TOP SECRET

Parametric Modeling of 3D Printed Rockets in Fusion

Create designs that can be easily scaled from shoulder fired to mobile launched rockets.

Written by Mr. Barbetta

TOP SECRET

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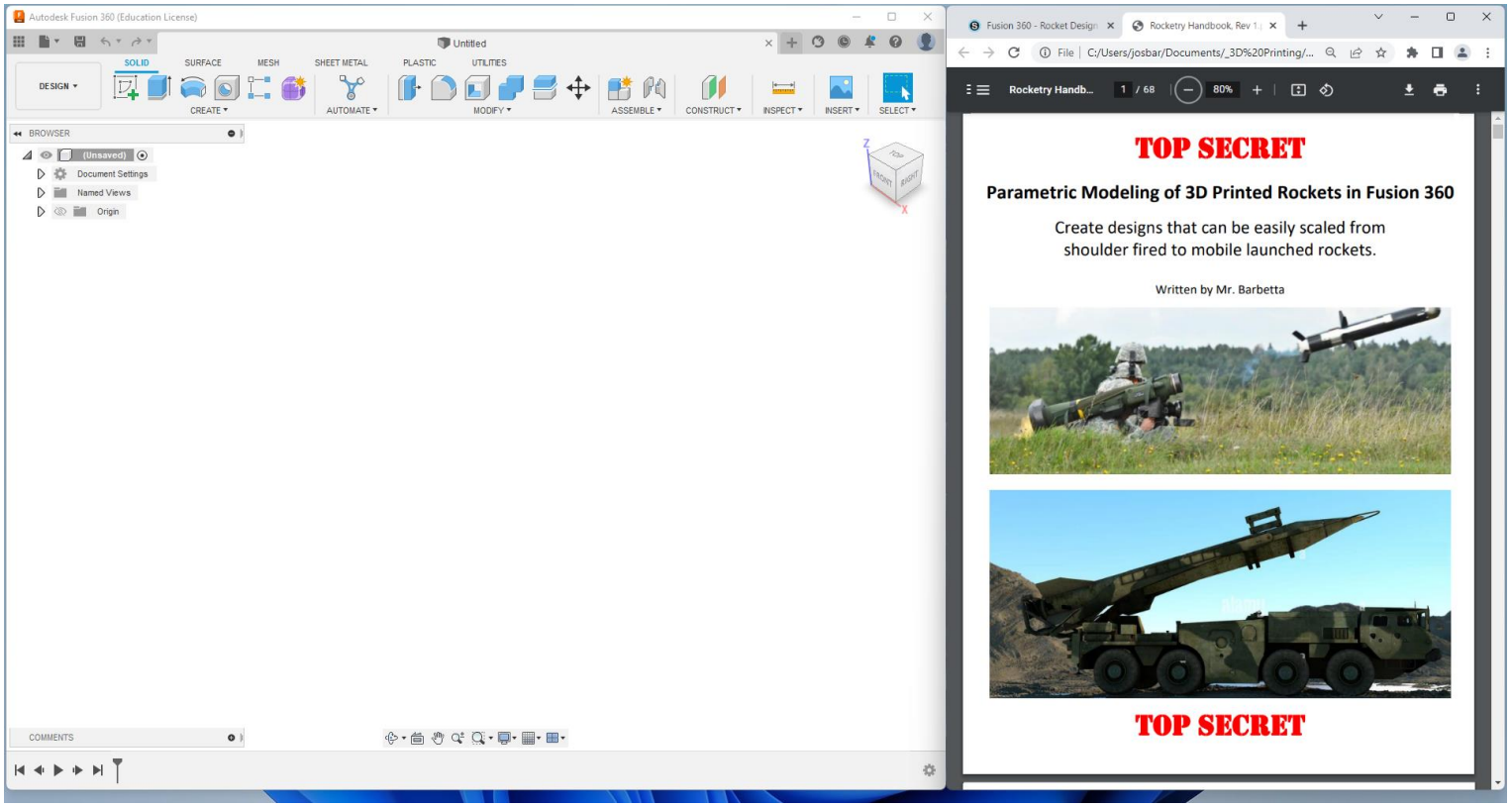
Using this Document

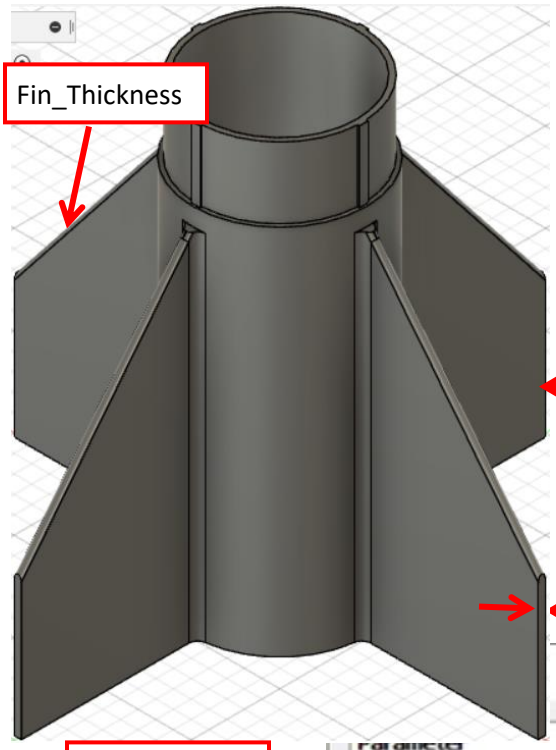
The best way to follow this document is to **reduce the width of the Fusion window** and have this pdf document open in Chrome browser as shown below. This document can be **downloaded from Schoology and then dragged into Chrome** and scaled down to 80%.



Introduction to Parametric Modelling

It is common with CAD (Computer Aided Design) work to set dimensions directly as one is creating the various features of a design. If a dimension needs to be later changed then one can find the feature and edit its dimension.

Parametric modelling is a feature of many CAD programs, wherein dimensions are stored in a "database" of parameters. One advantage allows a parameter to be correlated with multiple features. For example, if a design has many holes of the same size, it helps to use a parameter for the hole diameter. If the size of the holes needs to be changed, a single parameter can be updated, which will change all the holes at once. This is much more convenient compared to having to change the dimension for each hole individually. The advantages of using parameters also extend to feature dimensions that are based on others.



Below is an example parameter list for a fin set.

Changing Fin_Thickness from 0.040 in to 0.060 in would increase the thickness of each fin.

Note that a parameter can be a value or a function of other parameters.

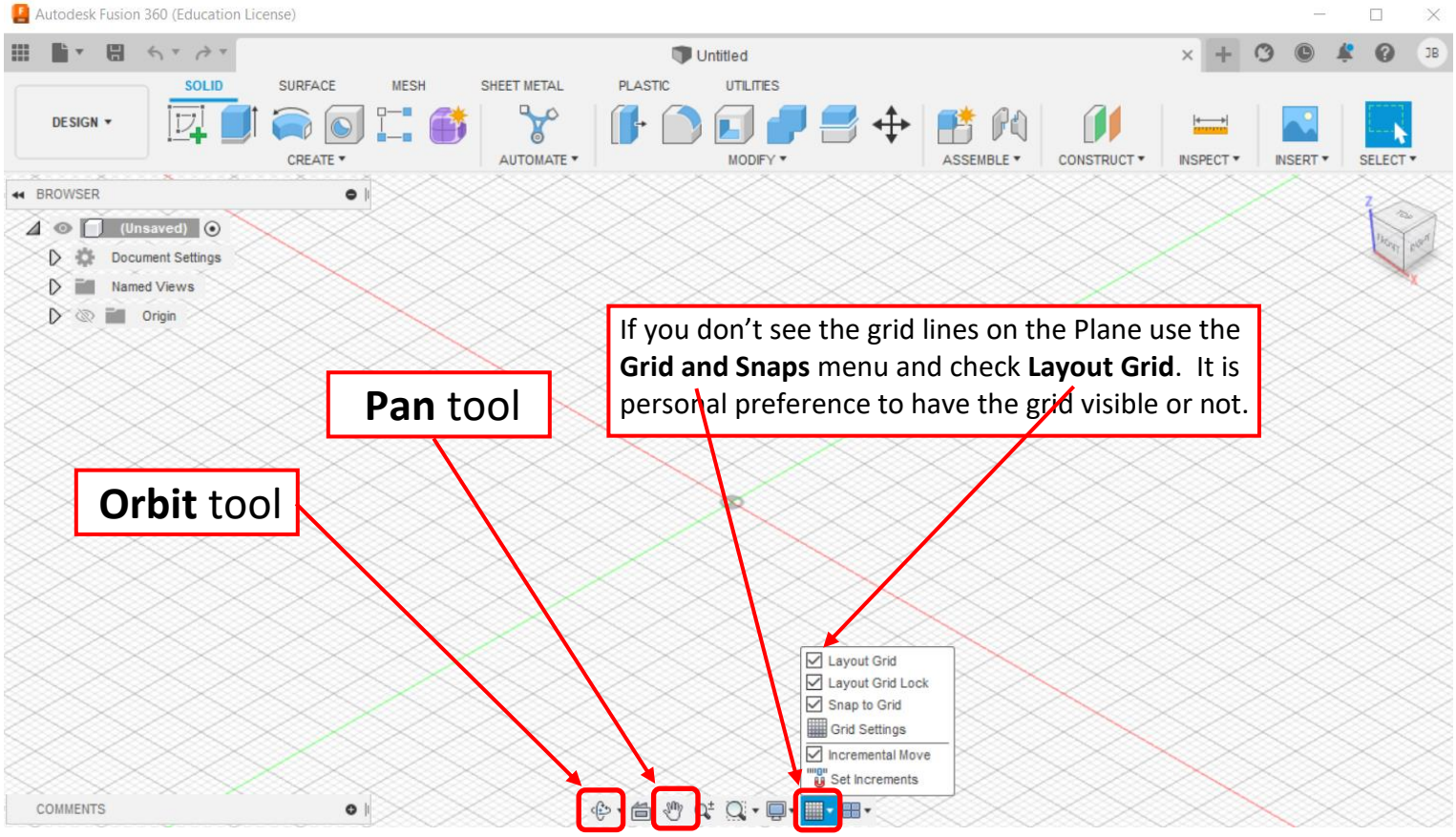
	Name	Unit	Expression	Value
Favorites				
User Parameters				
User Parameter	Fin_TubeOD	in	0.997 in	0.977
User Parameter	Fin_TubeID	in	0.870 in	0.87
User Parameter	Fin_TubeThickness	in	(Fin_TubeOD - Fin_TubeID) / 2	0.054
User Parameter	Fin_TubeLength	in	2.100 in	2.10
User Parameter	Fin_Thickness	in	0.040 in	0.04
User Parameter	Fin_Height	in	1.000 in	1.00
User Parameter	Fin_Count		4	4
User Parameter	Fin_RootChord	in	2.000 in	2.00
User Parameter	Fin_SweepLength	in	1.200 in	1.20
User Parameter	Fin_FilletRadius	in	0.050 in	0.05
User Parameter	Fin_CouplingOD	in	0.930 in	0.93
User Parameter	Fin_CouplingLength	in	0.400 in	0.40
User Parameter	Fin_CouplingRibOD	in	0.950 in	0.95
Model Parameters				
FinAssembly (ver 2.0.1...				

OK

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.

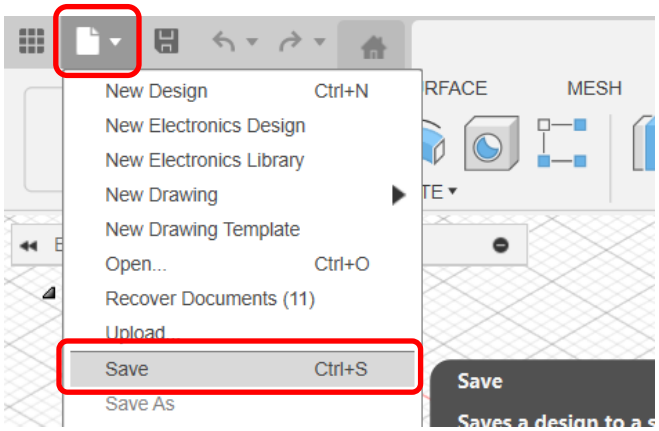
- click on the **View Cube** and move the cube while holding the mouse button down. This is another way to rotate the view.
- click on the Top of the View Cube and note how the view just jumped to a Top View.

The View Cube now resembles that on the right.

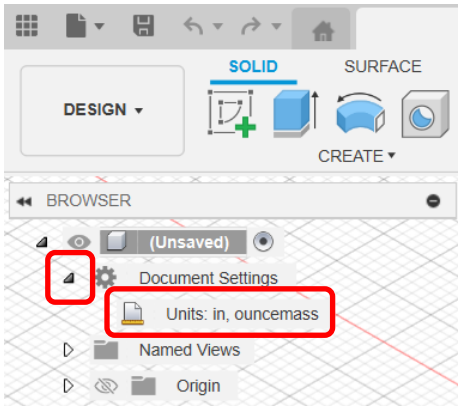
- click on the **Curved Arrows** at the upper right of the View Cube and practice Rotating the View.
- click on the **Arrows** at the sides of the View Cube to practice jumping to various Views.
- click on the **Home** icon to the upper left of the View Cube. This can always be used to reset the view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

Starting a Design in Fusion

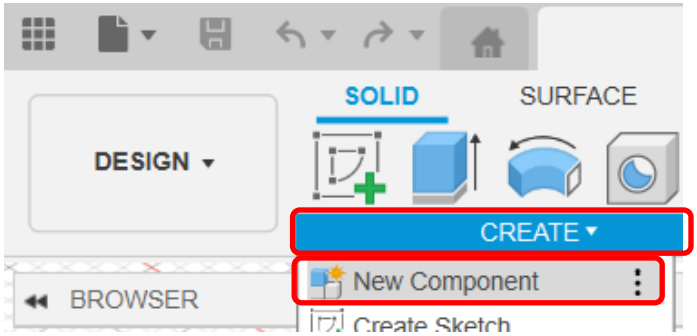
- open Fusion 360. If there is no icon on the Desktop, use the Windows search (magnifying glass icon) and type fusion
- from top **File** icon select **Save** and name the file with you name followed by **-BT50 Rocket**, e.g. JoeBarbetta-BT50 Rocket

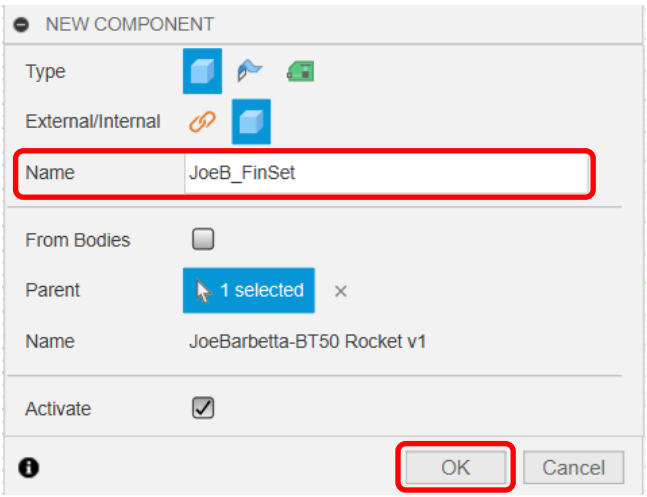


- in the left **"BROWSER"** click on the **arrow next to Document Settings**
- click on the **edit icon** that appears to the left when you hover over Units
- change **Unit Type** to **Inch** (If you set this in Preferences, it should already be **Units: in**, but it's good to verify.



- from top **CREATE** menu select **New Component**
- in Name enter **your first name and last name initial** followed by an **underscore** and **FinSet** e.g. **JoeB_FinSet**
- click **OK**

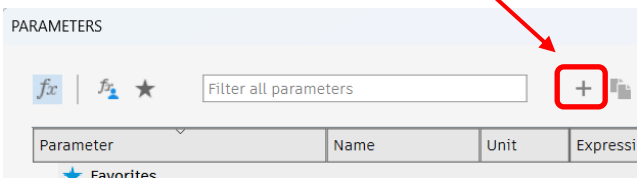


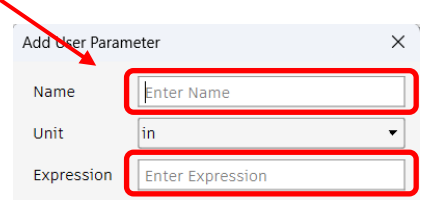


Entering User Parameters

Next, we will set some parameters that can be used to set dimensions to easily change dimensions after the design is completed. We can thus use this model for shoulder-fired rockets up to long range missiles.

- from the top **MODIFY** menu select **Change Parameters** near the bottom of the list
- next to **User Parameters** click on **+** and enter the **Name and Expression** for each item in the below list. Note that you will need to click **+** for each item. **Don't forget to click OK when done!**





Name	**Expression**
Fin_TubeOD	0.997
Fin_TubeThicknes	0.050
Fin_TubeLength	2.100

OD is a common abbreviation for ***Outer Diameter*** or ***Outer Dimension***.

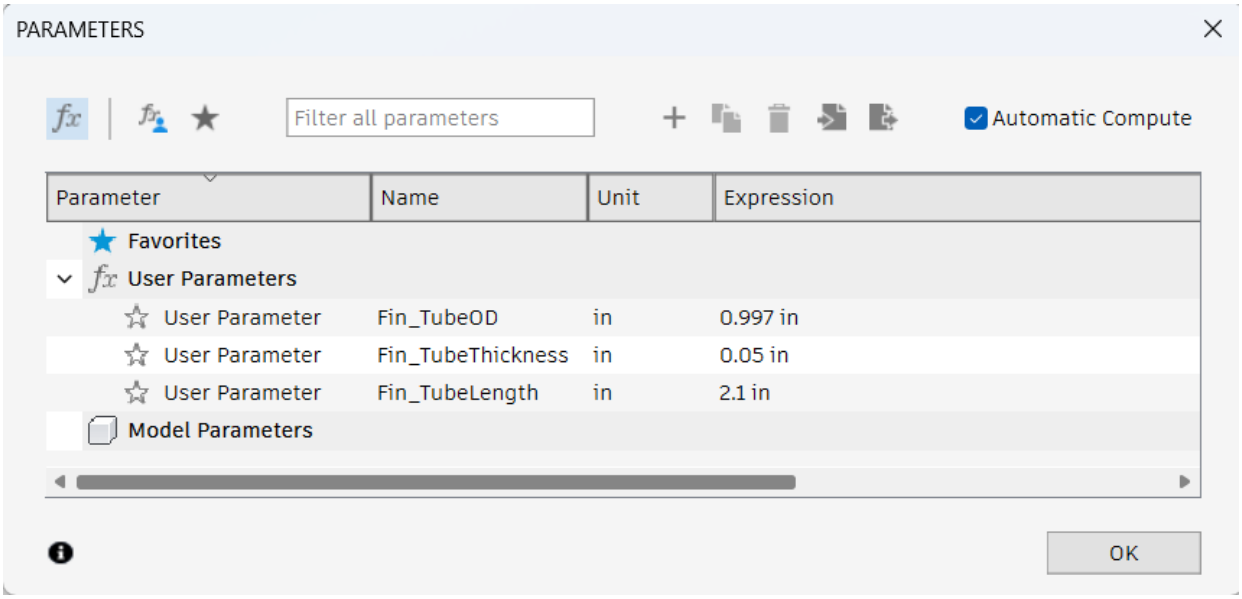
0.997 is slightly larger than the OD for a common model rocket body tube BT-50, which has a 0.976" OD.

Note that values are specified with 3 digits after the decimal point, i.e. 0.050, 2.100. This use of significant digits conveys the ***tolerance*** of the dimensions. A common manufacturing tolerance could be +/- 0.005".

Note the underscores in the names.

After entries you can always reopen the **Parameters** window to **add more parameters or edit any** by clicking on a name or expression. For parameters that may be frequently used, its star can be clicked to have it also appear in Favorites

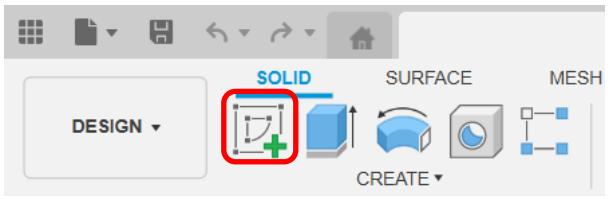
The Parameters window should look as below.



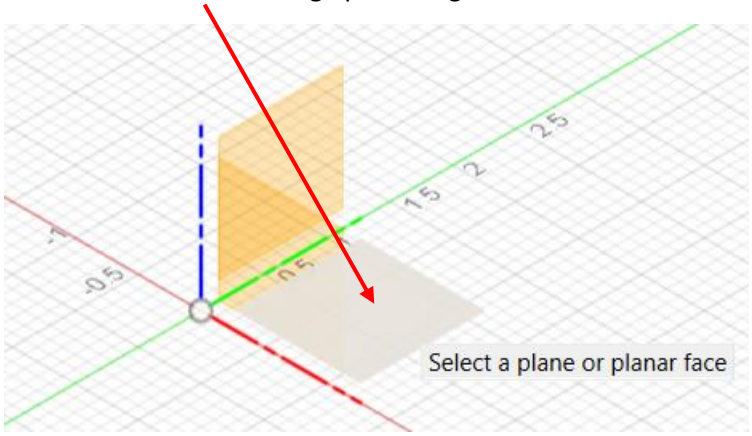
Many complicated steps will follow. Being a rocket scientist is not easy. If you find this too complicated, you may want to consider an easier career, such as that of a brain surgeon.

Creating the Fin Tube (Sketch and Extrude)

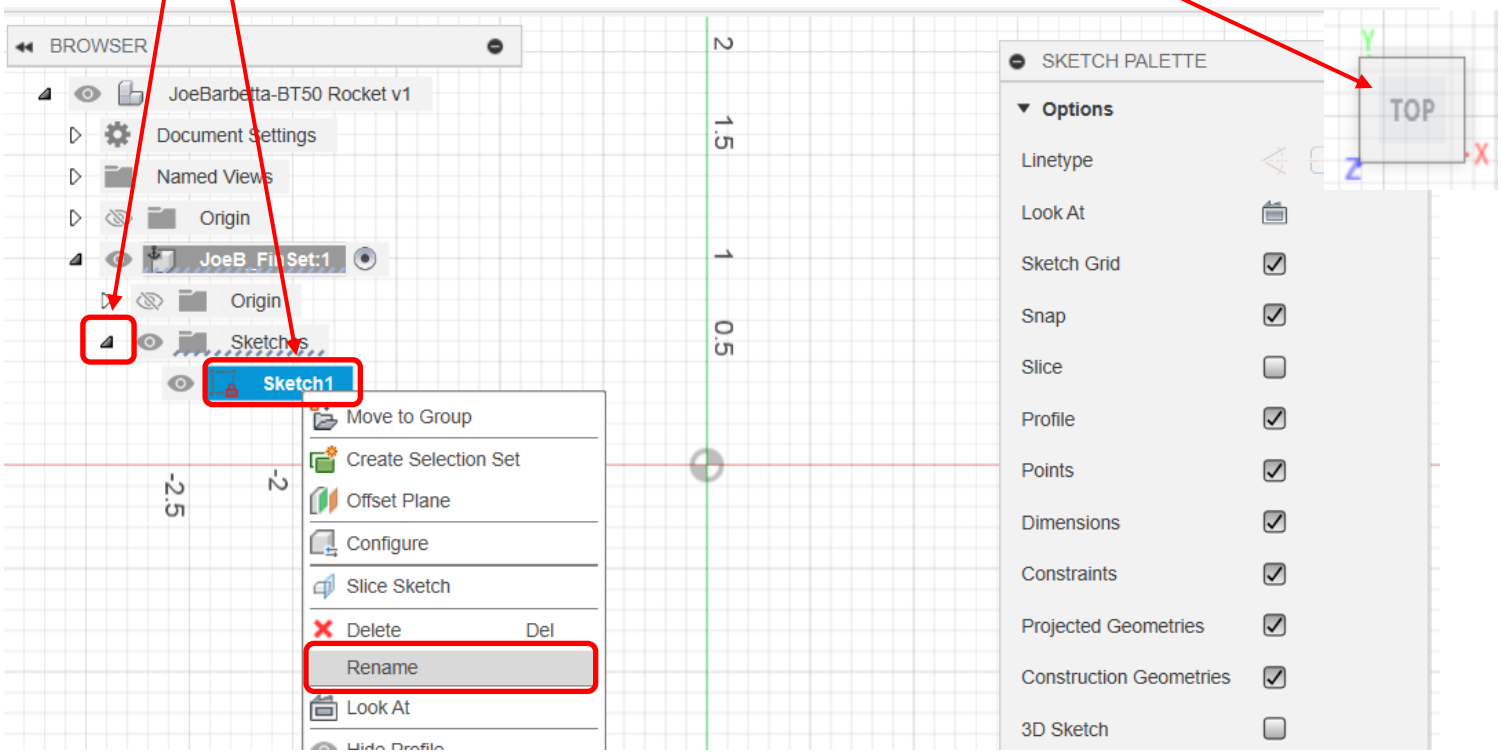
- click on **Create Sketch** (if you don't see the icon select Create Sketch from the CREATE menu.)



