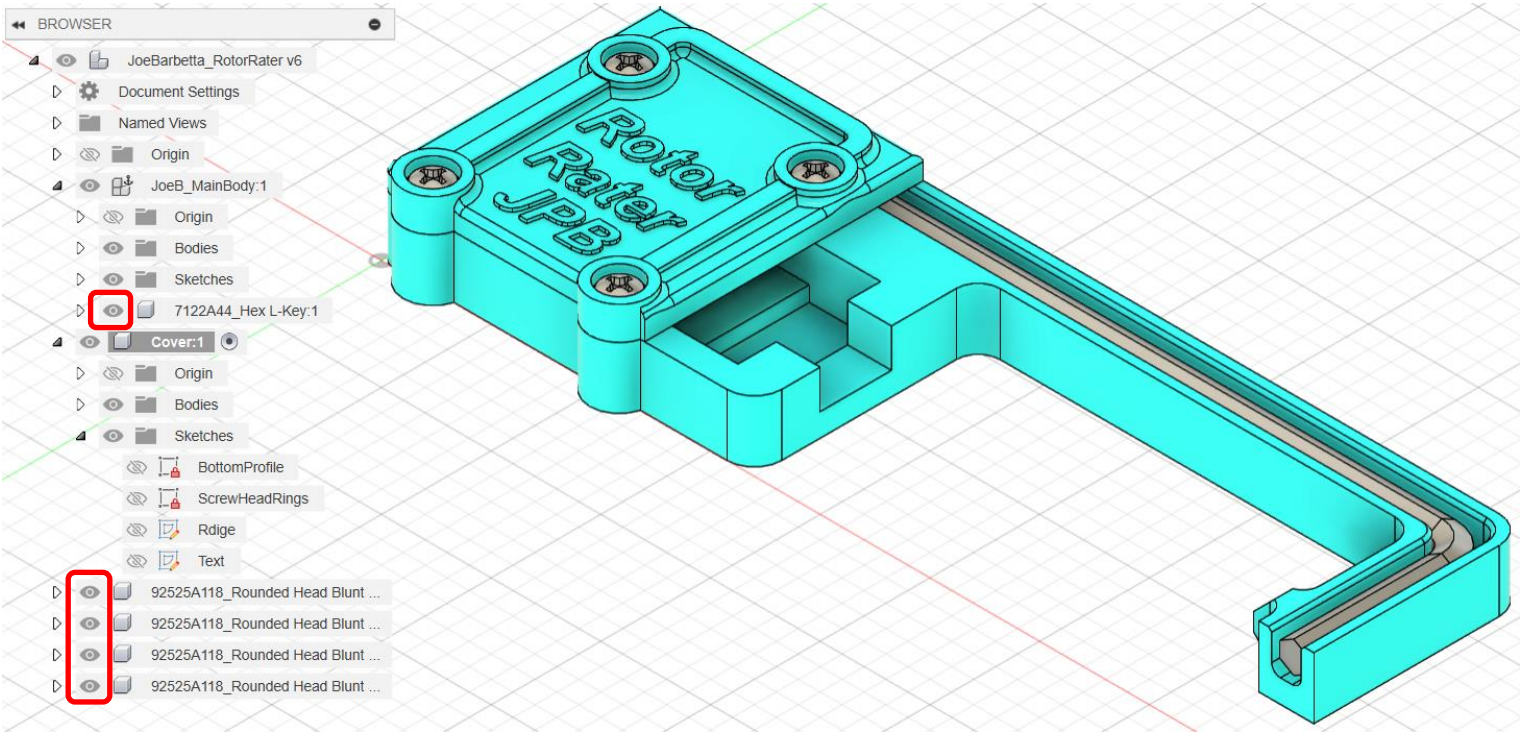
- return to the **Home** view and zoom into the top cover
- hold the **Shift key** and click on a **letter of each word** to turn them blue



- select the **Extrude** tool, type **0.015**, ensure the Operaton is set to **Join**, and click **OK**



- click on the **eye icon** for the **Hex Key** and **4 screws** to make them visible



Using the Rotor Rater on a front rotor.