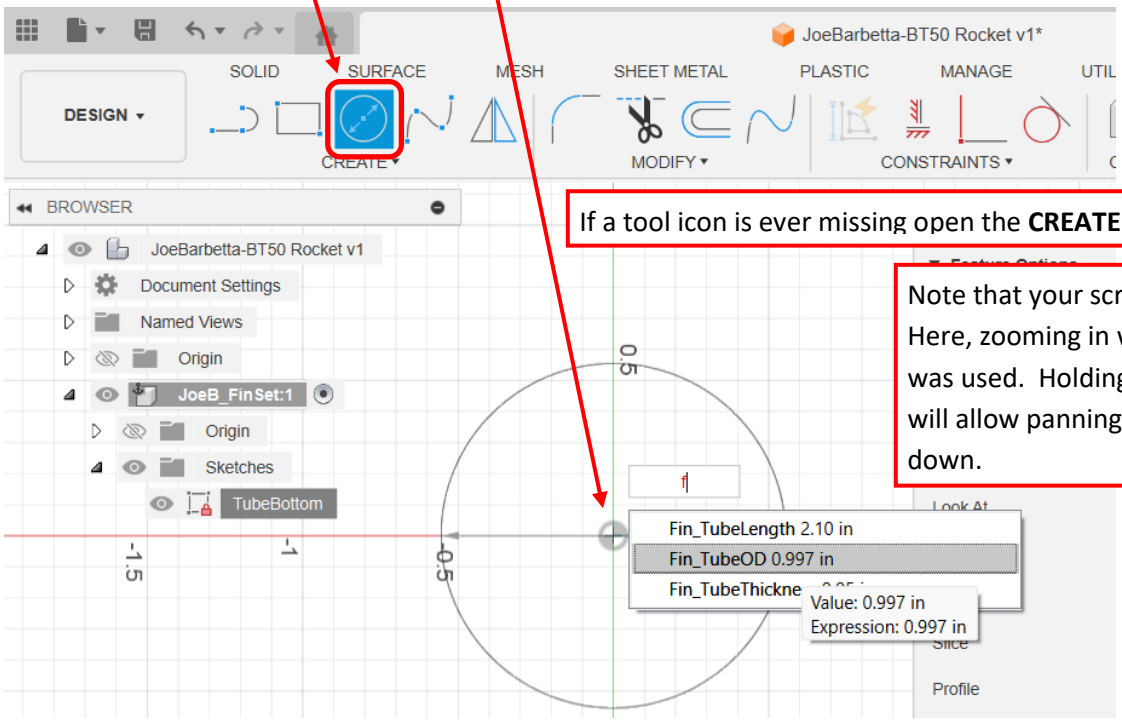
- hover over **bottom** orange parallelogram in center to cause it to turn gray and then click on it



- the view should be similar to that below and the View Cube should show **TOP**.
- click on the **arrow next to Sketches**
- right-click on **Sketch1** and select **Rename** and enter **TubeBottom**



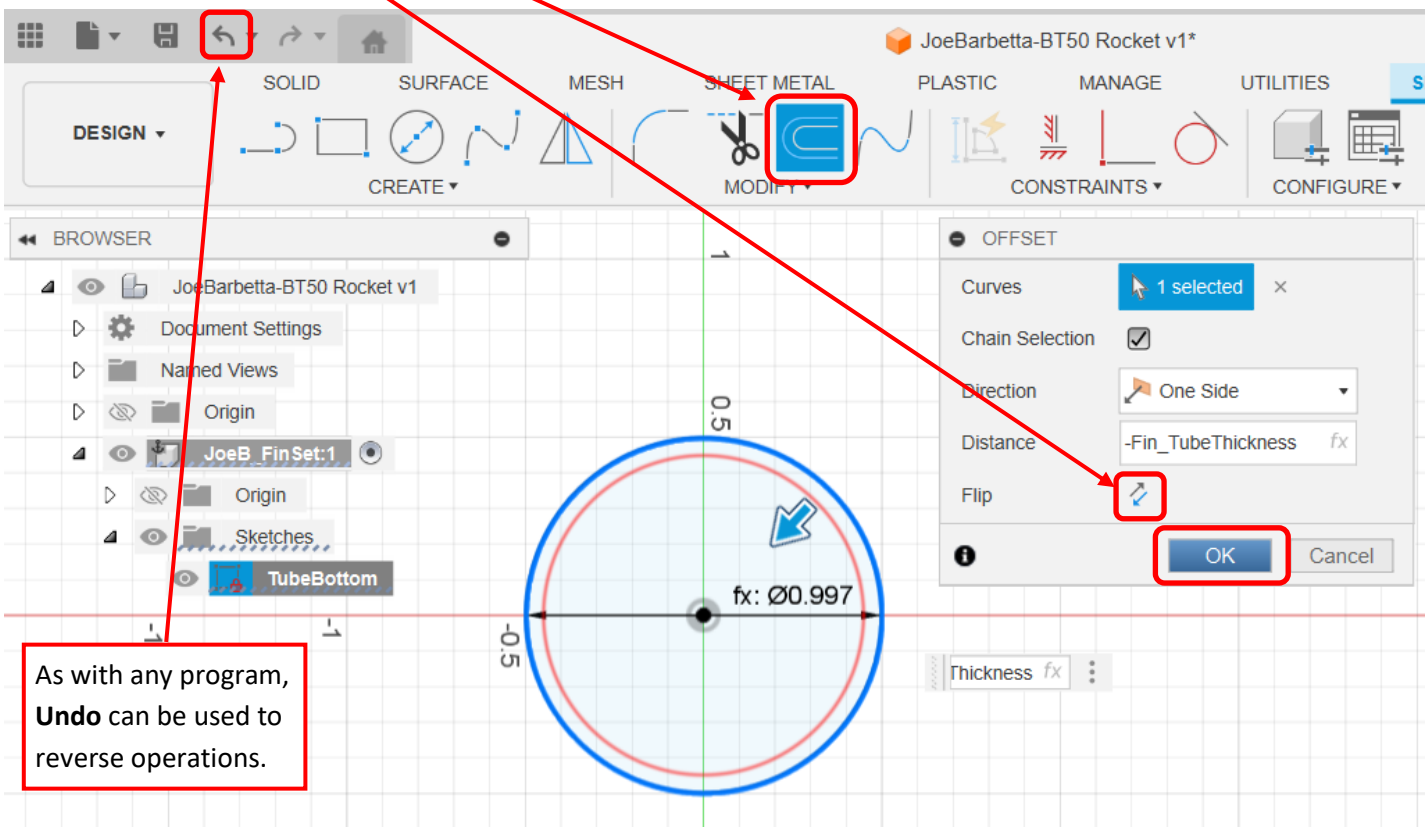
- select the **Circle tool** (if the icon is not there, from the CREATE menu select Circle and then Center Diameter Circle)
- place the center point where the axes cross and left-click
  At this point the circle size will change with mouse moves and the dimension box will show with a blue value.
- Type the first letter of the parameter you want to use, **f**, and a list of parameters starting with that letter will appear.
- select the **Fin_TubeOD** parameter in the list and then press **Enter**. OD stands for *Outer Diameter*.



If a tool icon is ever missing open the **CREATE** or **MODIFY** menu to find it.

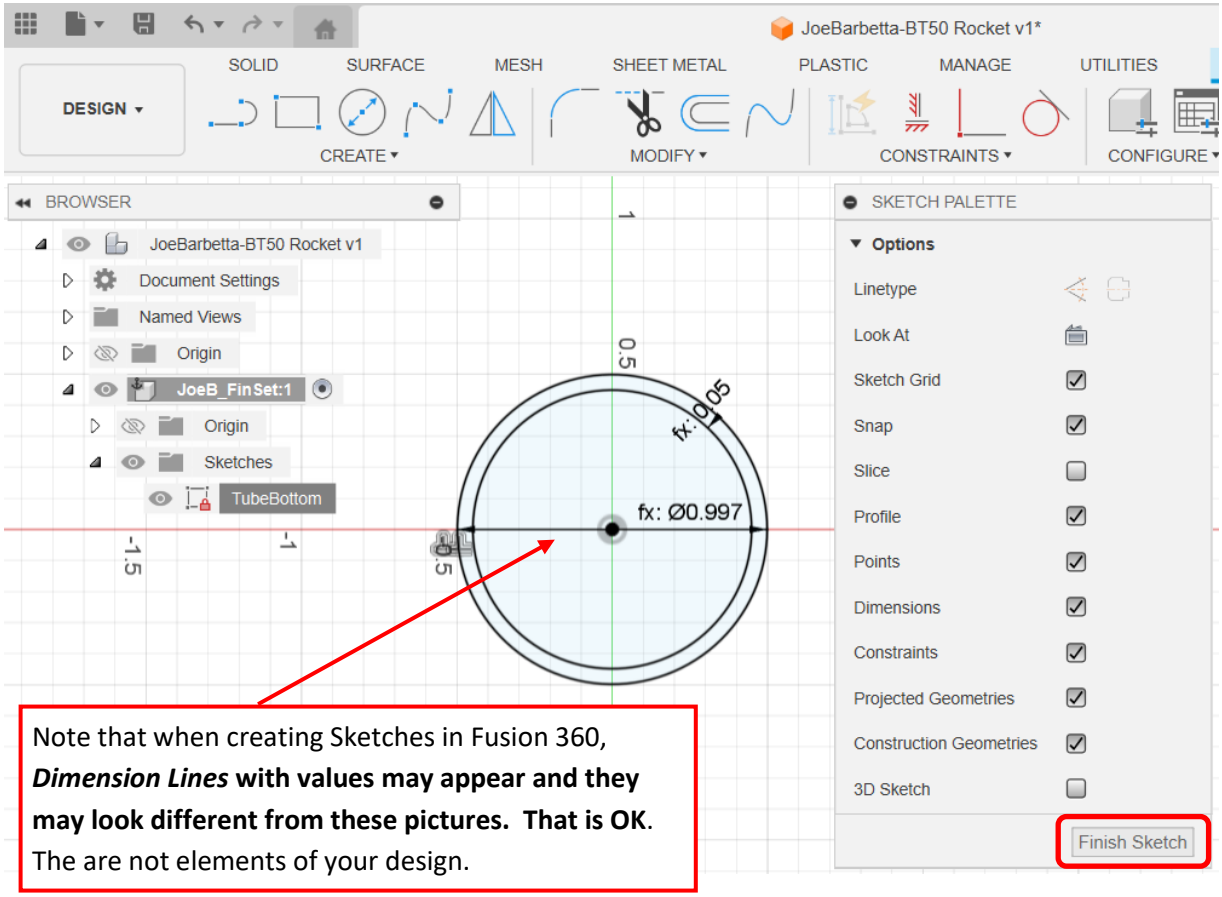
Note that your screen may look different. Here, zooming in with the mouse scroll wheel was used. Holding the mouse wheel down will allow panning left and right or up and down.

- over MODIFY select the **Offset tool**, type **f** and select **Fin_TubeThickness**.
- you may have to **click the Flip icon** to move the **red circle inside the blue circle** and click **OK**



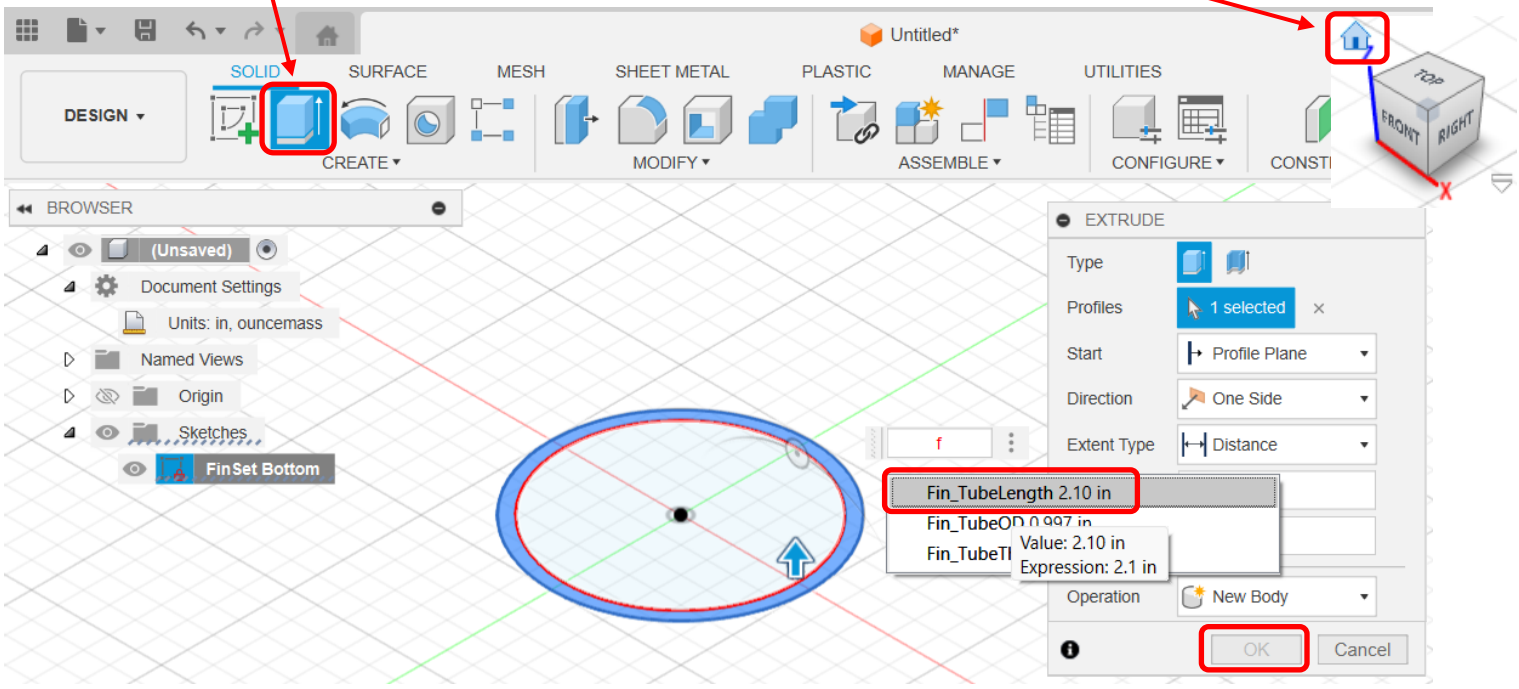
As with any program, **Undo** can be used to reverse operations.

- click **Finish Sketch**.



Note that when creating Sketches in Fusion 360, ***Dimension Lines*** **with values may appear and they may look different from these pictures. That is OK**. The are not elements of your design.

- click the **Home** button at the **View Cube**
- select the top **Extrude** icon
- type **f** to get the parameter list and select **Fin_TubeLength**
- click **OK**
- **Zoom out** to view the entire cylinder



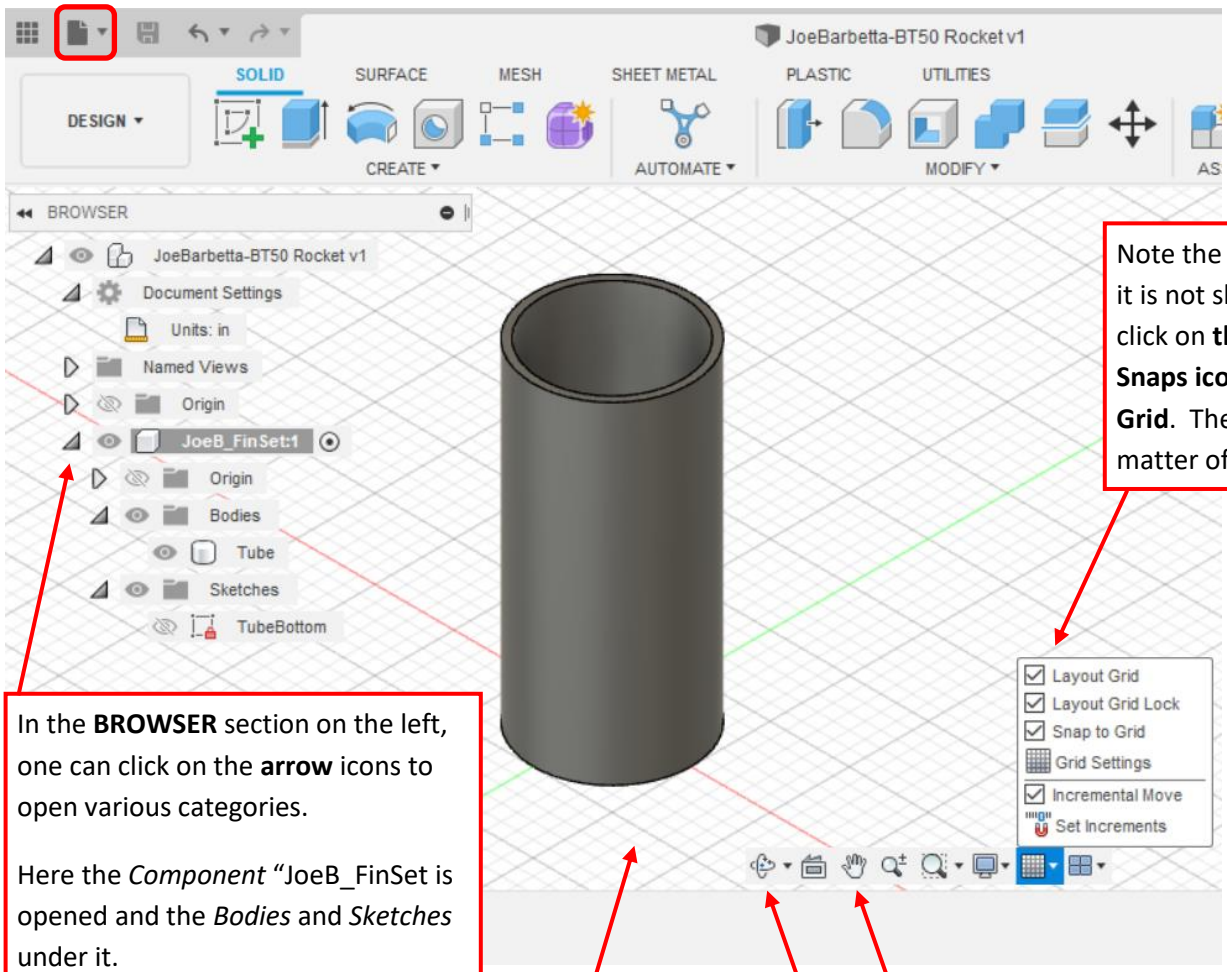
Saving Your Fusion Project

At this point your tube may look like that below.

- click on the top **File** icon and select **Save** and create a name for your design, which should show in the top tab.

From time to time save your work in case Fusion crashes. Yes. This happens.

By default Fusion 360 saves your work "in the cloud" to be opened next time you log into Fusion 360 from any computer. However, if you are afraid that adversaries may hack the AutoDesk servers and spy on your rocket program, you can save files to your computer instead. One can also select **Export** instead of **Save** if you want to save a .f3d Fusion file to a location on your computer.



Note the background grid. If it is not showing, one can click on **the bottom Grid and Snaps icon** and **check Layout Grid**. The layout grid is just a matter of preference.

In the **BROWSER** section on the left, one can click on the **arrow** icons to open various categories.

Here the *Component* "JoeB_FinSet is opened and the *Bodies* and *Sketches* under it.

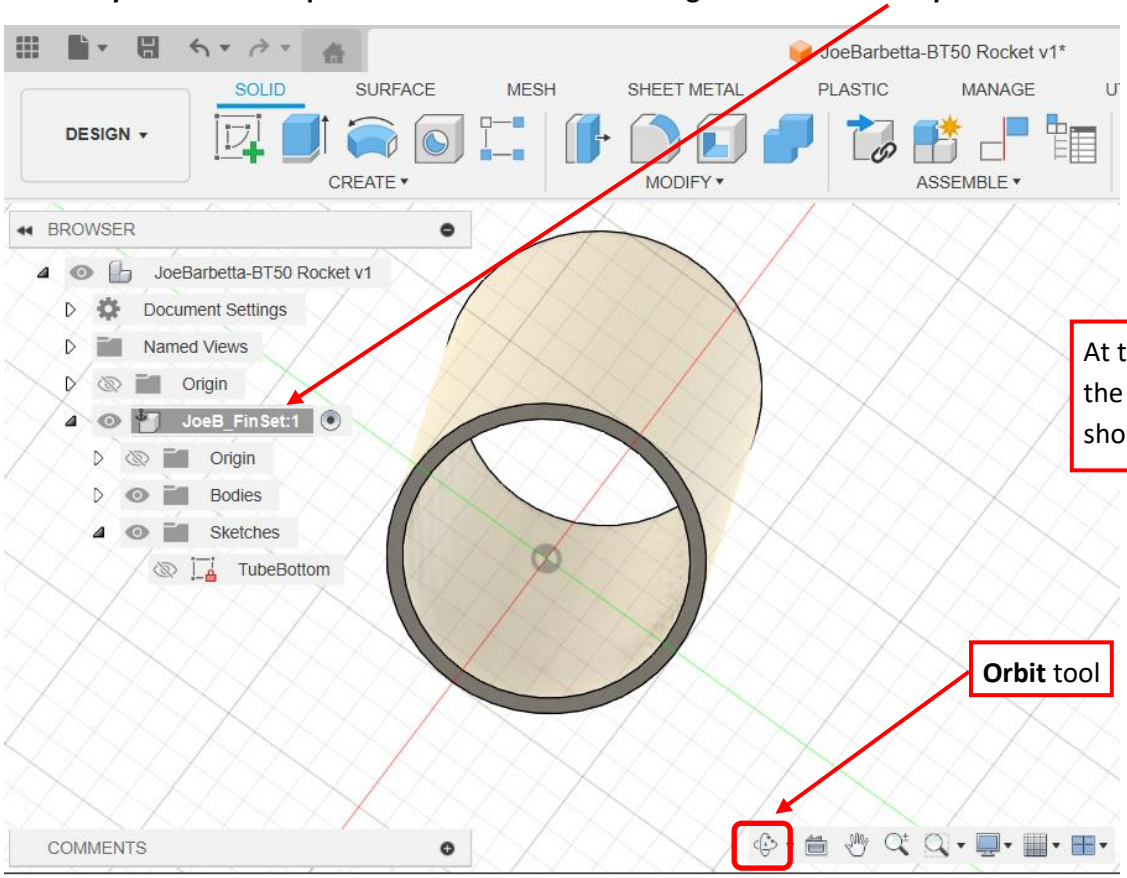
It is a good idea to **right-click on the Body and Sketch names and select Rename** to enter meaningful names.

Note that this "bottom" X-Y plane represents the print bed of the 3D printer.

When using the mouse scroll wheel to zoom in and out, the positioning can be controlled by positioning the mouse cursor before zooming in or out.

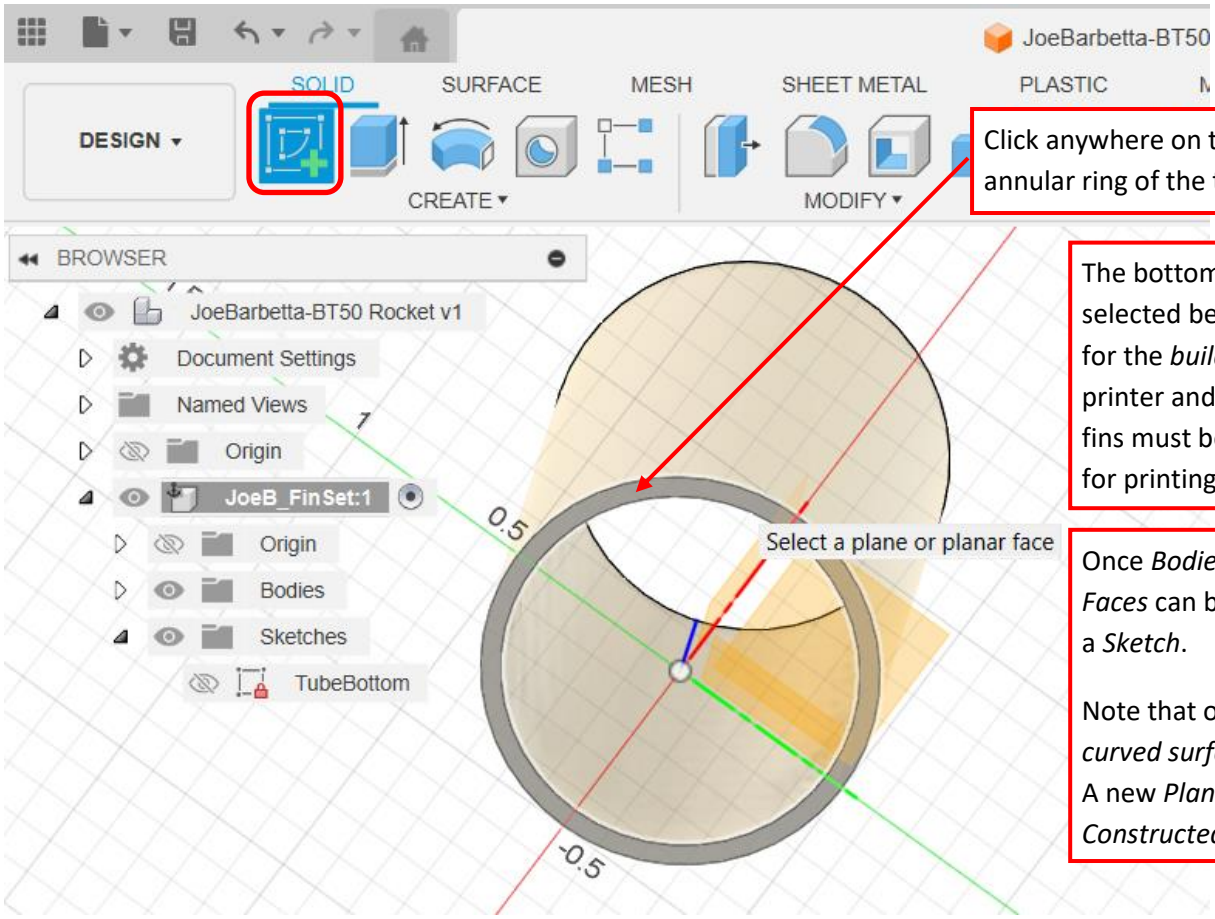
There is also the hand "Pan" tool to move the view around and the Orbit tool to adjust the view angle.

- rotate the **top right view cube** or the **Orbit** tool to position the tube to view its bottom.
- **ensure your FinSet component is still selected**. If not **right-click on the *Component* name and select Activate.**.



At this point the view cube in the top-right of the screen should show BOTTOM.

- click on the top **Sketch** icon and **click on the bottom** of the tube and **Not** on the orange square.



Click anywhere on the bottom annular ring of the tube.

The bottom of the tube is being selected because this is the plane for the *build plate* of the 3D printer and the bottom of the fins must be on the build plate for printing.

Once *Bodies* are created their *Faces* can be selected for starting a *Sketch*.

Note that one cannot select a *curved surface* to start a sketch. A new *Plane* would have to be *Constructed* to do so.

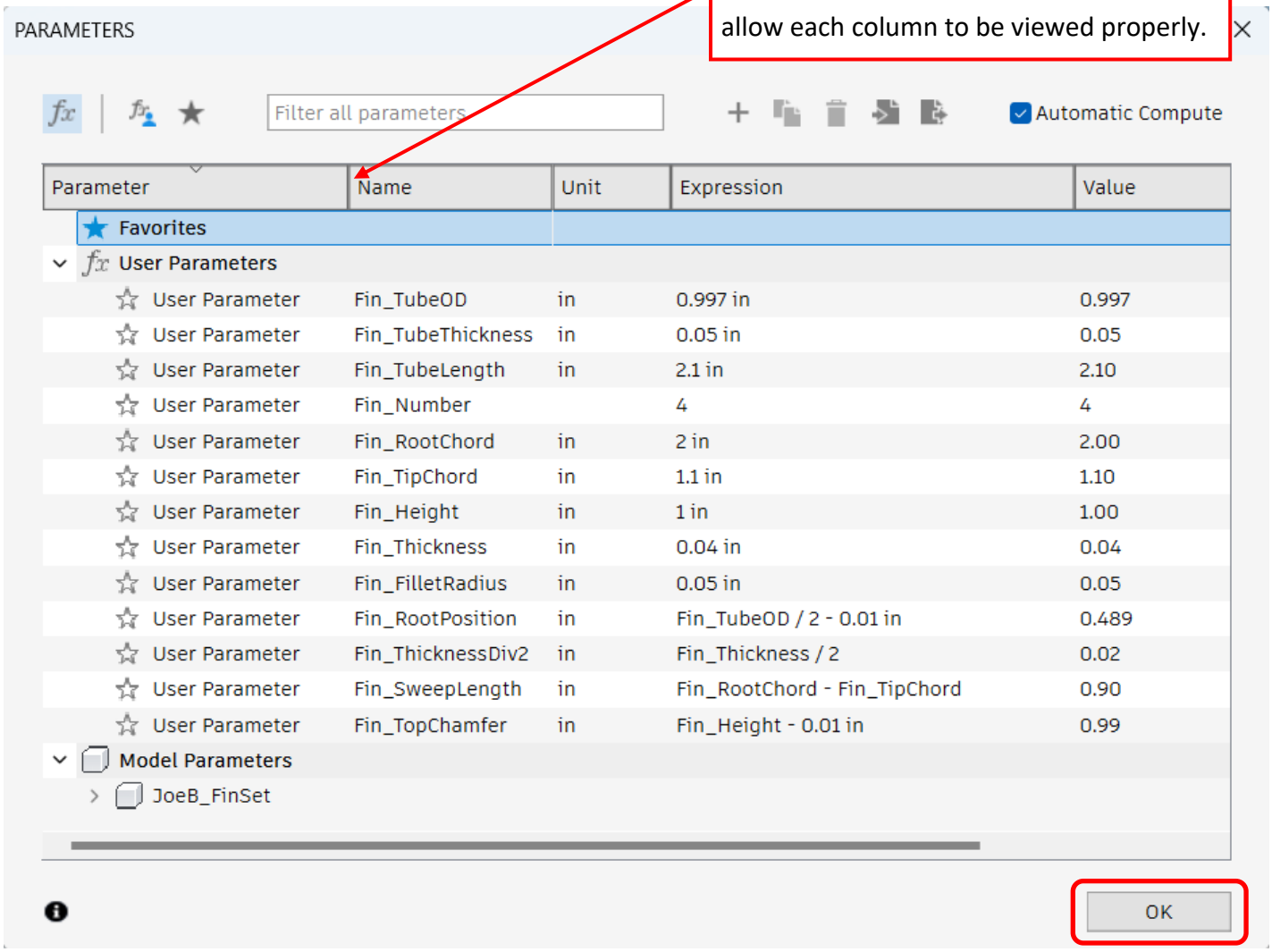
As done previously, open the *Parameters window* and add additional parameters.
- from the top **MODIFY** menu select **Change Parameters** near the bottom of the list
- next to **User Parameters** click on **+** and enter the **Name** and **Expression** for each item in the below list. Note that you will need to click **+** for each item. You may be able to copy and paste Expressions, but **if it shows as red you may have to type the expression.**
- **Don't forget to click OK when done!**

Name	**Expression**
Fin_Number	4 (Change units to **No Units** for this one)
Fin_RootChord	2.000
Fin_TipChord	1.100
Fin_Height	1.000
Fin_Thickness	0.040
Fin_FilletRadius	0.050
Fin_RootPosition	Fin_TubeOD / 2 – 0.01
Fin_ThicknessDiv2	Fin_Thickness / 2
Fin_SweepLength	Fin_RootChord – Fin_TipChord
Fin_TopChamfer	Fin_Height – 0.01

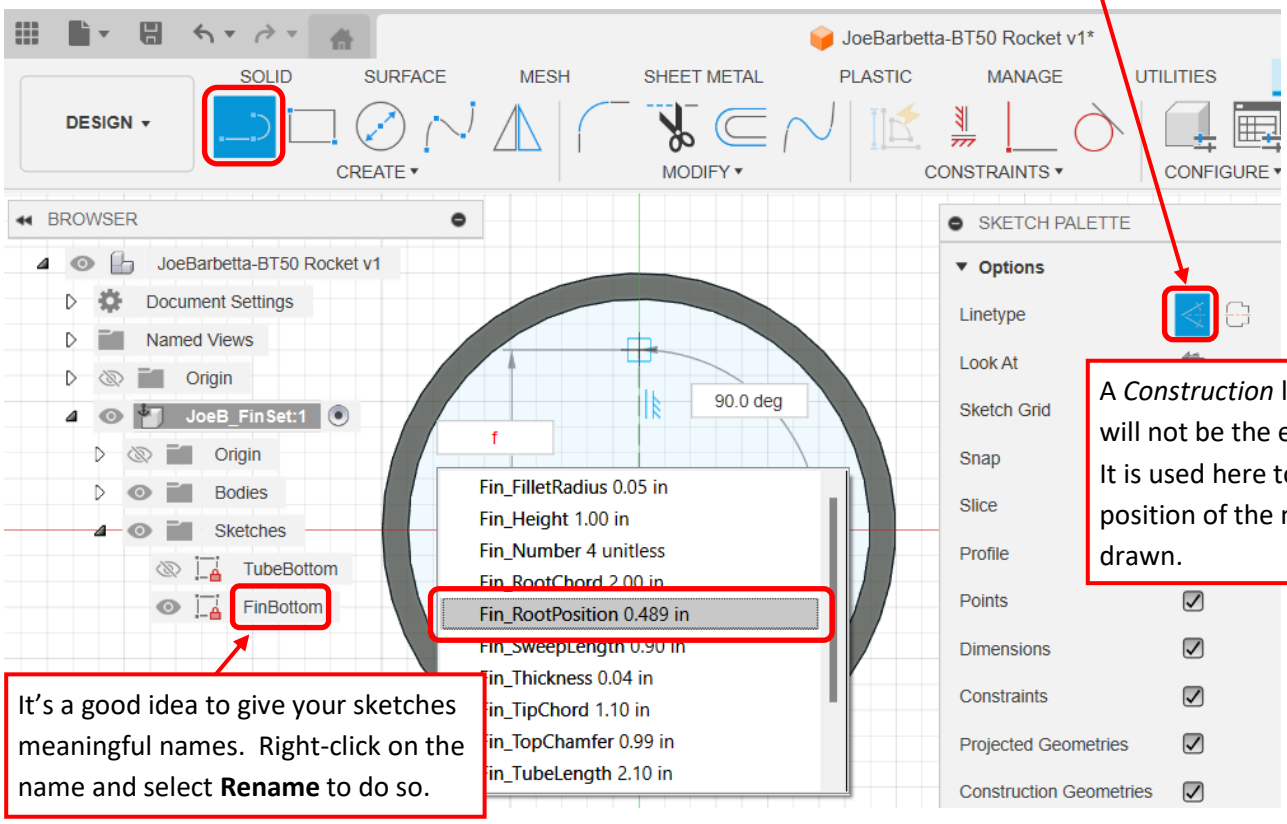
Some names are shown in blue to indicate that they match the Trapezoidal fin set parameters used in the OpenRocket simulator software.

The Parameters window should appear as below.

The borders between column titles can be clicked on and moved with the mouse allow each column to be viewed properly.



- click on the **Line** icon and on the Sketch Palette window click on the **1st icon next to Linetype** for a *Construction* line.
- draw the line **from the center of the circle** upward and close to the circle's edge
- type **f** and select **Fin_RootPosition**

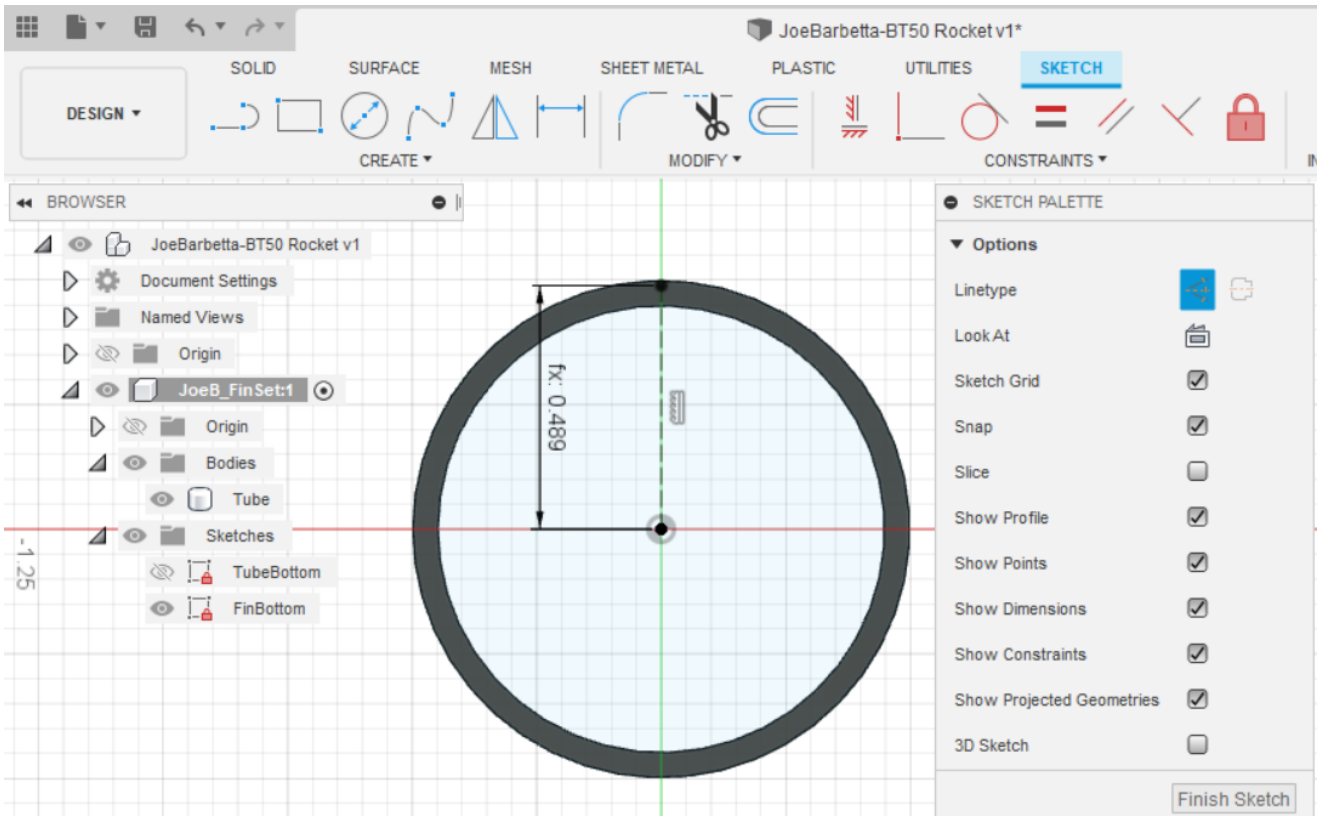


A *Construction* line is a line that will not be the edge of a feature. It is used here to control the position of the next lines to be drawn.

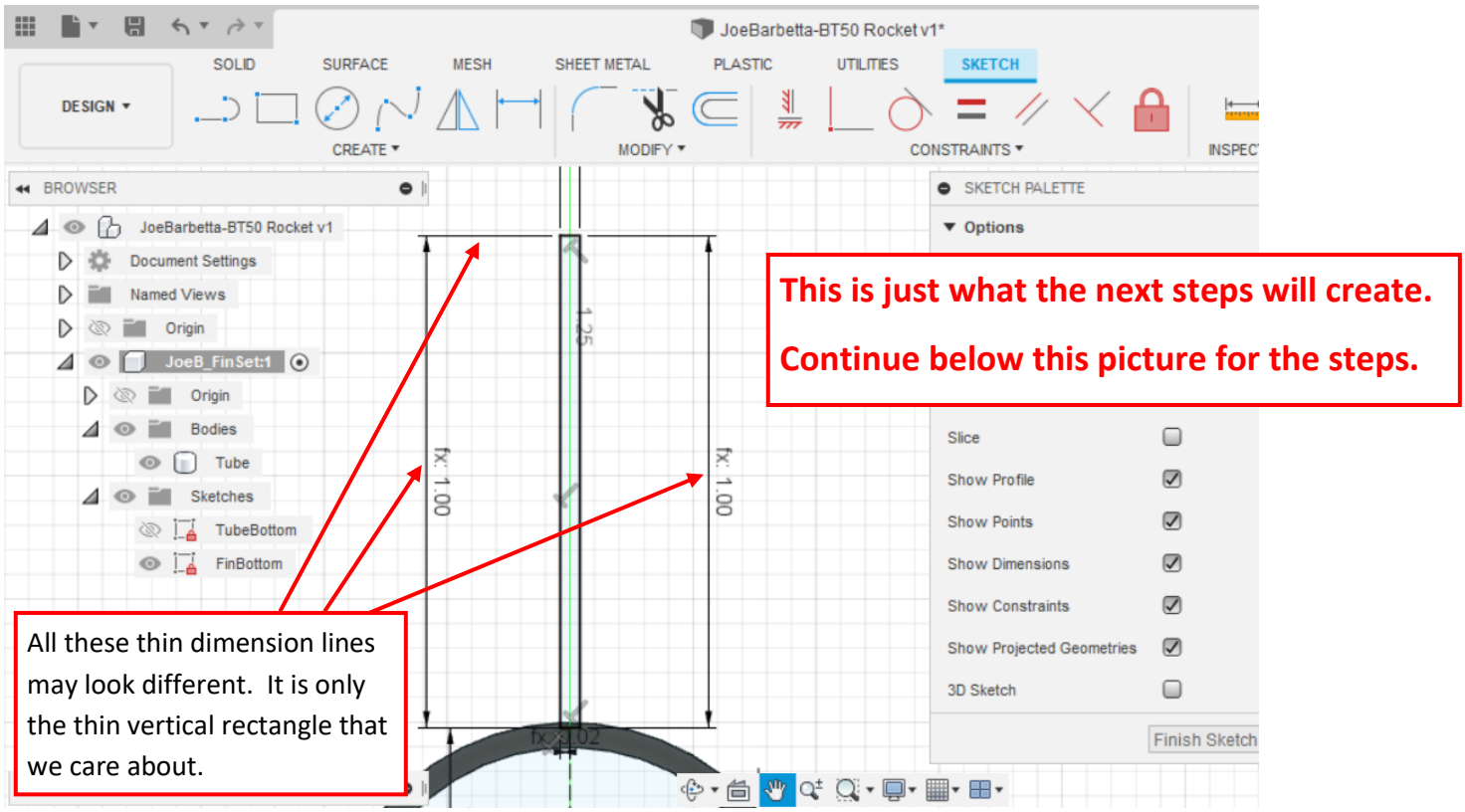
It's a good idea to give your sketches meaningful names. Right-click on the name and select **Rename** to do so.

This is what that line should look like. The dash pattern of the line indicates it is a Construction line.
It is OK if the thin black dimension lines look different.

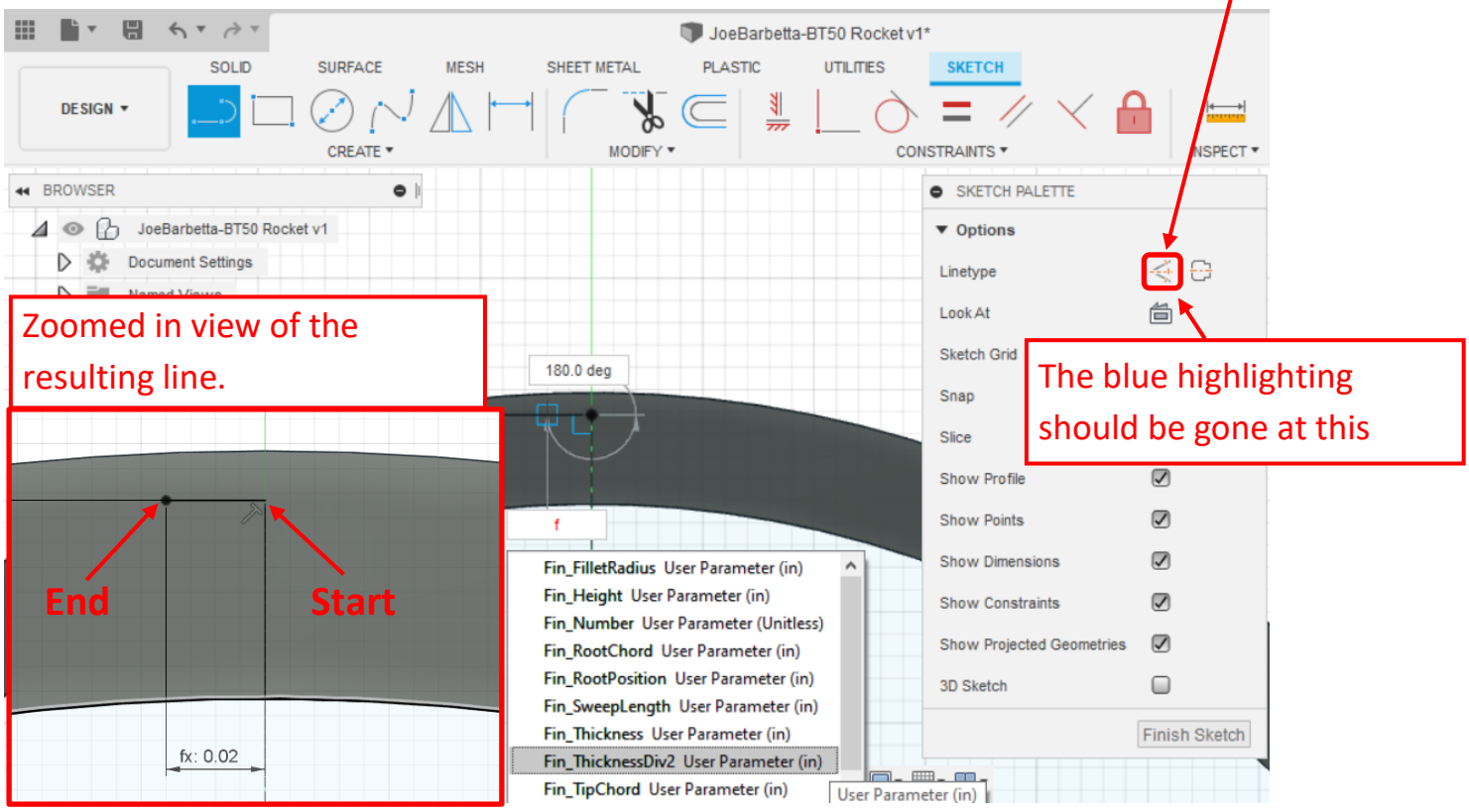
Do Not click Finish Sketch after each line. You will only do so after the entire Sketch is done.



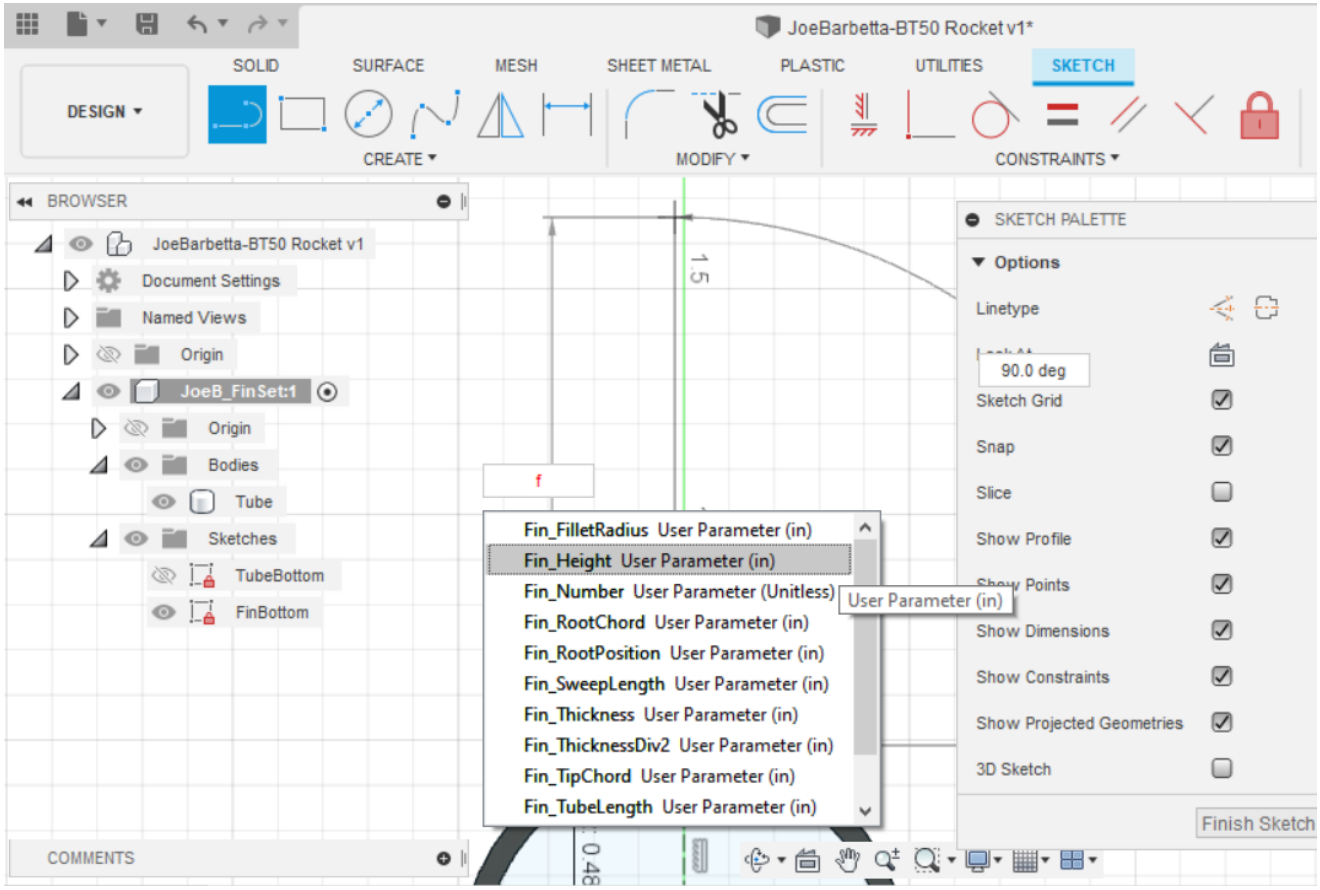
The steps after the below picture may be a difficult, but **try not to say bad words**.
As shown below, the end result will be a **vertical rectangle extending up from the tube**. This will be the bottom of a fin.
As mentioned, **don't worry if the thin dimension lines look different**. It is the center rectangle that we care about.



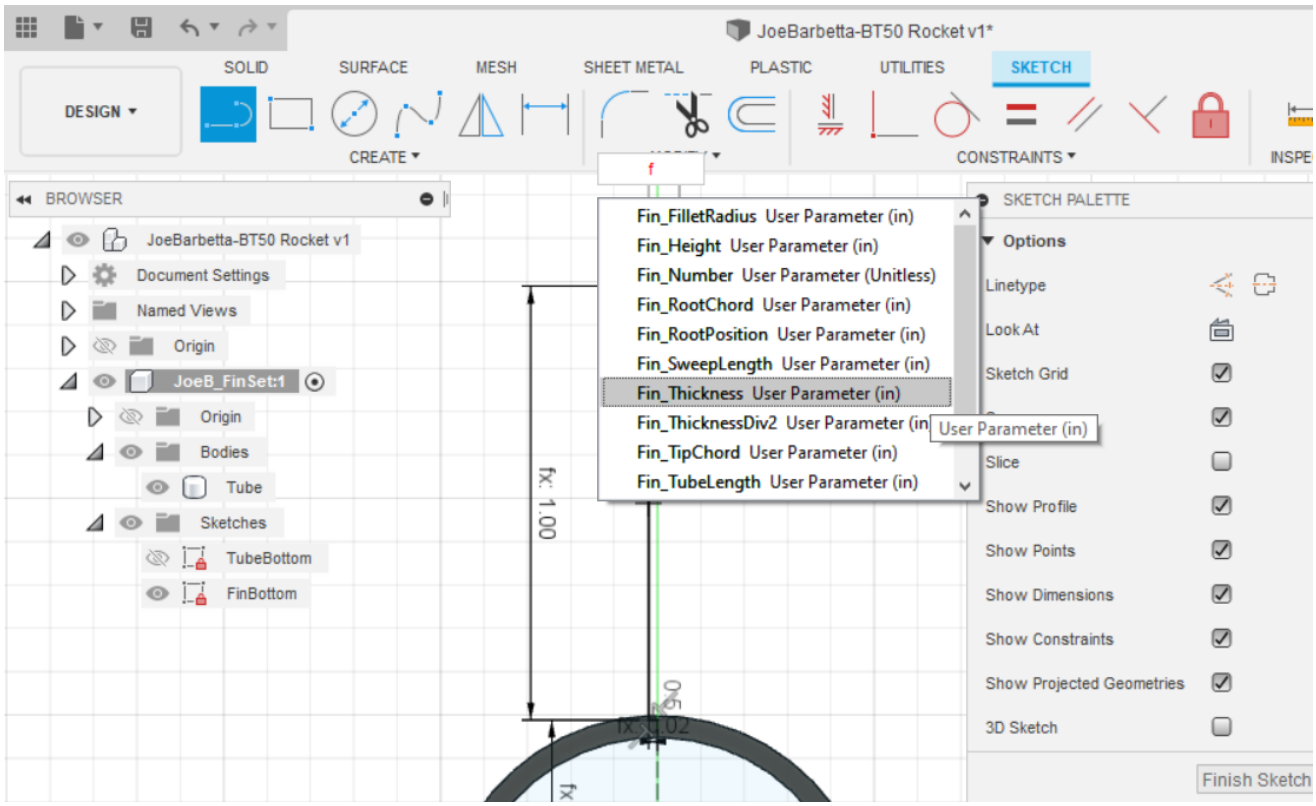
- using the **mouse wheel**, zoom into the top of the circle. You can use the **bottom hand icon** to reposition the view if needed.
- click on the **Construction line icon** again to remove the blue highlighting to return to drawing normal lines.
- draw a **line** from the top of the last line and to the left and type **f** to select **Fin_ThicknessDiv2**.



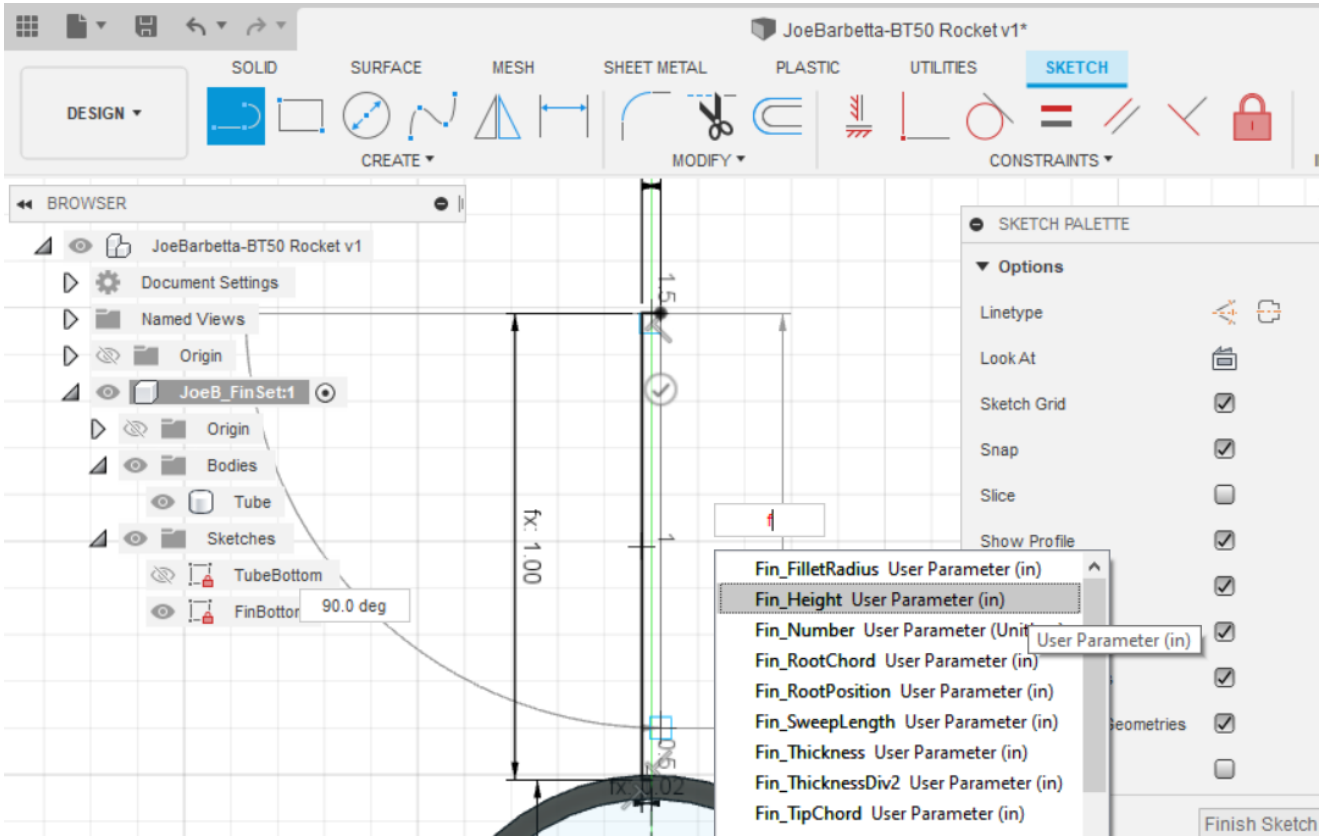
- draw a **line** the last line upward and type **f** to select **Fin_Height**. You will need to Zoom out for this.



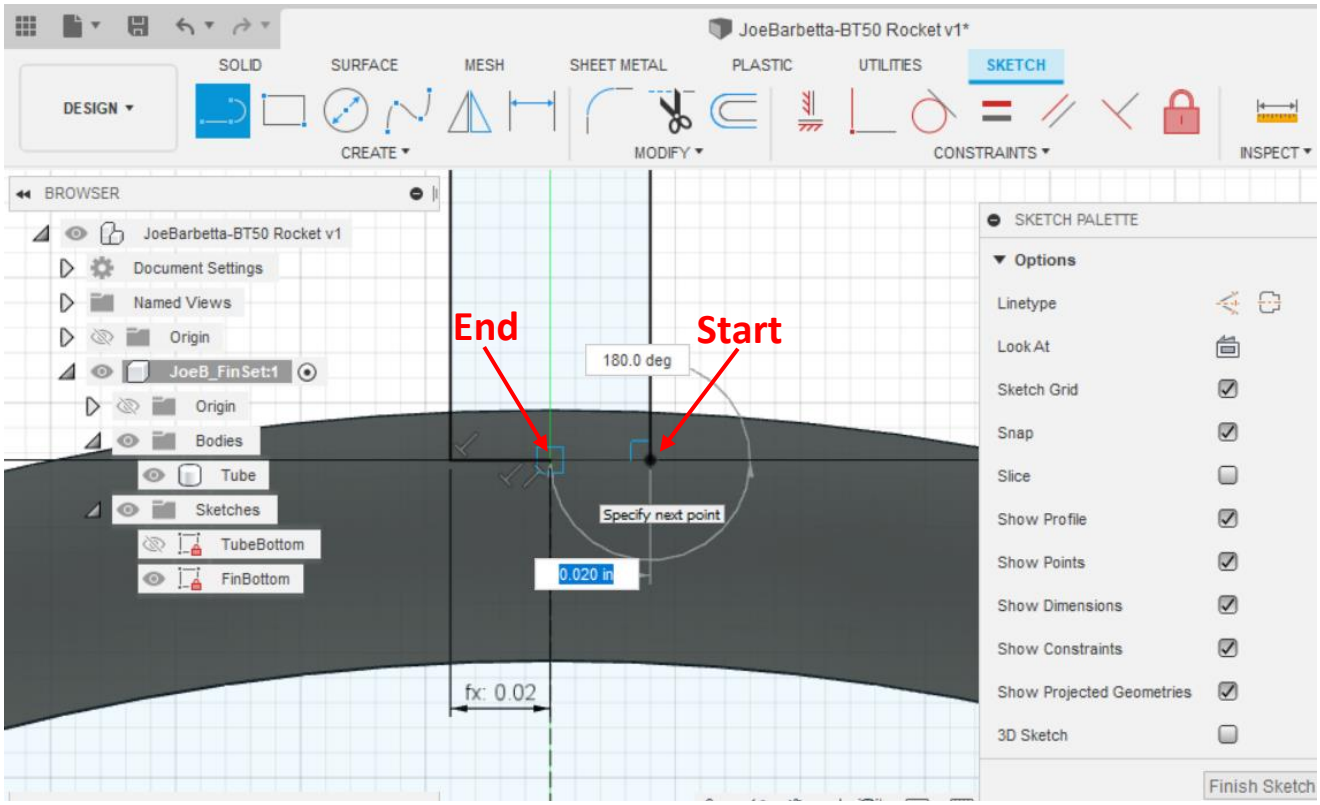
- from that point draw a line a small amount to the right, type **f** and select **Fin_Thickness**.



- from that point draw a line down, type **f** and select **Fin_Height**.

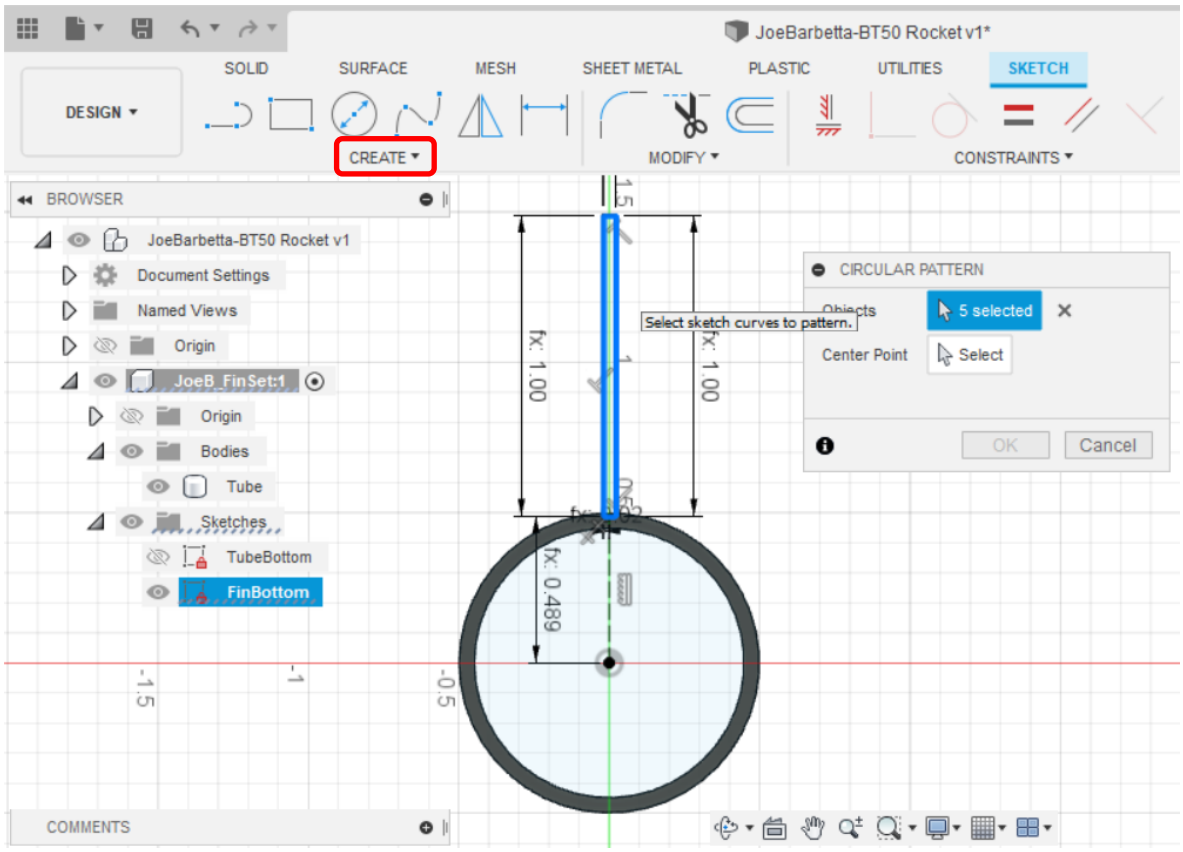


- from that point draw a to the left slightly to close the rectangle.

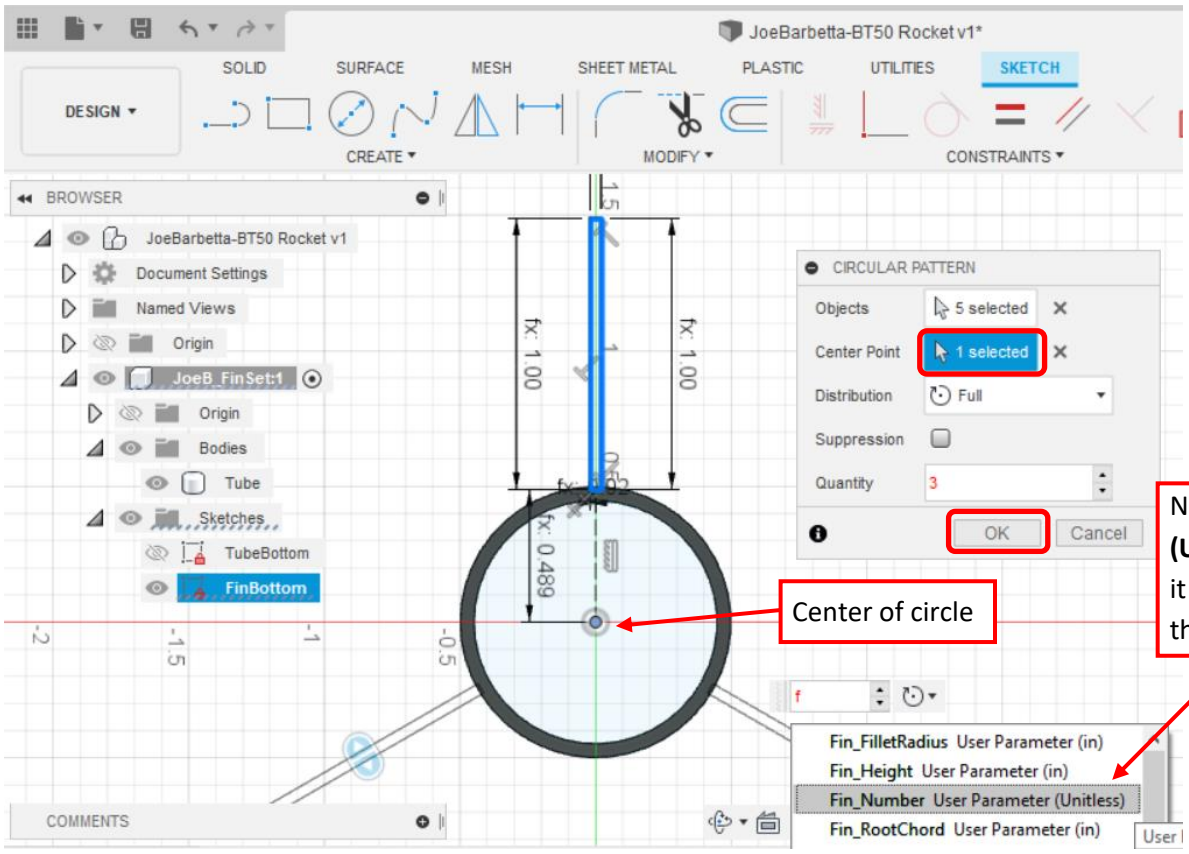


Creating a Circular Pattern

- **zoom out** and click on **CREATE** and select **Circular Pattern** near the bottom of the list.
- **double-click on any edge of the rectangle**. This should cause each edge to turn blue.

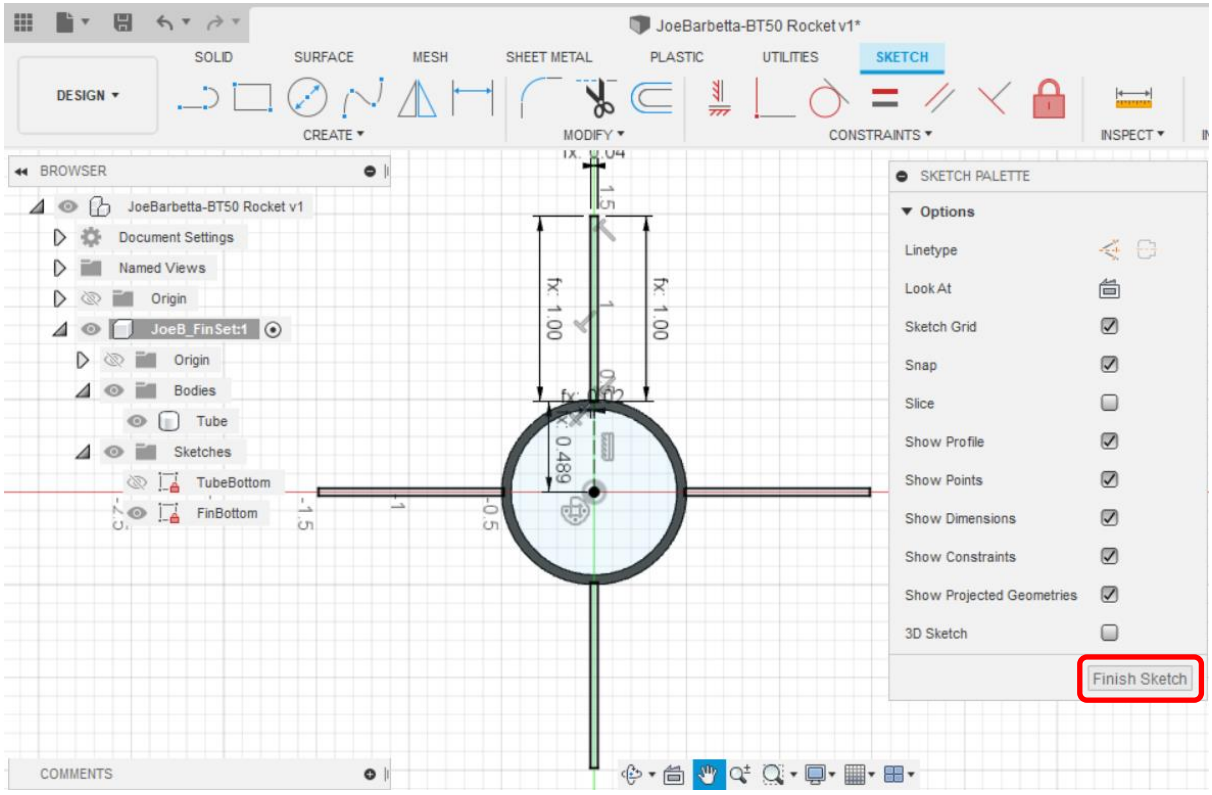


- click **next to Center Point** and **click on the center of the circle**.
- type **f** and select **Fin_Number**. Then click **OK**.

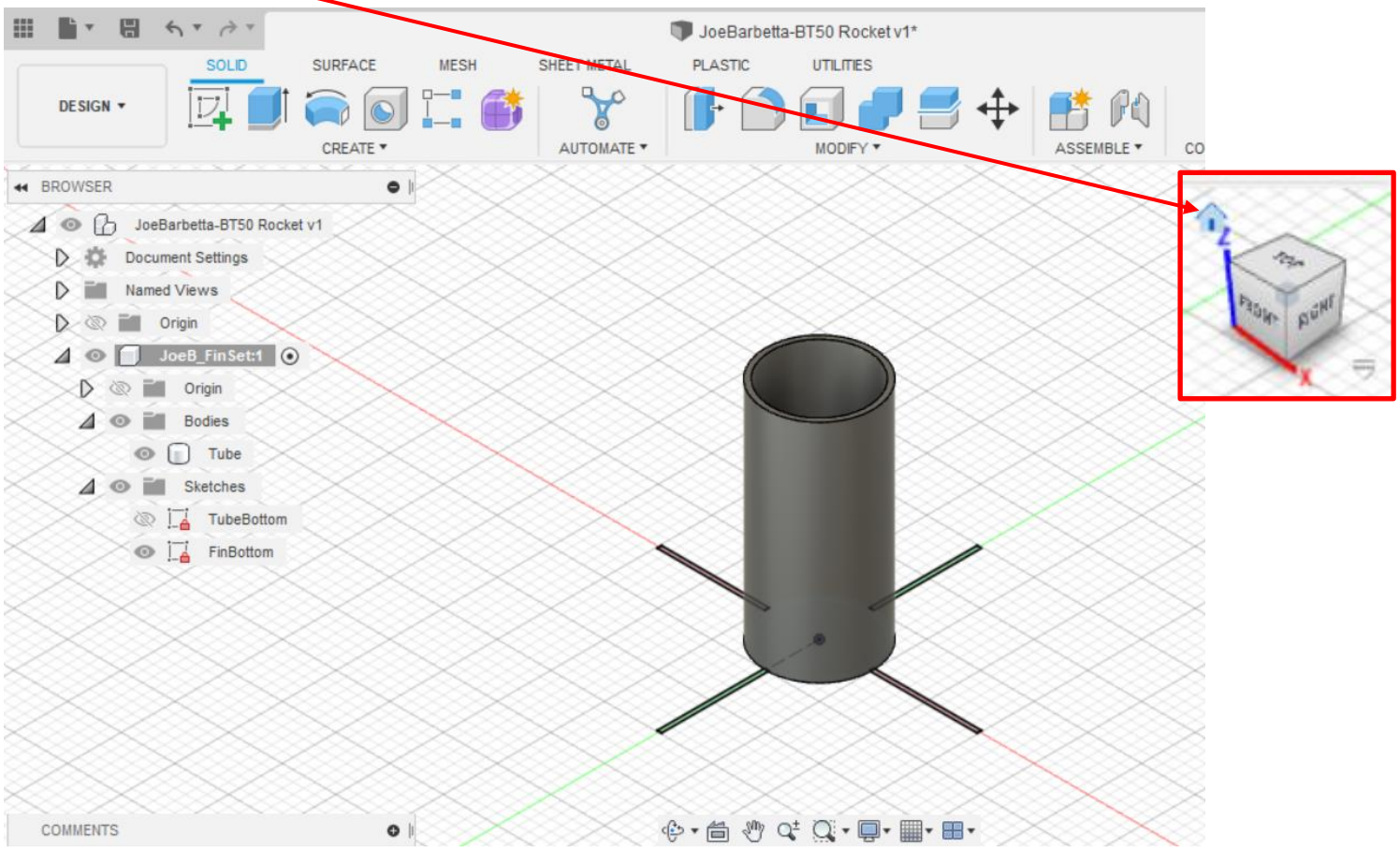


Note that Fin_Number is **(Unitless)**. If it shows (in), it must be corrected in the Parameters window.

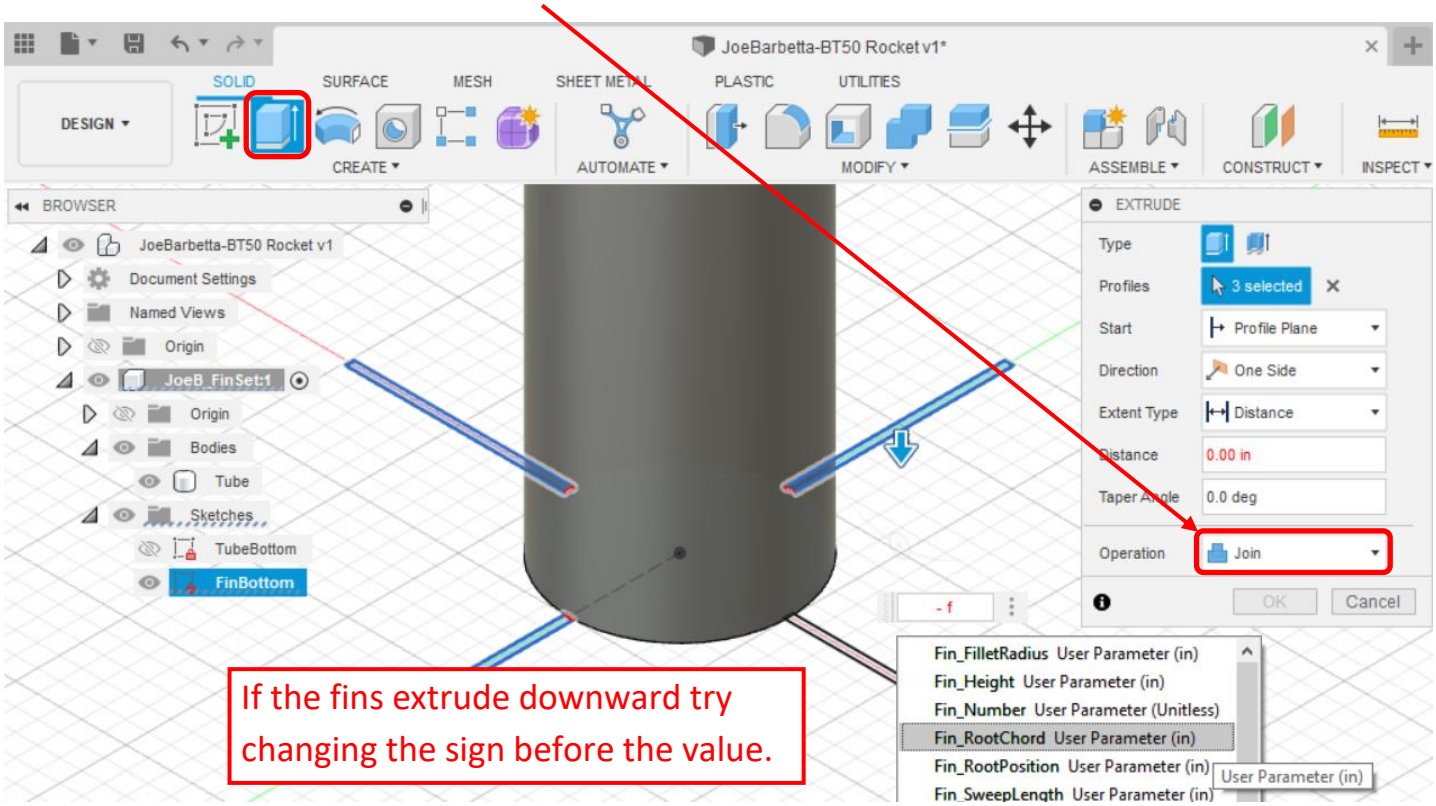
- click **Finish Sketch**.



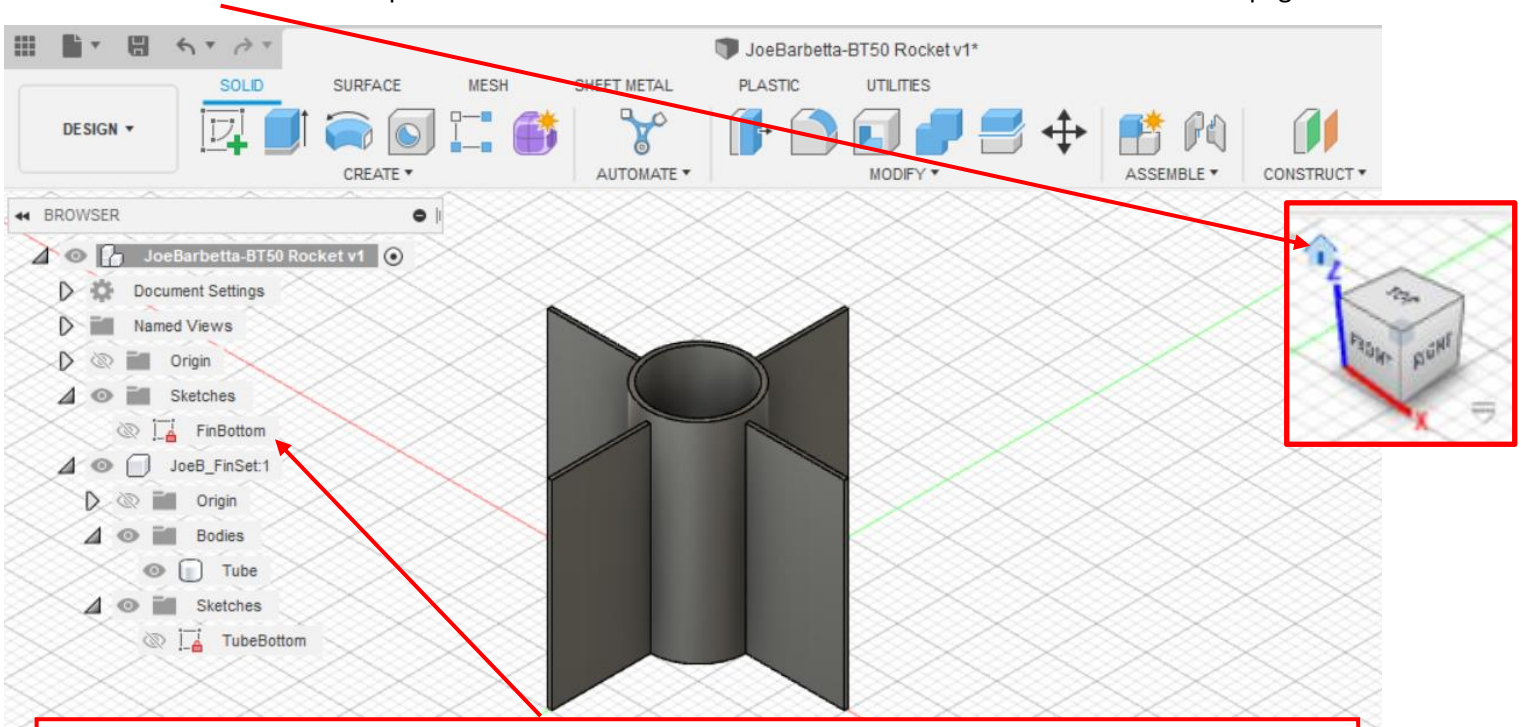
Click on the **Home** icon at the top-left of the **View Cube**. This will reset the view.



- **zoom in** and position, using the bottom hand icon if needed, the view as shown below.
- click of top **Extrude** tool and click on each fin *Profile* rectangle, which will cause then to turn blue.
- type **- f** and select **Fin_RootChord**. The **minus sign must be typed first** to cause the fins to be extruded up. Otherwise, they will extrude down. We need the negative because we created the Sketch from the underside.
- ensure that **Join** is selected for the **Operation**. Sometimes the operation can change and it has to be changed again.



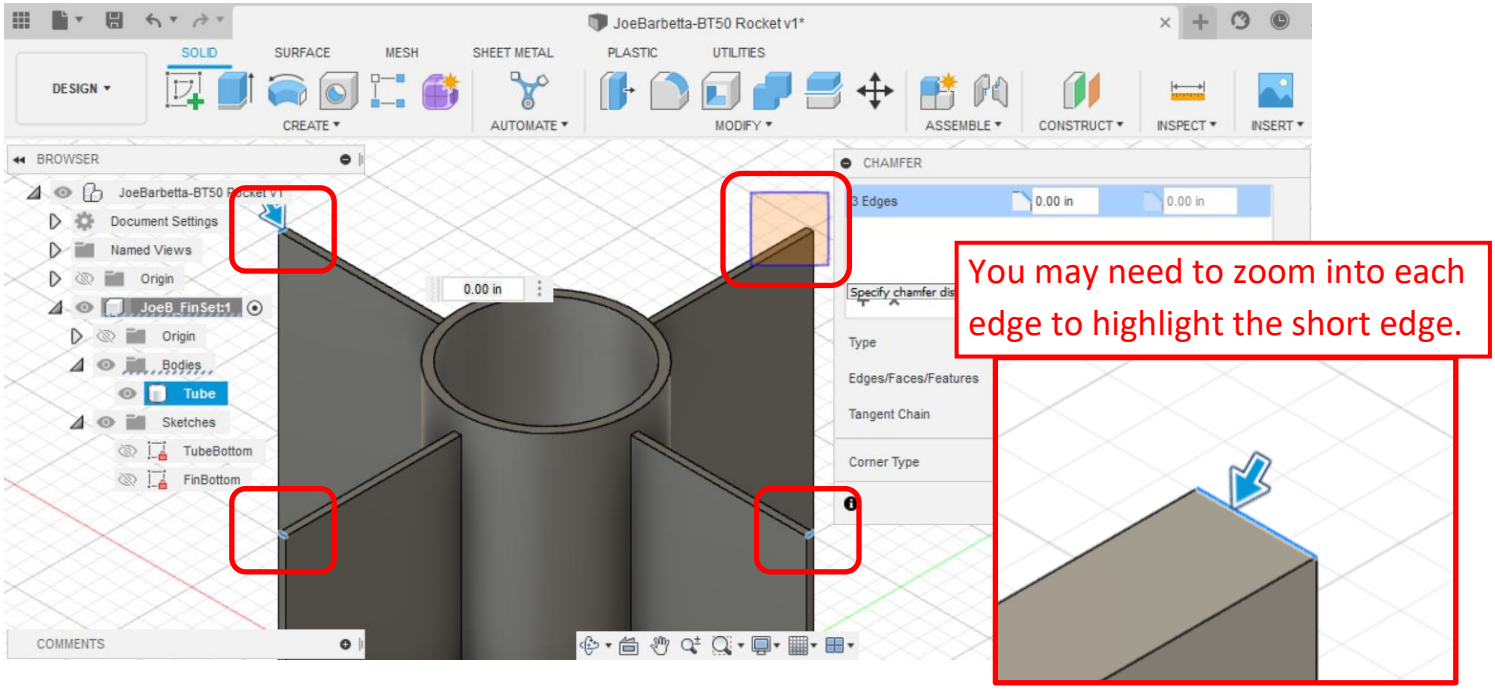
Click on the **Home** icon at the top-left of the **View Cube**. This will reset the view. See note at bottom of page.



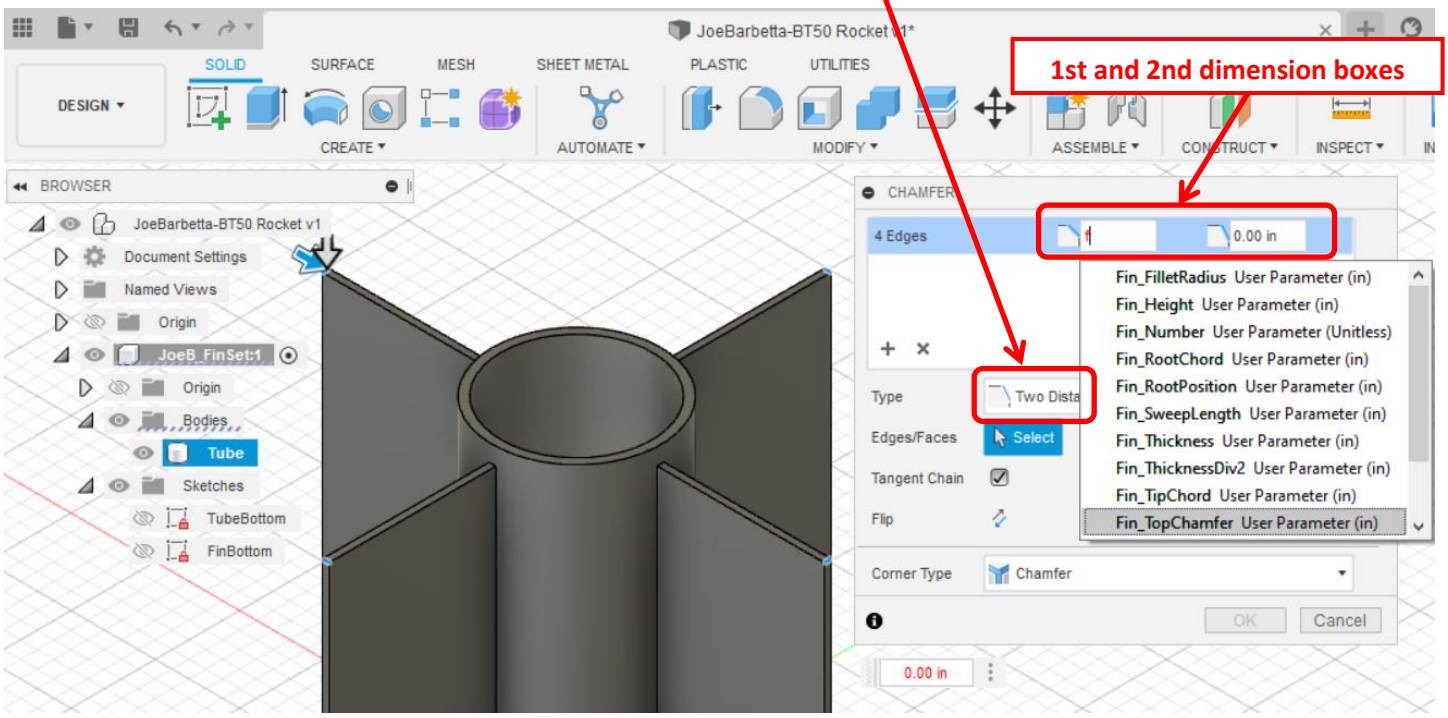
Here is shown the result of a common mistake, wherein the **FinBottom** *Sketch* was started without the **JoeB_FinSet** being the *Active Component*. If this occurred, **right-click on the *Sketch* name and select Activate** and then **drag the *Sketch* name onto the FinSet *Component* name**, e.g. JoeB_FinSet here.

Using the Chamfer Tool

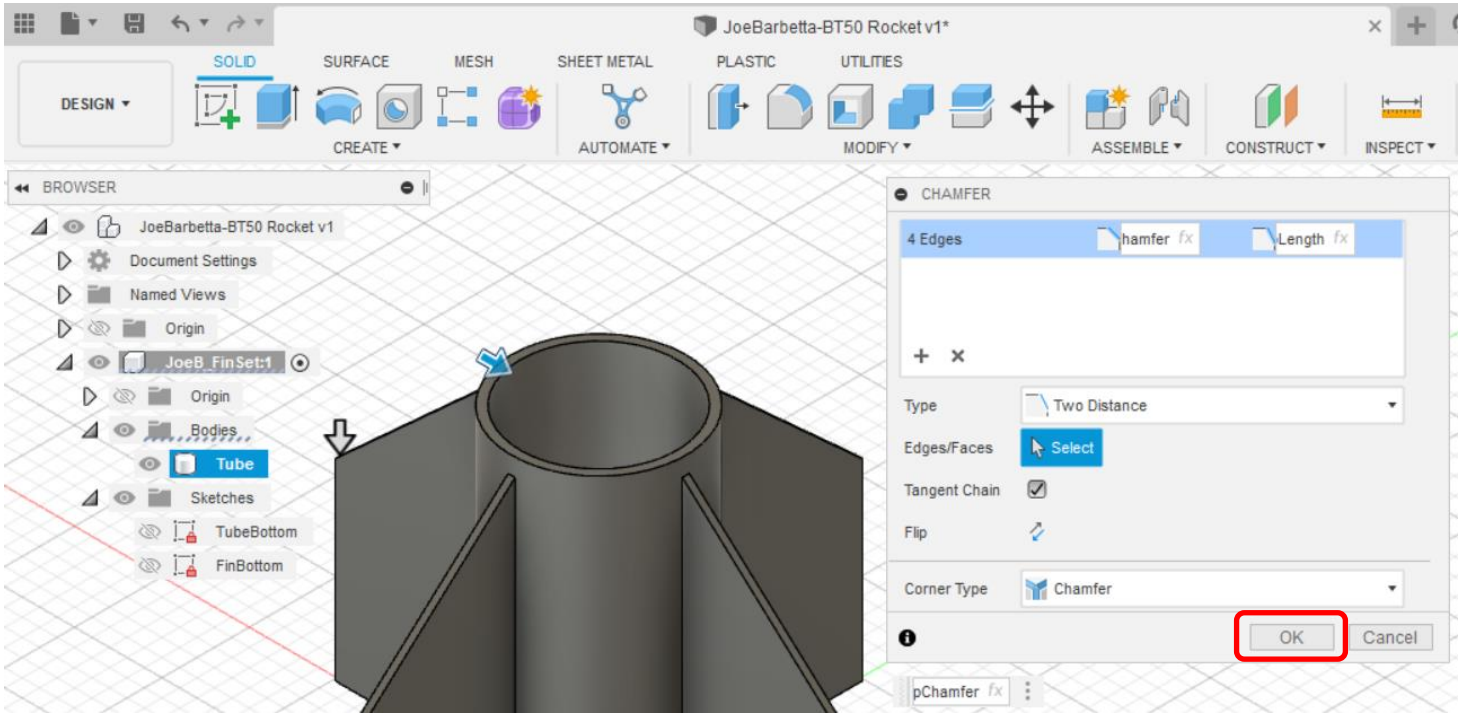
- under **MODIFY** select **Chamfer**
- each of the 4 top outer edges must be selected by **clicking on the background near the upper left of the edge** and then **releasing the mouse button after surrounding the edge with the Selection rectangle**. Below the last edge is being selected. Each edge should turn blue. Yes. They are small and hard to see.



- in the **CHAMFER** window change **Type** from **Equal Distance** to **Two Distance**
- in the **1st dimension box** type **f** and then select **Fin_TopChamfer**.
- in the **2nd dimension box** type **f** and then select **Fin_SweepLength**



- when the parameters are selected, the FinSet Component should look as below.
- click **OK**

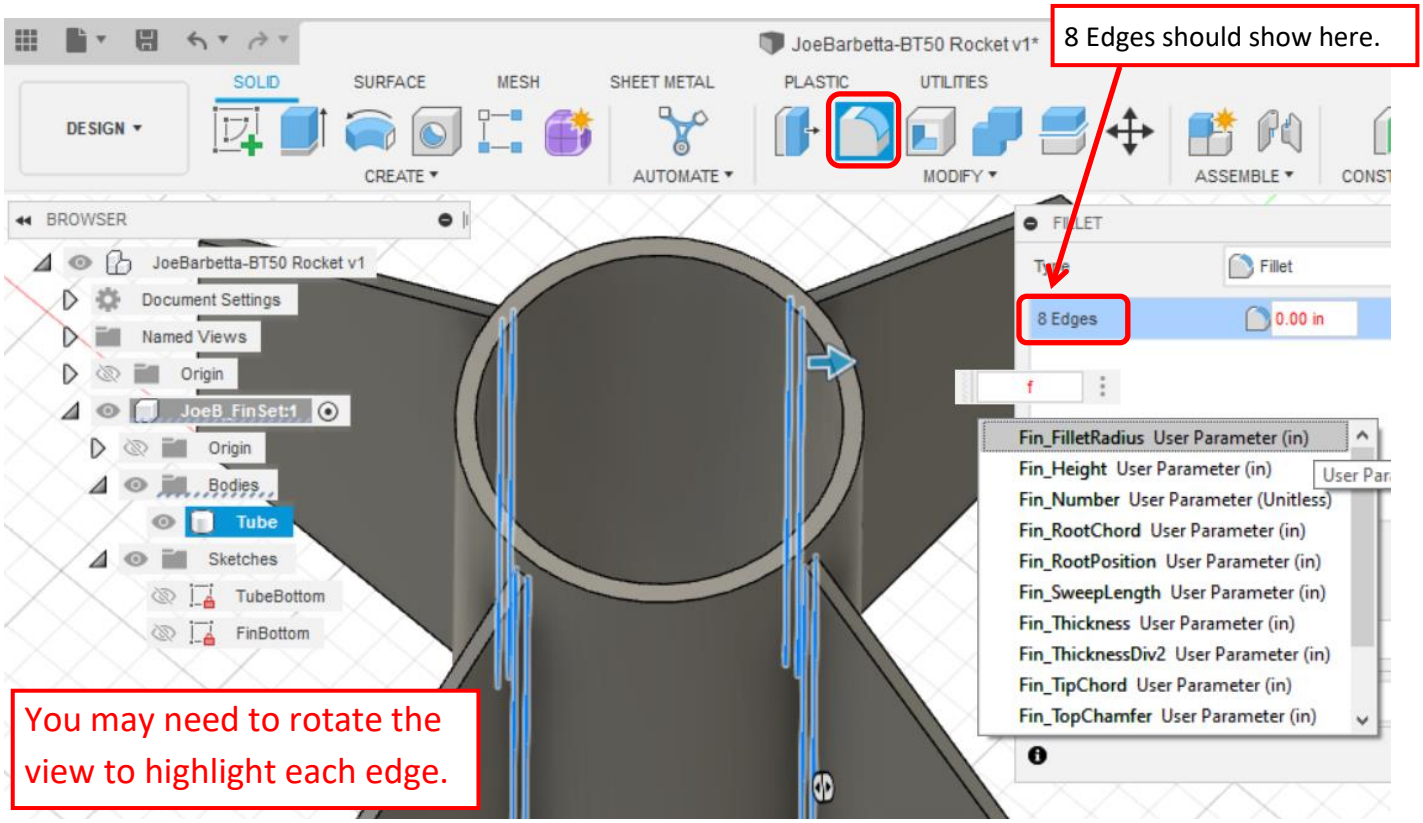


Using the Fillet Tool

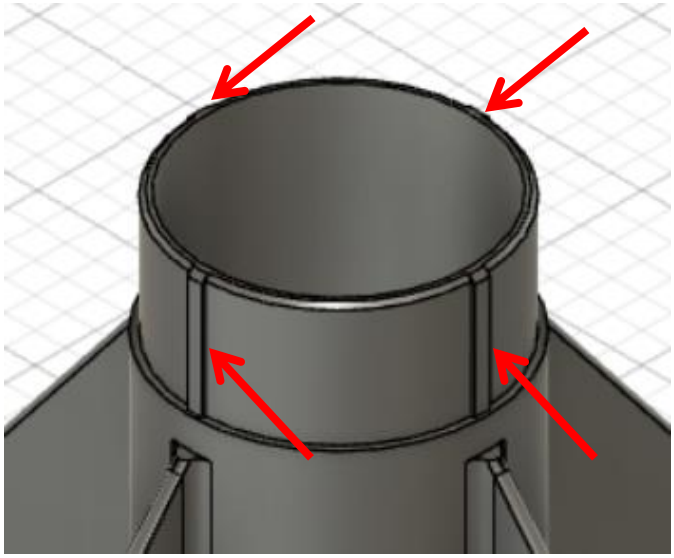
- select the top **Fillet** tool or select it from the **MODIFY** menu
- adjust the view to be similar to below

Pronounce the **t** in Fillet and Not like Fillet of fish.

- click on the edges where the fins connect to the tube. You should be able to select those on the other side of the tube that are hidden. This may be a pain in the butt. There should be **8 edges in total that are highlighted in blue** as shown below.
- type **f** and select **Fin_FilletRadius**. Then click **OK** and the fillets should show.



Fin Tube Coupling Ribs



3D printed parts do not always print accurately. For example, the inner and outer dimensions of a tube can vary about 0.01". This tolerance doesn't matter for many dimensions, however, when components must fit together this can be a problem.

If the resultant outer diameter of a tube is larger than designed, it may have to be sanded down to fit inside another component. Plastic, especially PLA, which is the most common 3D printing material, is laborious to sand.

One solution is to make design the tube with a slightly smaller outer diameter and add ribs in the design. The design on the left has four ribs around the tube. This can result in a more reliable "push fit" into another tube. To correct the fit, the ribs can be sanded as needed or shaved down. It is much easier to sand down thin ribs compared to the entire tube outer surface.

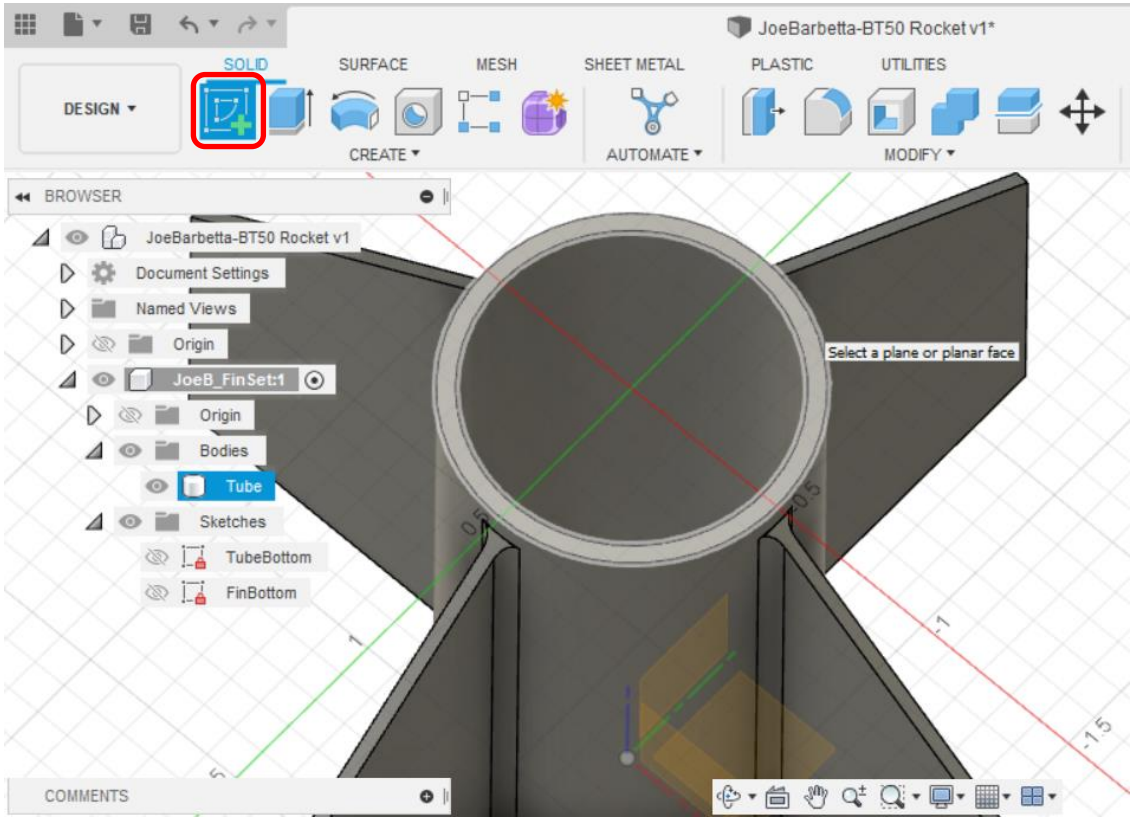
As done previously, open the *Parameters window* and add additional parameters.
- from the top **MODIFY** menu select **Change Parameters** near the bottom of the list
- next to **User Parameters** click on **+** and enter the **Name** and **Expression** for each item in the below list. Note that you will need to click **+** for each item. You may be able to copy and paste Expressions, but **if it shows as red you may have to type the expression.**
- **Don't forget to click OK when done!**

Name	**Expression**
Fin_ TubeCouplingID	Fin_TubeOD - Fin_TubeThickness * 2
Fin_TubeCouplingOD	0.940
Fin_TubeCouplingLength	0.400
Fin_TubeCouplingRibOD	0.970
Fin_TubeCouplingRibPos	Fin_TubeCouplingID / 2 + 0.01

The values just entered appear as below.

☆	User Parameter	Fin_TubeCouplingID	in	Fin_TubeOD - Fin_TubeThickness * 2	0.897
☆	User Parameter	Fin_TubeCouplingOD	in	0.94 in	0.94
☆	User Parameter	Fin_TubeCouplingLength	in	0.4 in	0.40
☆	User Parameter	Fin_TubeCouplingRibOD	in	0.97 in	0.97
☆	User Parameter	Fin_TubeCouplingRibPos	in	Fin_TubeCouplingID / 2 + 0.01 in	0.459

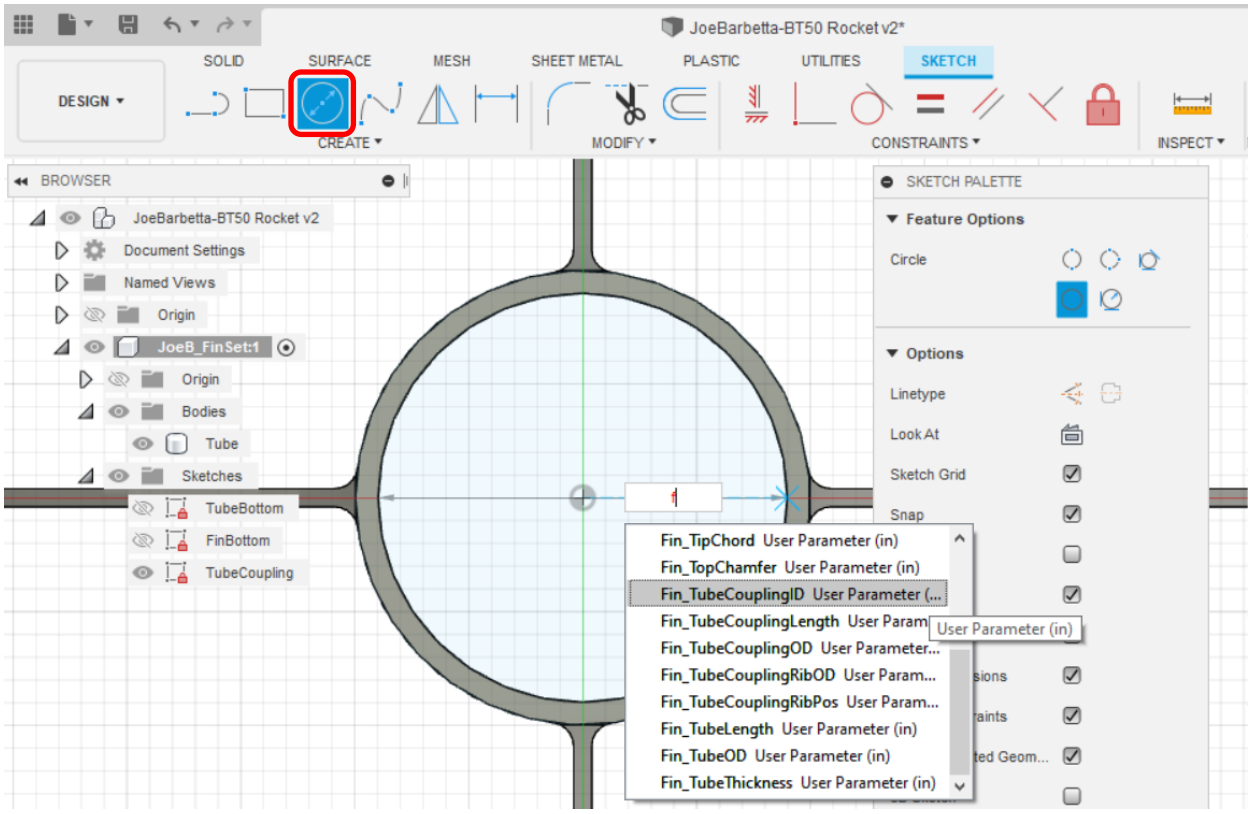
- click on the top **Sketch** icon and **click on the top of the fin set**.



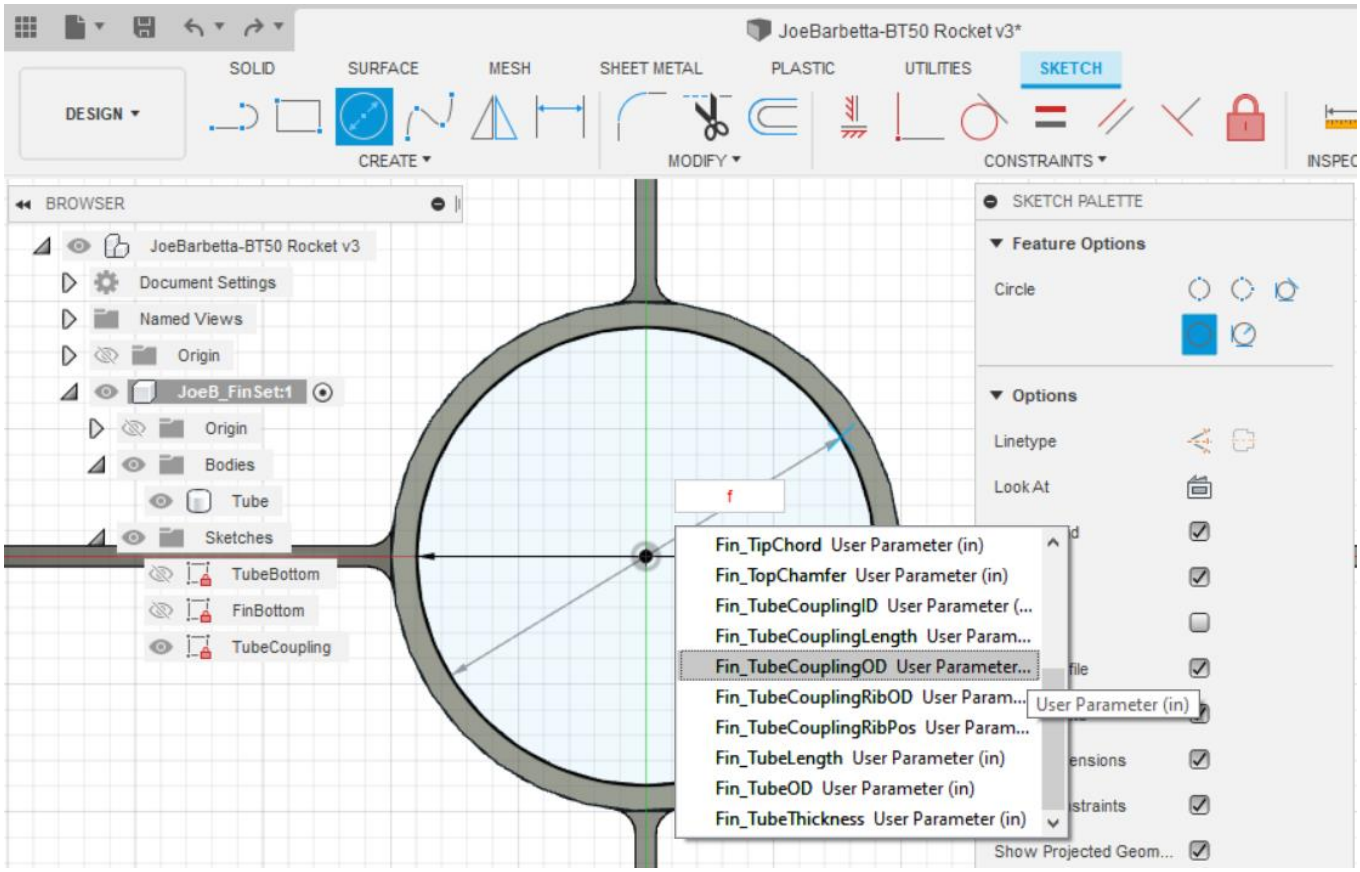
- select the **Circle tool** over **CREATE**
- place the center point at the center of the tube and left-click

At this point the circle size will change with mouse moves and the dimension box will show with a blue value. Type the first letter of the parameter you want to use, **f**, and a list of parameters starting with that letter will appear.

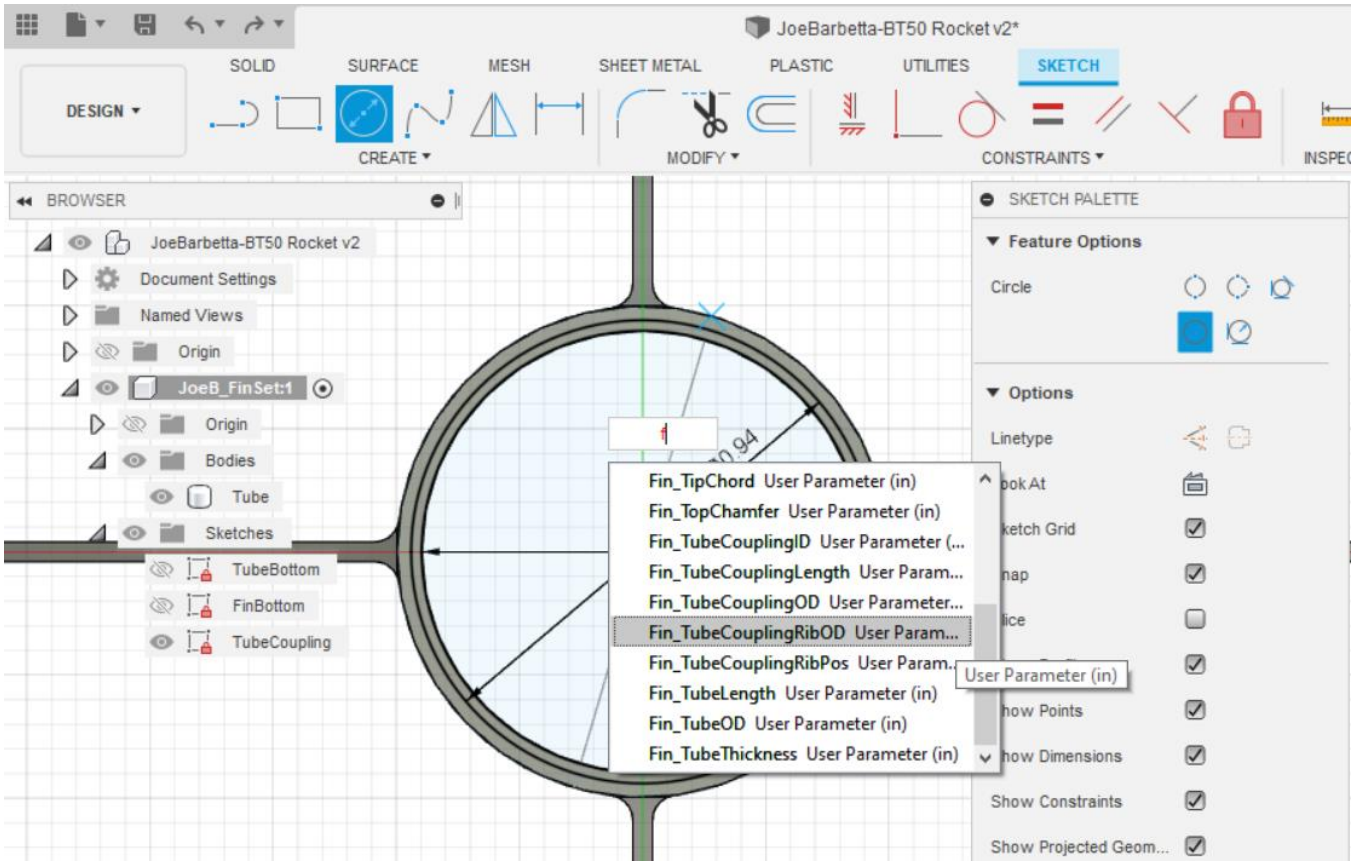
- select the **Fin_TubeCouplingID** parameter in the list and then press **Enter**. ID stands for *Inner Diameter*.



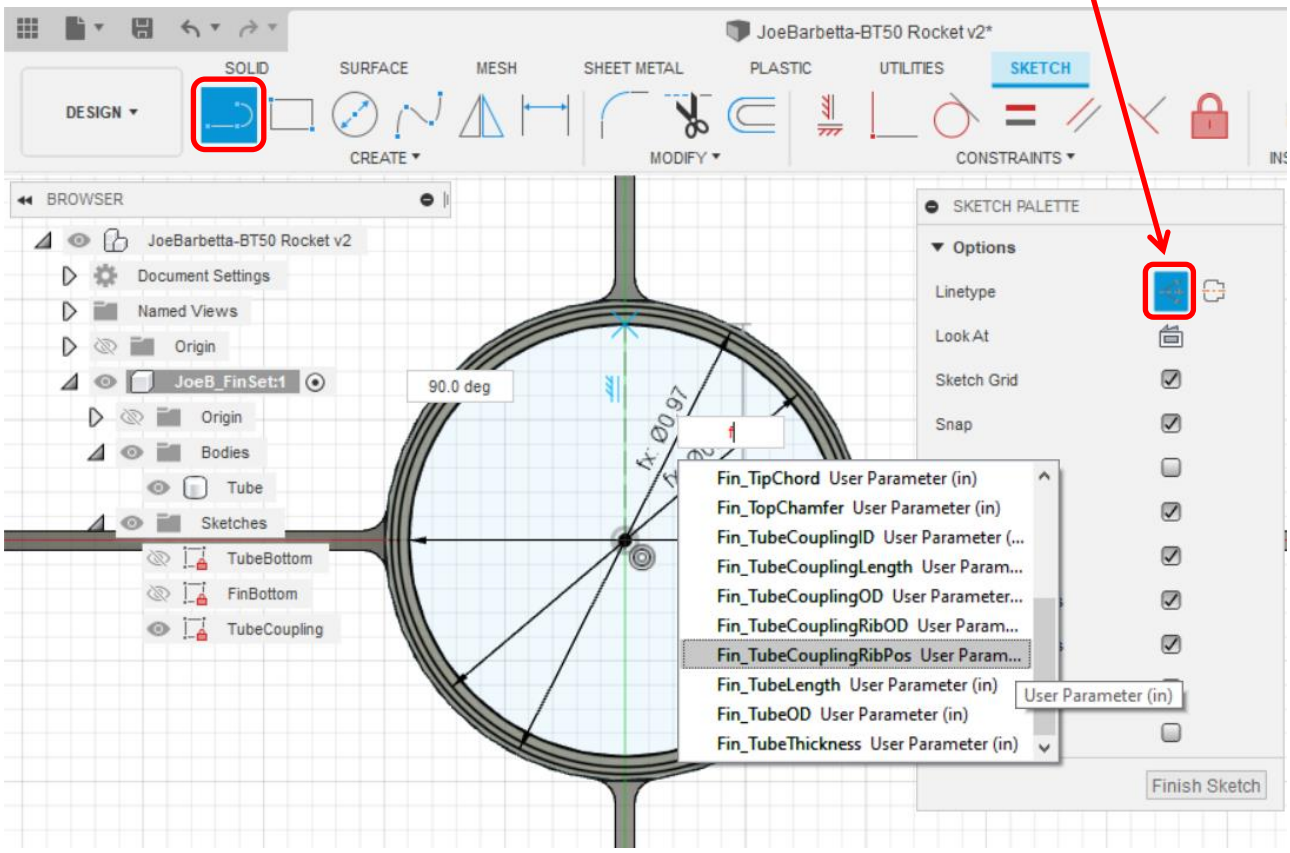
- create a 2nd circle from the center point and select the **Fin_TubeCouplingOD**



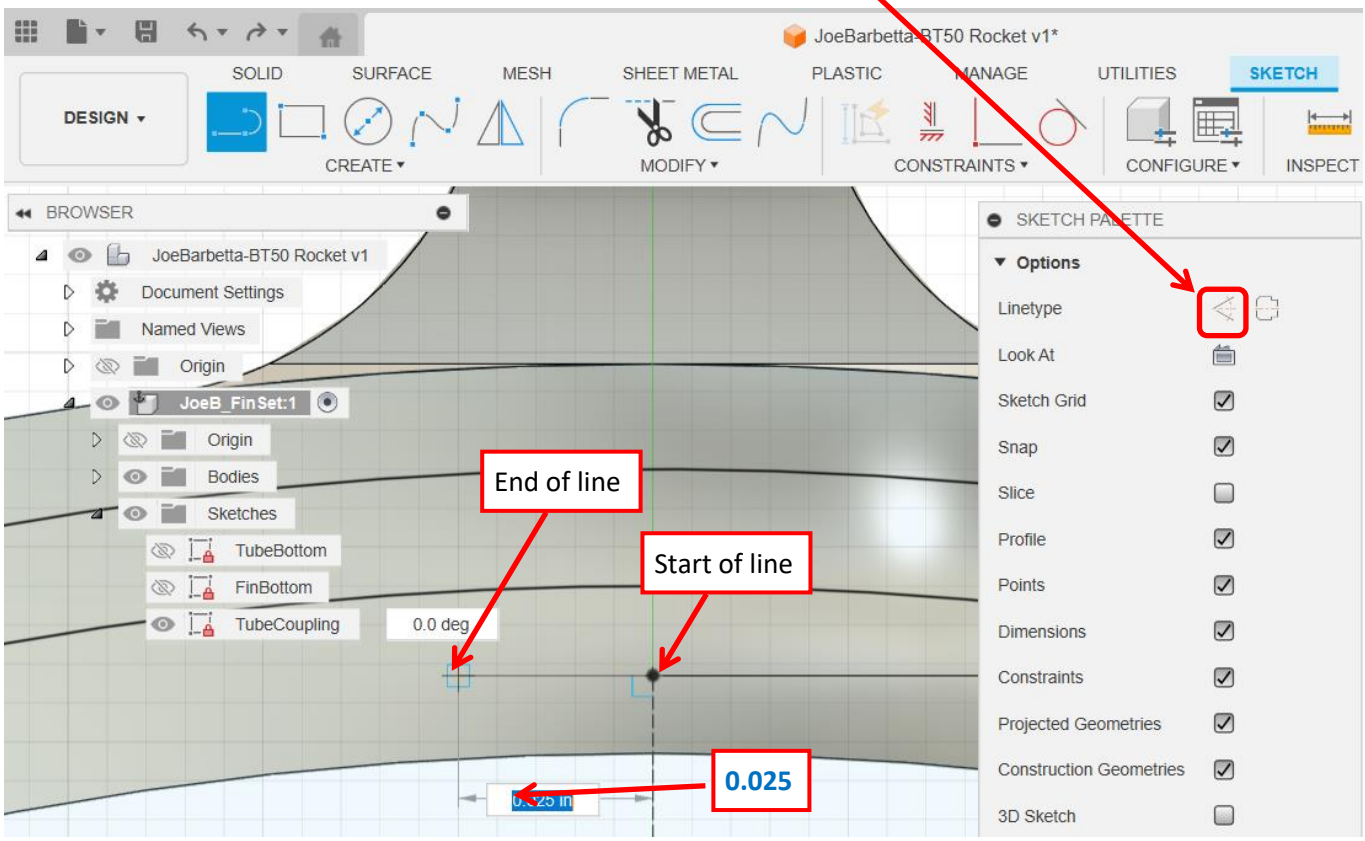
- create a 3rd circle from the center point and select the **Fin_TubeCouplingRibOD**



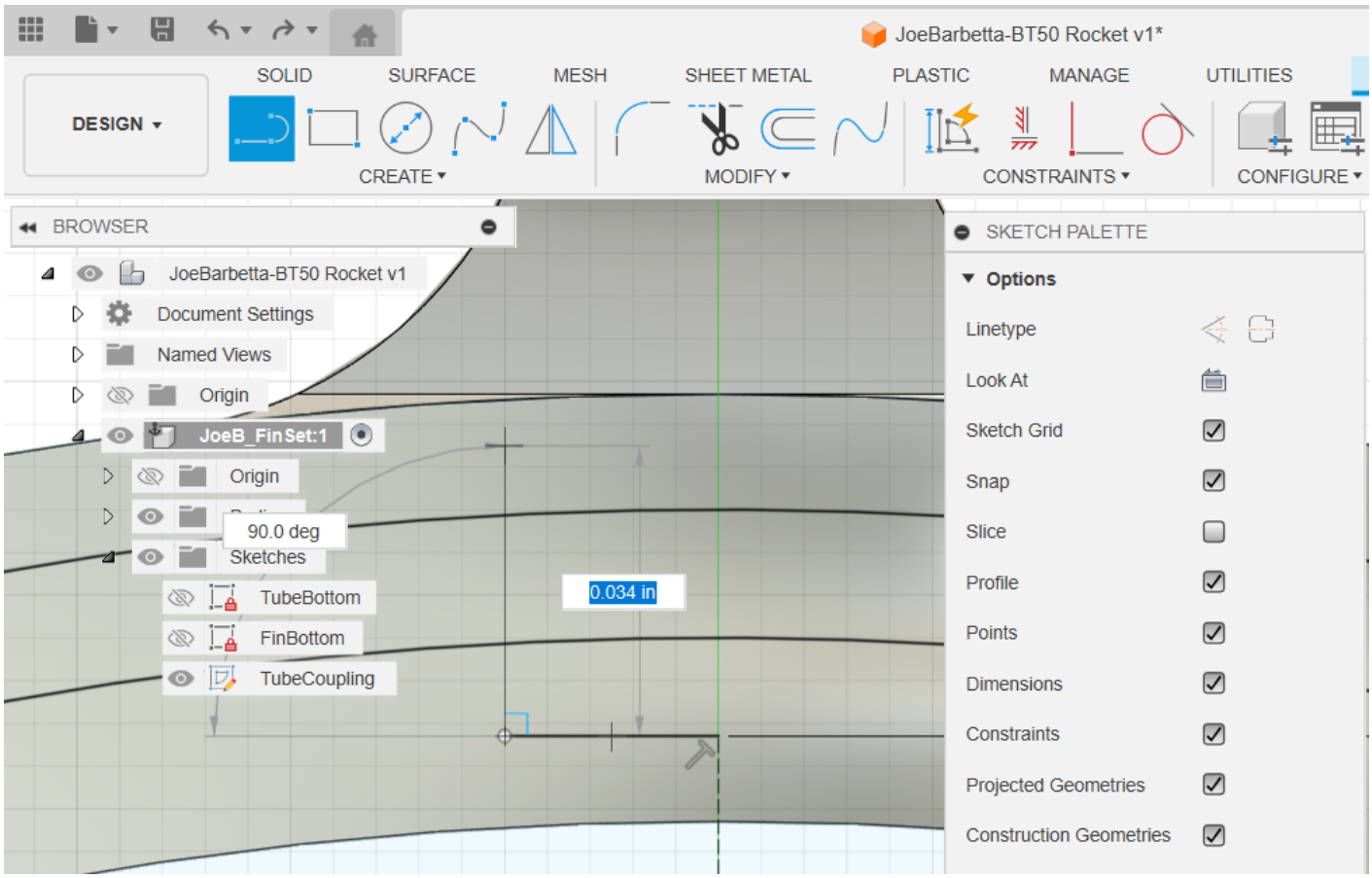
- click on the **Line** icon and on the Sketch Palette window click on the **1st icon next to Linetype** for a *Construction* line.
- draw a line starting at the center of the circles up towards the edge of the tube, type **f** and select **Fin_TubeCouplingRibPos.**



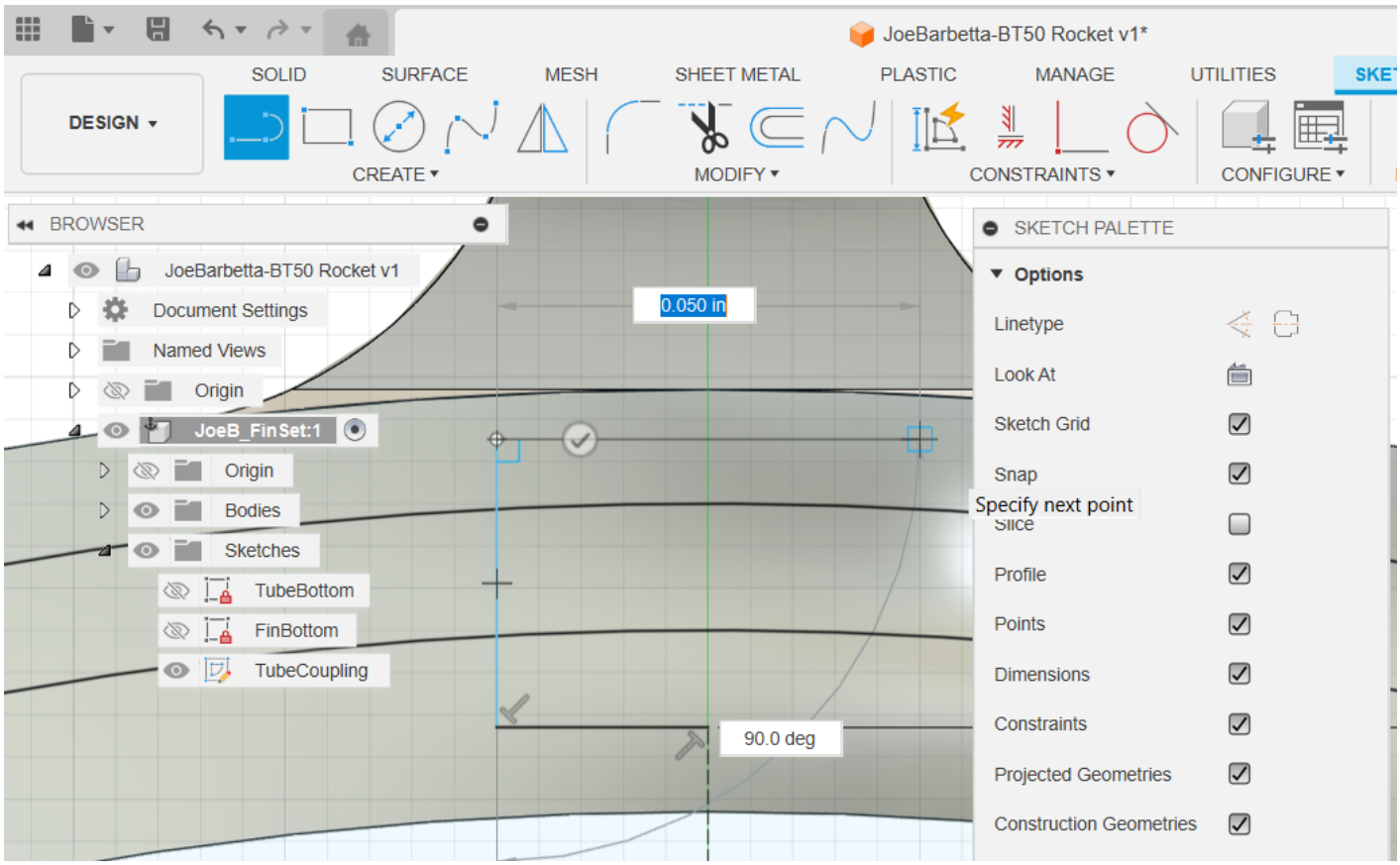
- zoom in to the top of the tube. This will now be like brain surgery under a microscope.
- click on the Construction Line icon again to **turn off the highlighting**.
- from the top of the Construction line draw a line to the left until the dimension box shows **0.025**. One can also type in 0.025.



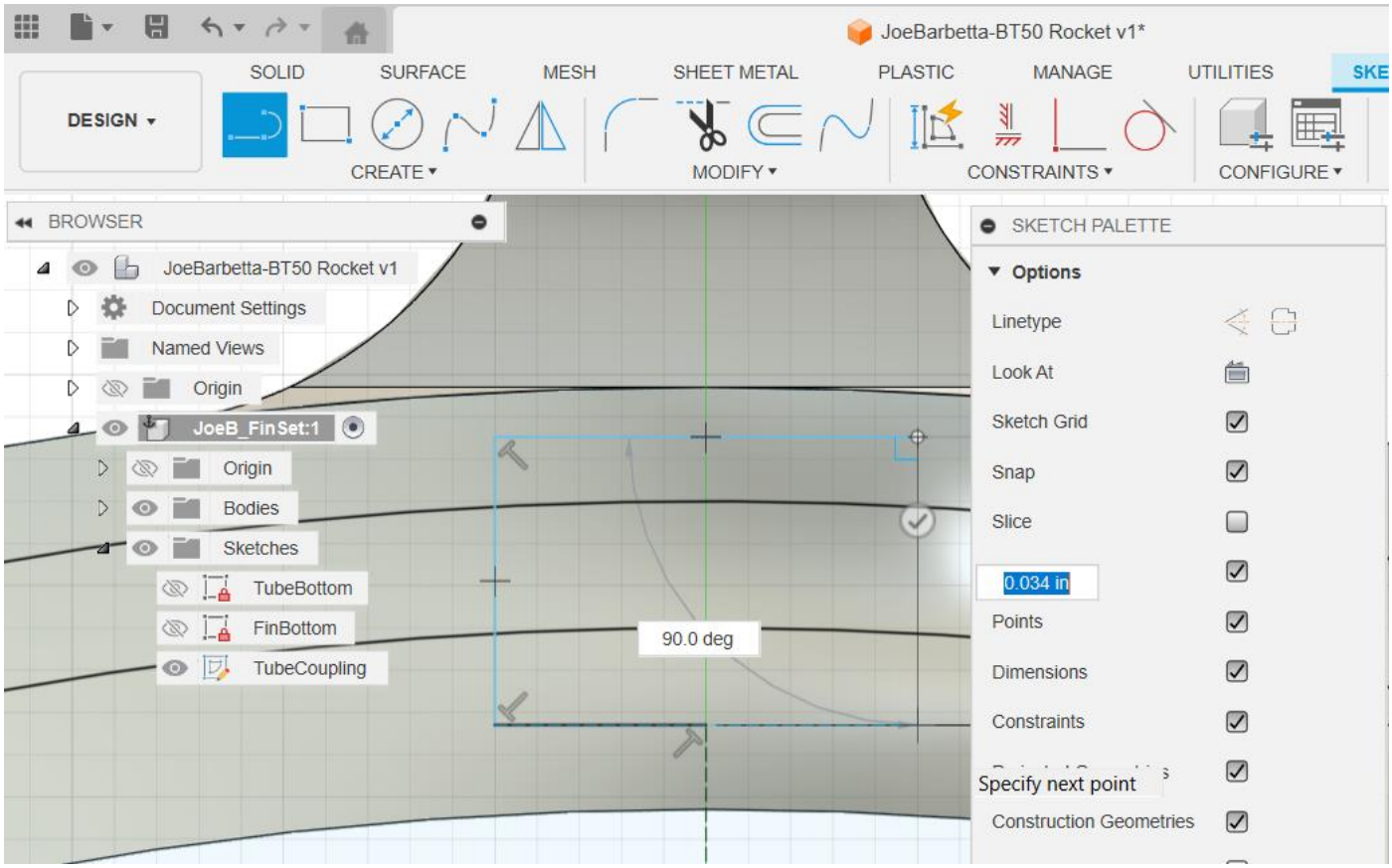
- draw a line from the last point up by **0.034**.



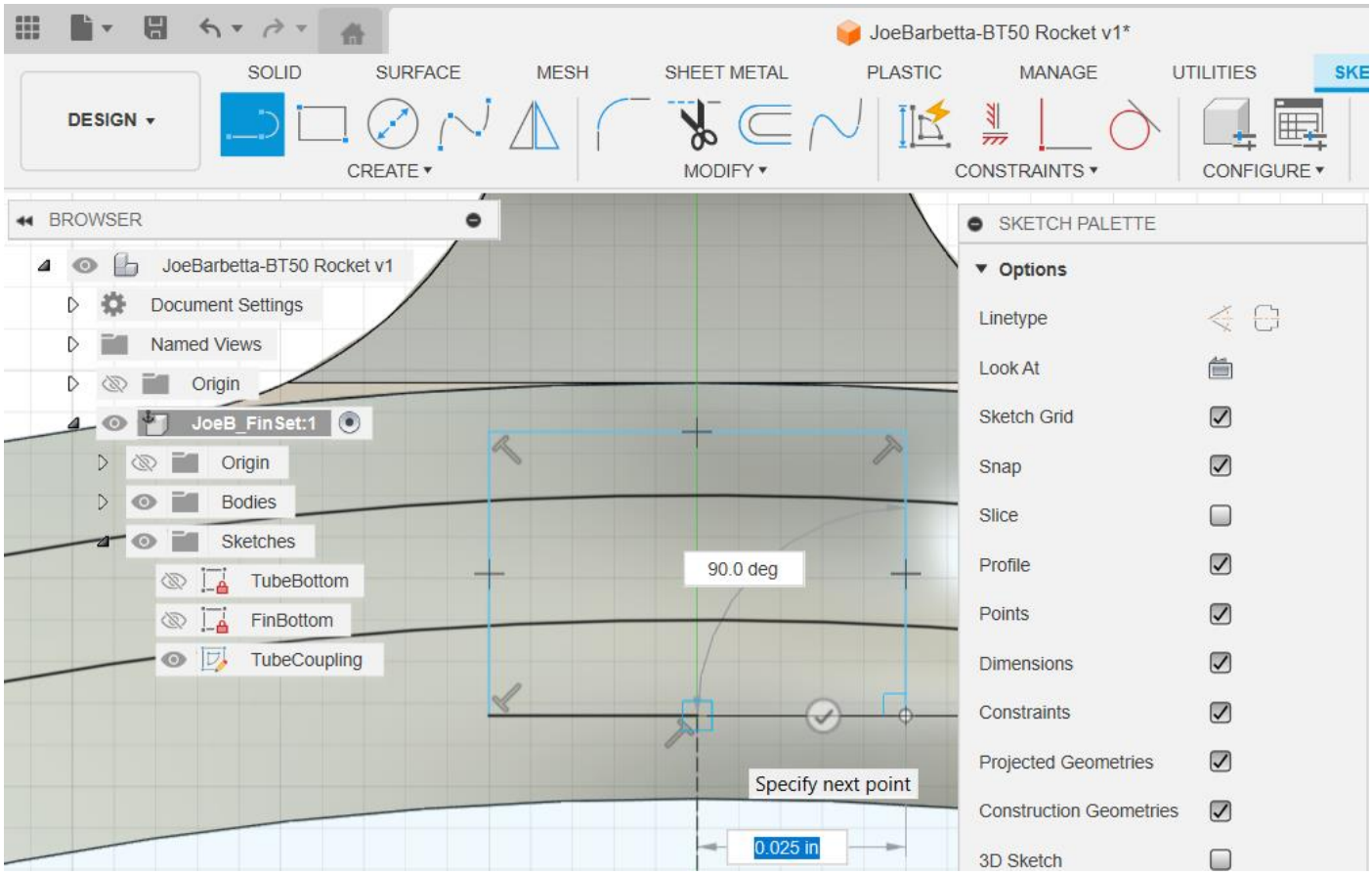
- draw a line from the last point to the right by **0.050**.



- draw a line from the last point to down by 0.044.

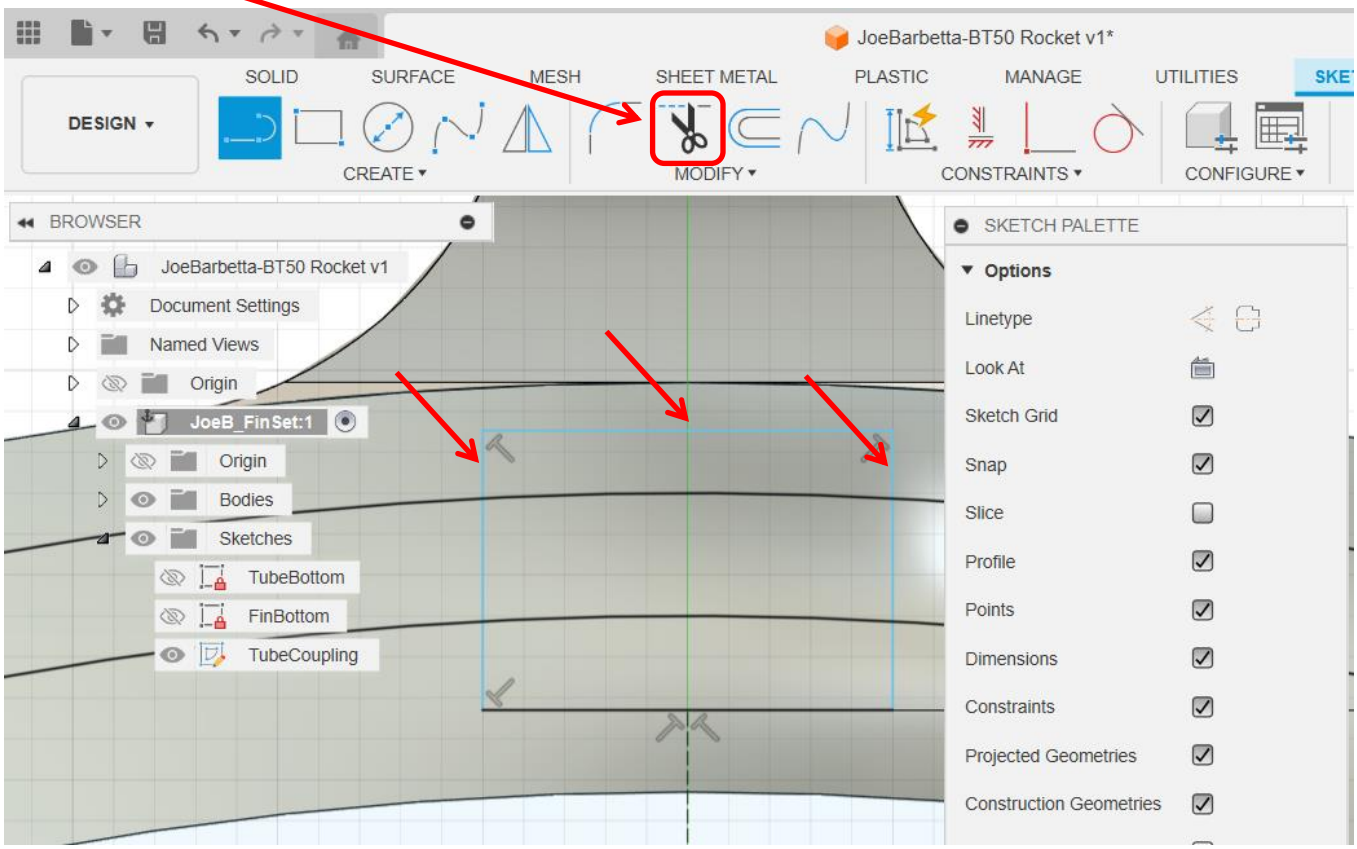


- draw a line from the last point to the left by 0.025. You should now have a closed rectangle.

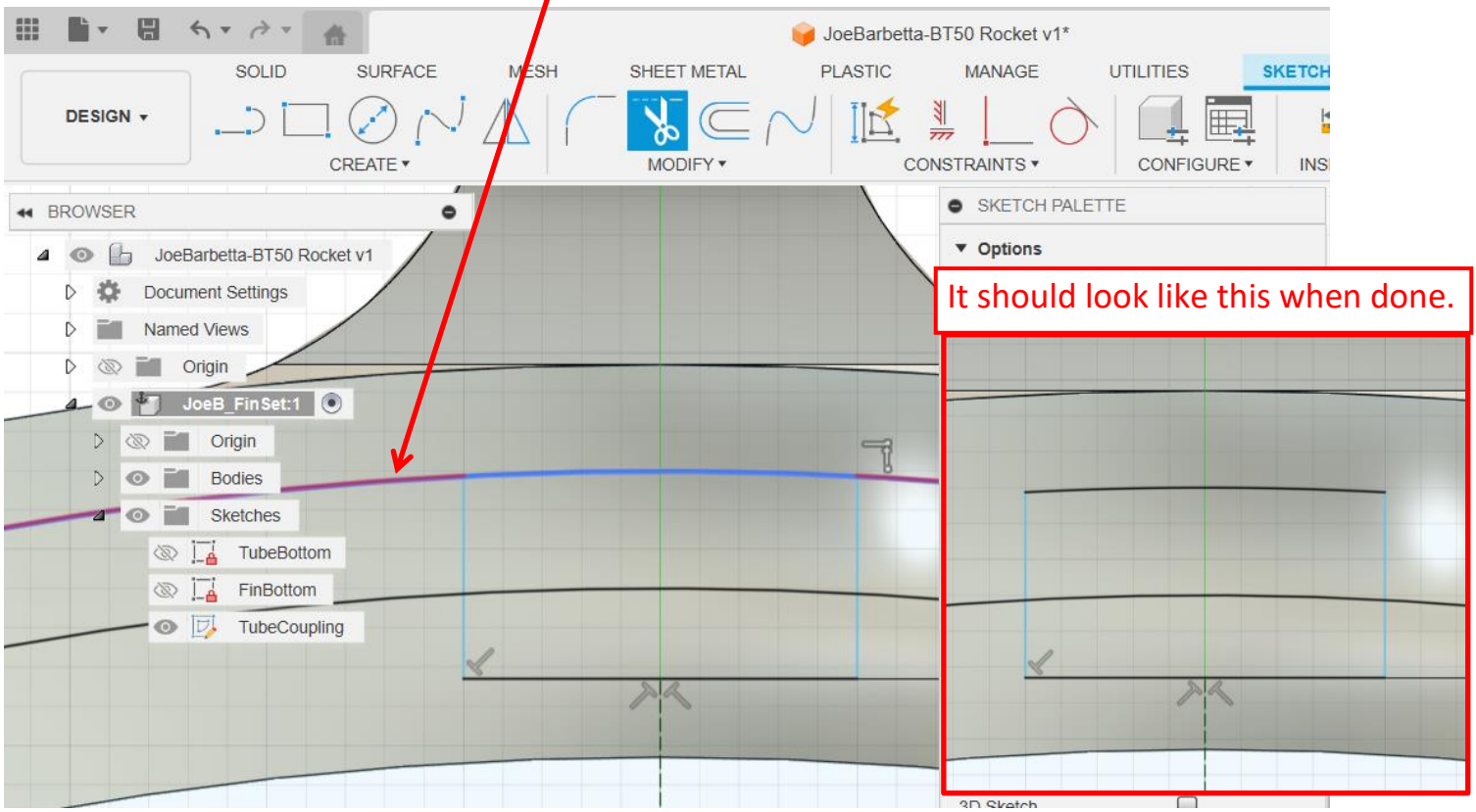


Using the Trim Tool

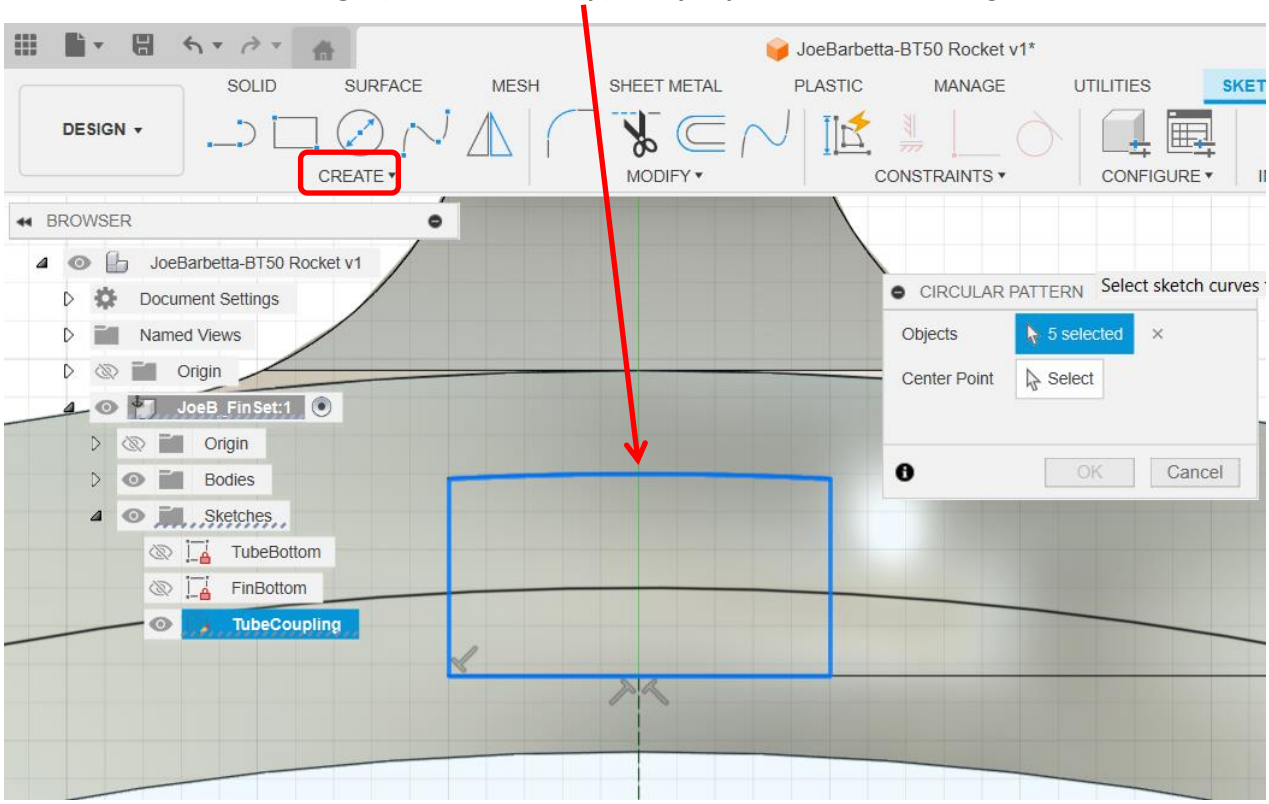
- select the **Trim** tool. Click on the **top 3 segments** (indicated with arrows below) to remove them.



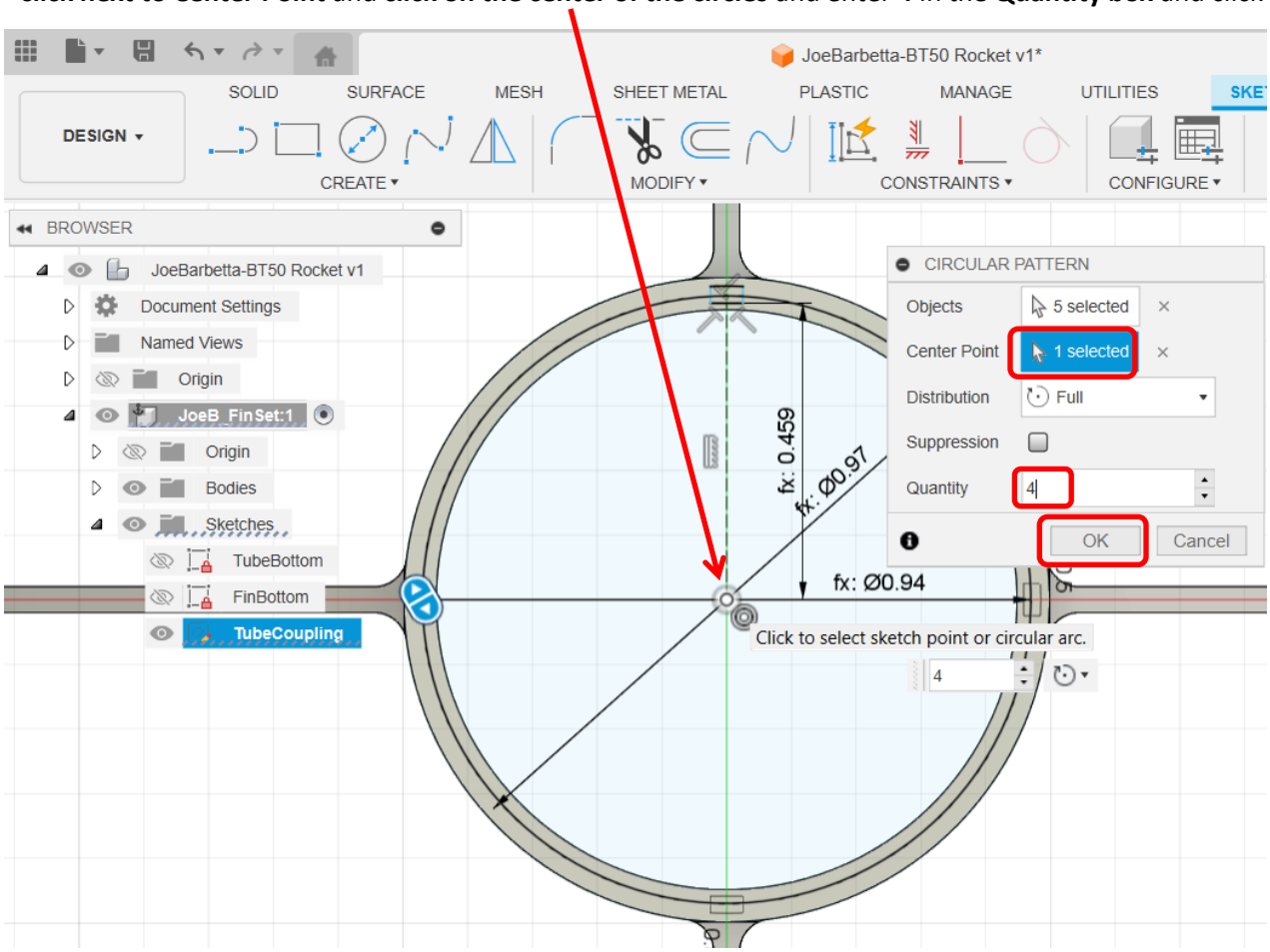
- Click once on the **outer circle** to the left of rectangle that you just created, to remove the violet portion of the circle. This will leave a rectangle with a slightly curved top.



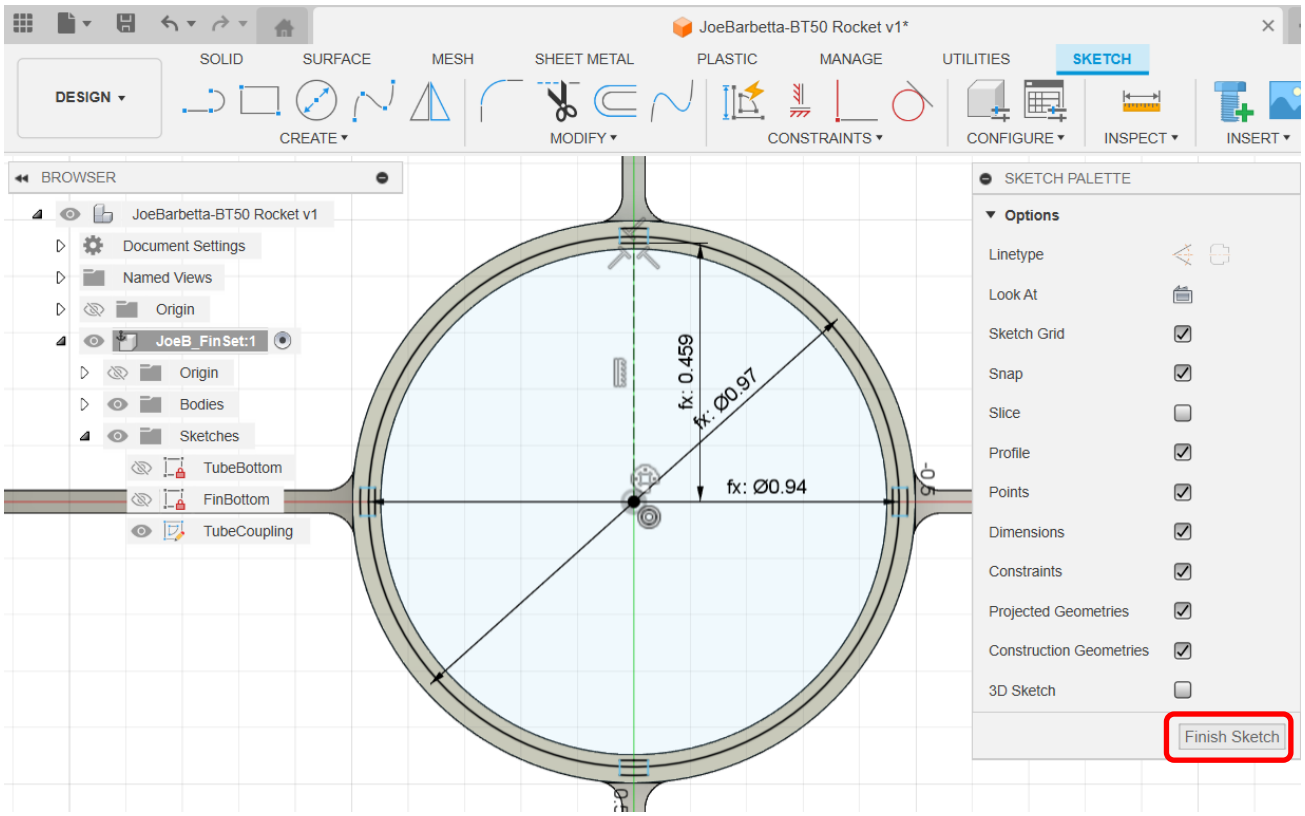
- from the **CREATE** menu select **Circular Pattern**
- **double-click on the rectangle (with rounded top) that you just created**. Each edge should turn blue.



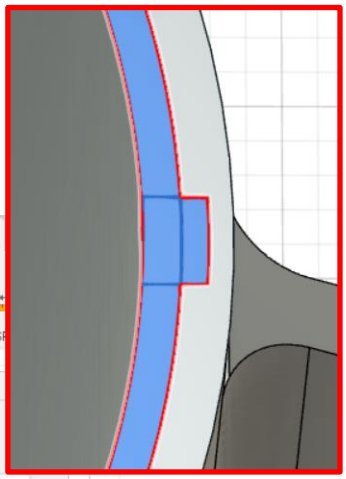
- zoom out as shown below. As stated in earlier sections, **don't worry if the light *Dimension* lines look different**
- **click next to Center Point** and **click on the center of the circles** and enter **4** in the **Quantity box** and click **OK**.

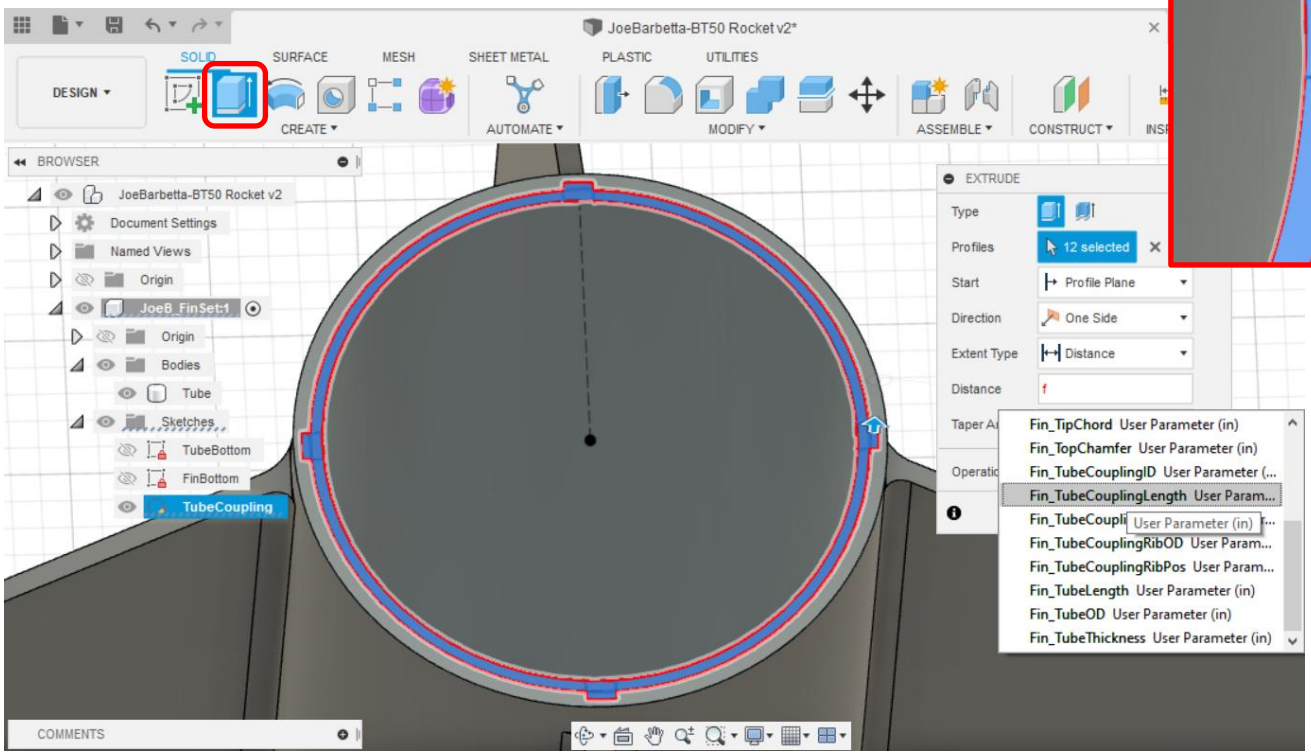


- click **Finish Sketch**.



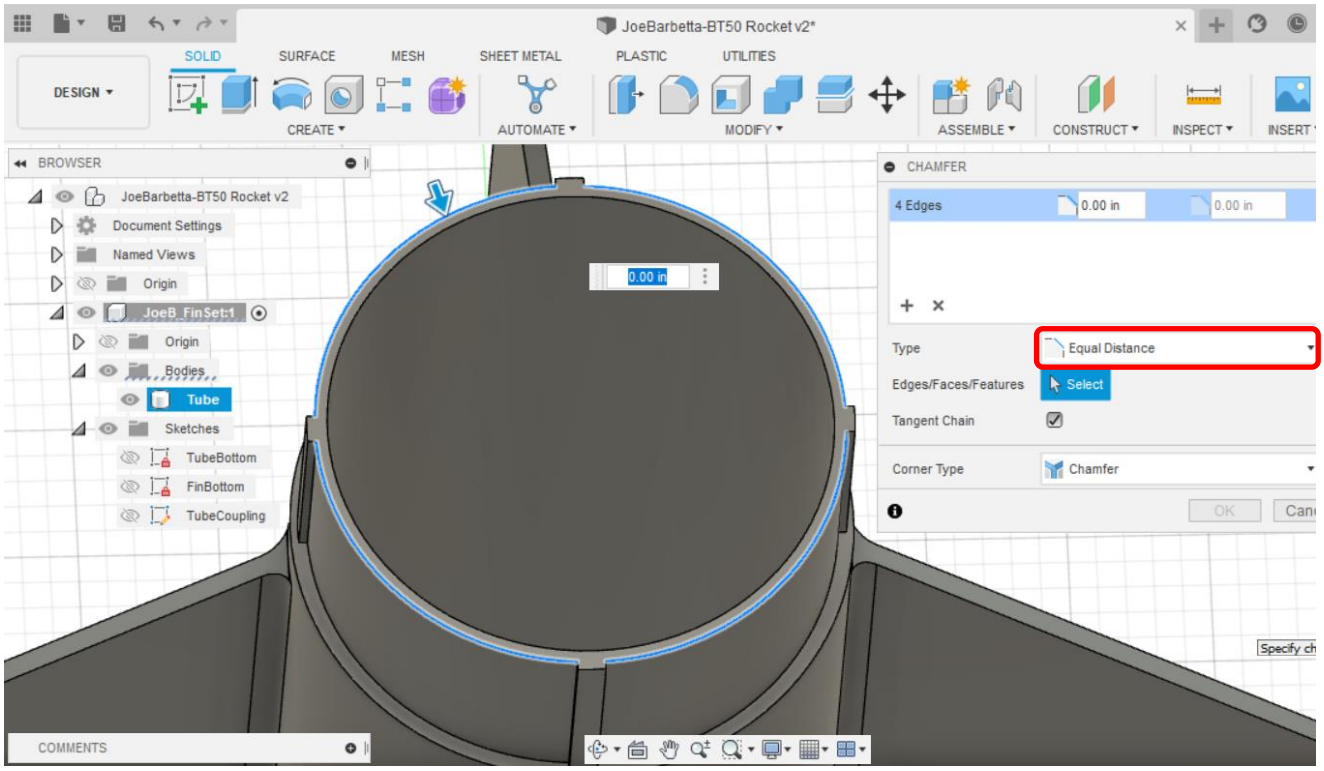
- click the Home icon next to the View Code and zoom in as shown below.
- select the **Extrude** tool
- click on the ring and rib profiles so their interiors turn blue. As shown in the magnified view on the right, each rib has two sections. It is important Not to select the outer ring section.
- type **f** and select **Fin_TubeCouplingLength** and click **OK**.



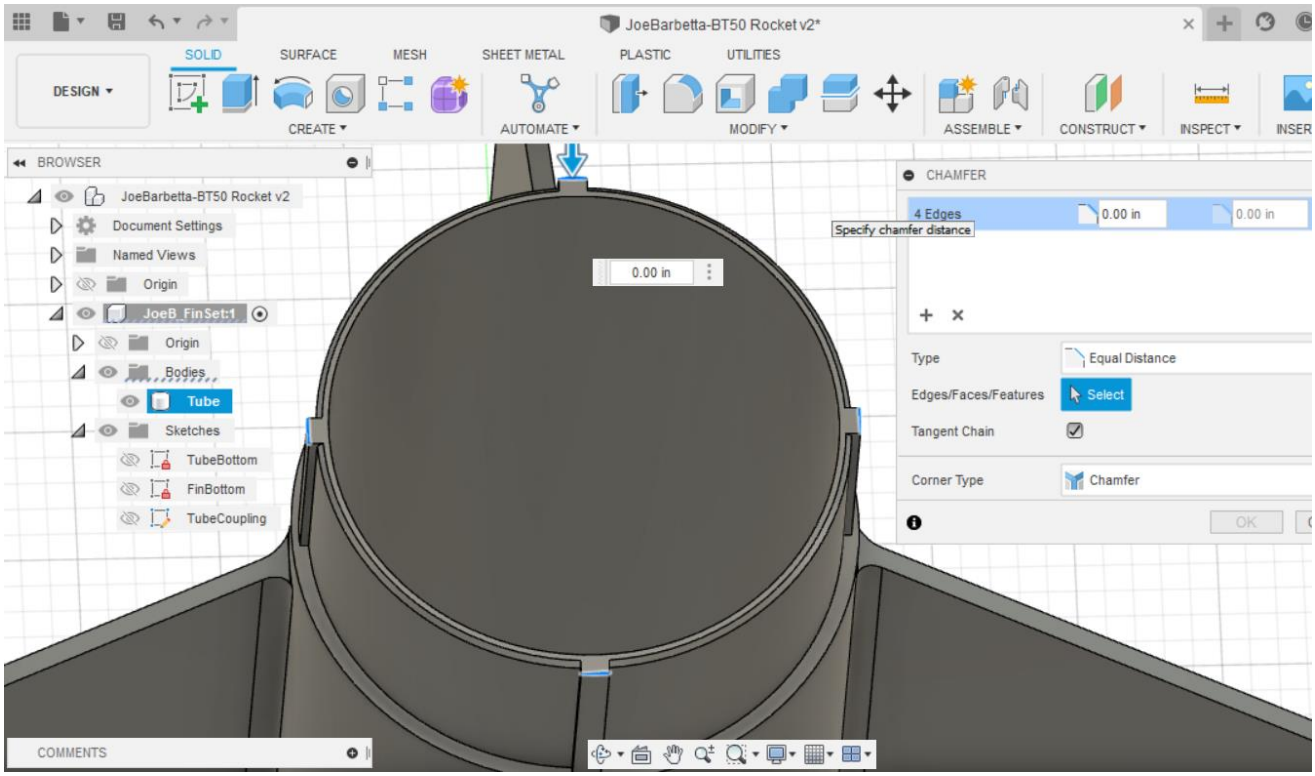


- within the **MODIFY** menu select **Chamfer** and ensure that **Type** is set to **Equal Distance**.
- click on the 4 edges between the ribs, type **0.01** and click **OK**.

If a Chamfer or Fillet operation ever causes an error, try a smaller value.



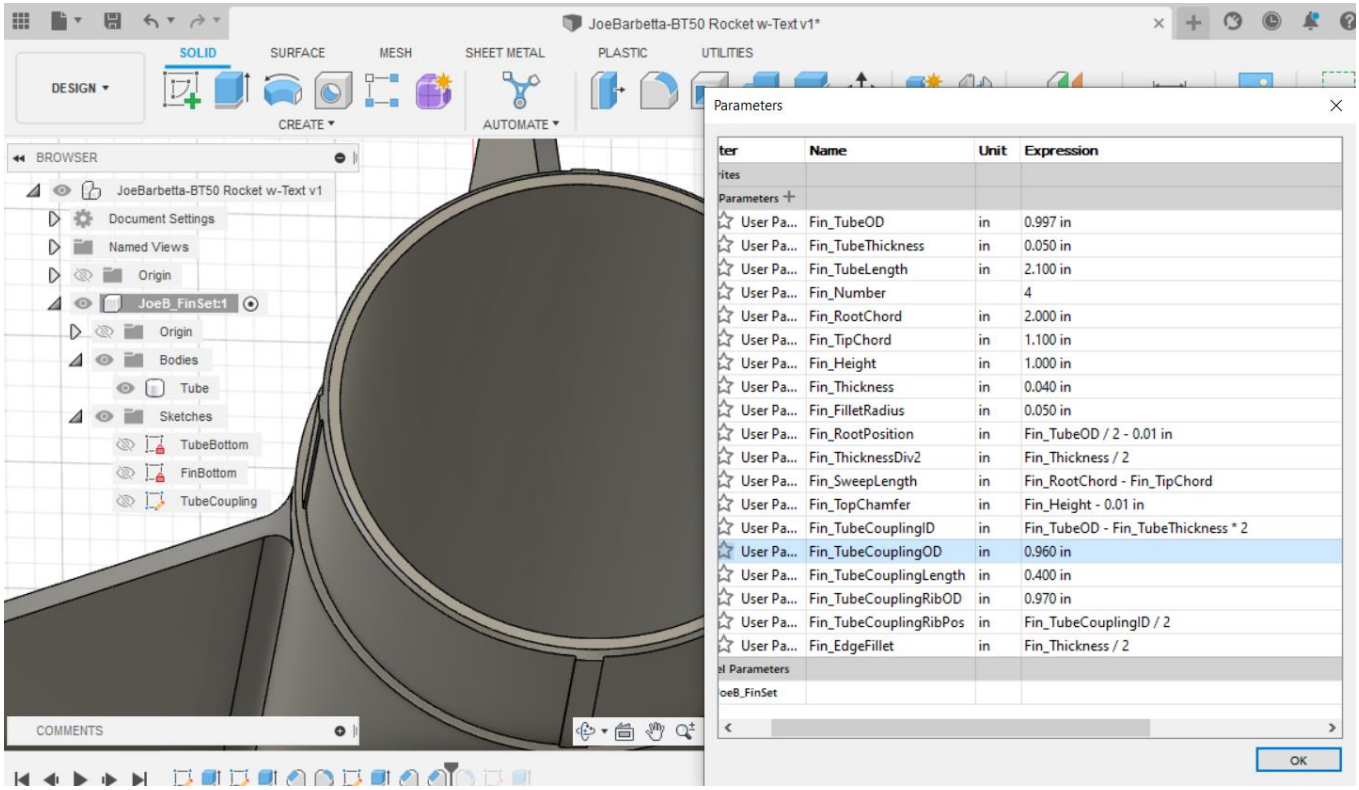
- use the **Chamfer** tool again, but this time click on the edges of the ribs and use a value of **0.015**.



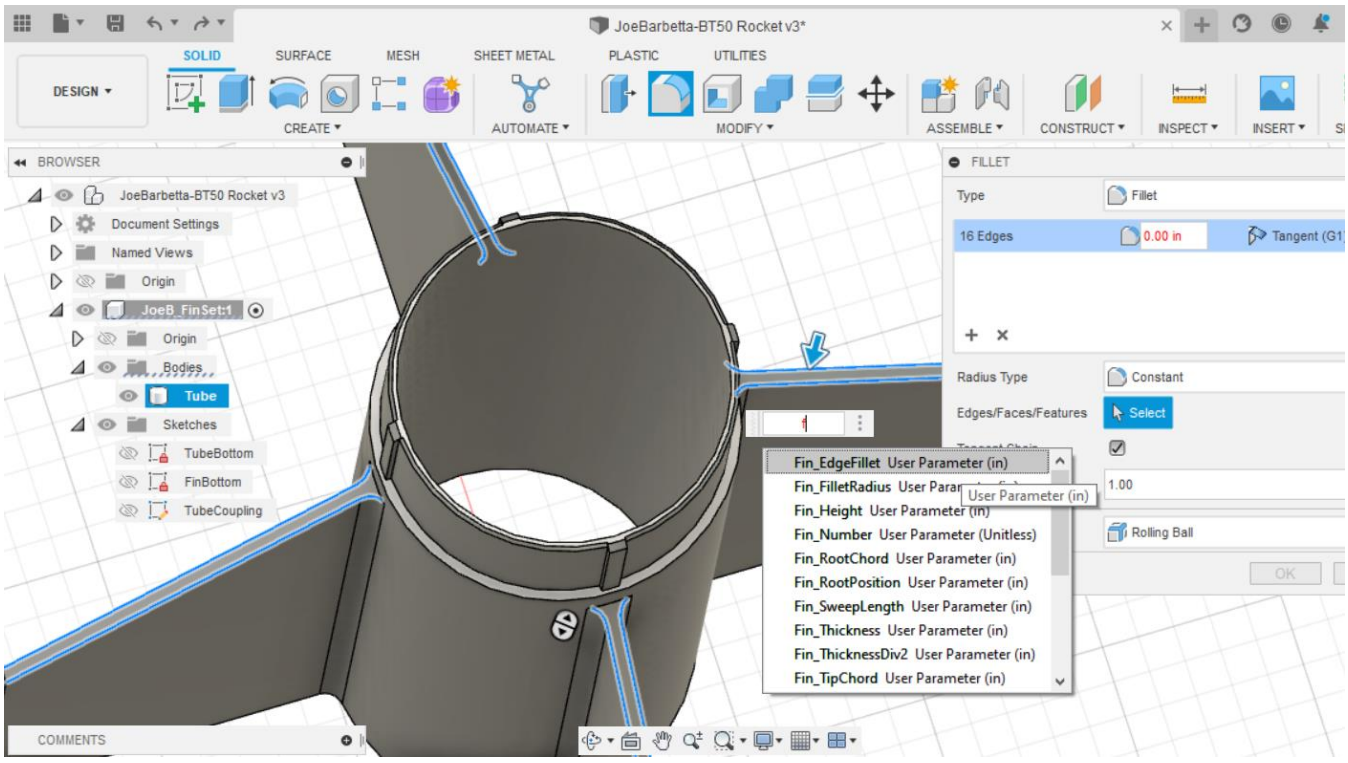
The coupling section has a thin wall thickness that should be made thicker. This is an opportunity to change a parameter.

- near the bottom of the **MODIFY** menu select **Change Parameters**. Change the **Fin_TubeCouplingOD** to **0.960**

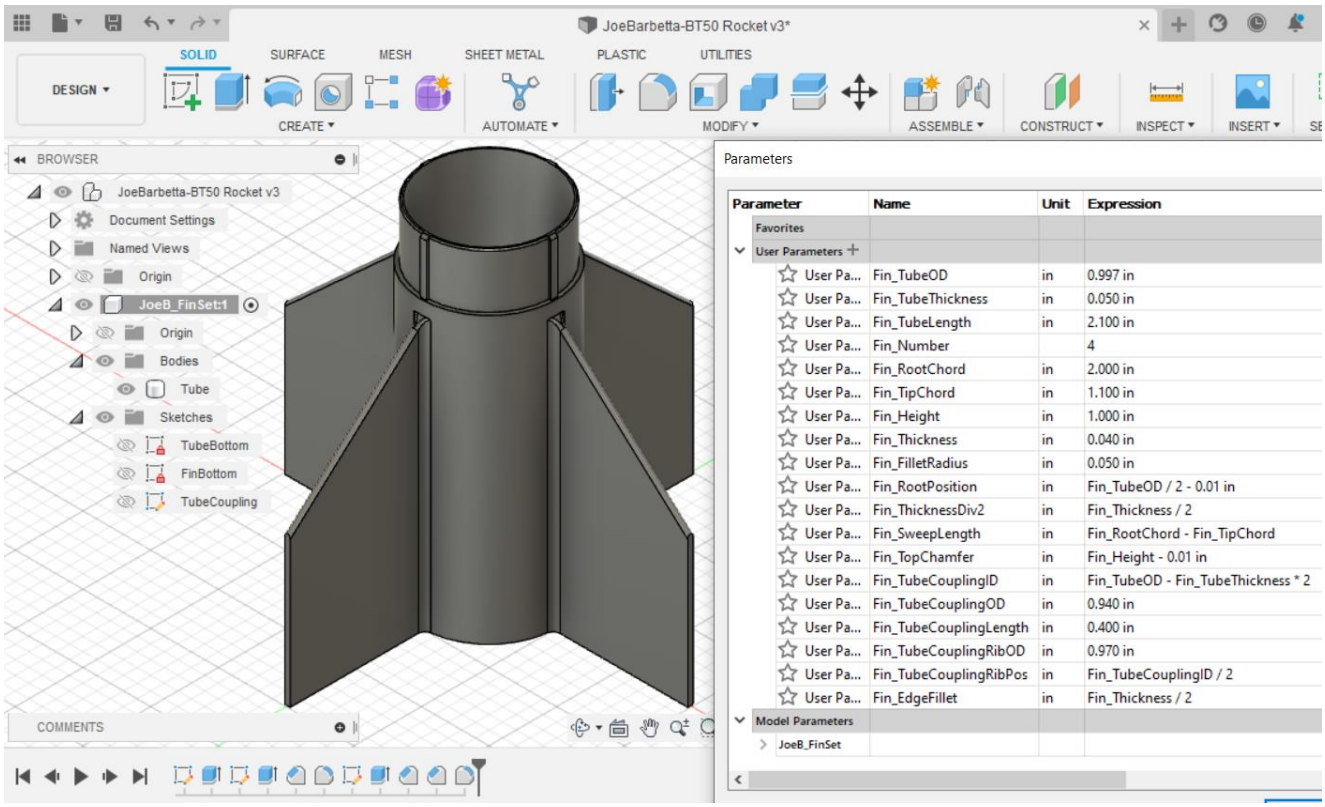
Note that this increased the wall thickness of the coupling section.



- zoom out as shown below.
- in the **MODIFY** menu select **Fillet** and click on the two top edges of each fin
- type **f** and select **Fin_EdgeFillet** and click **OK**



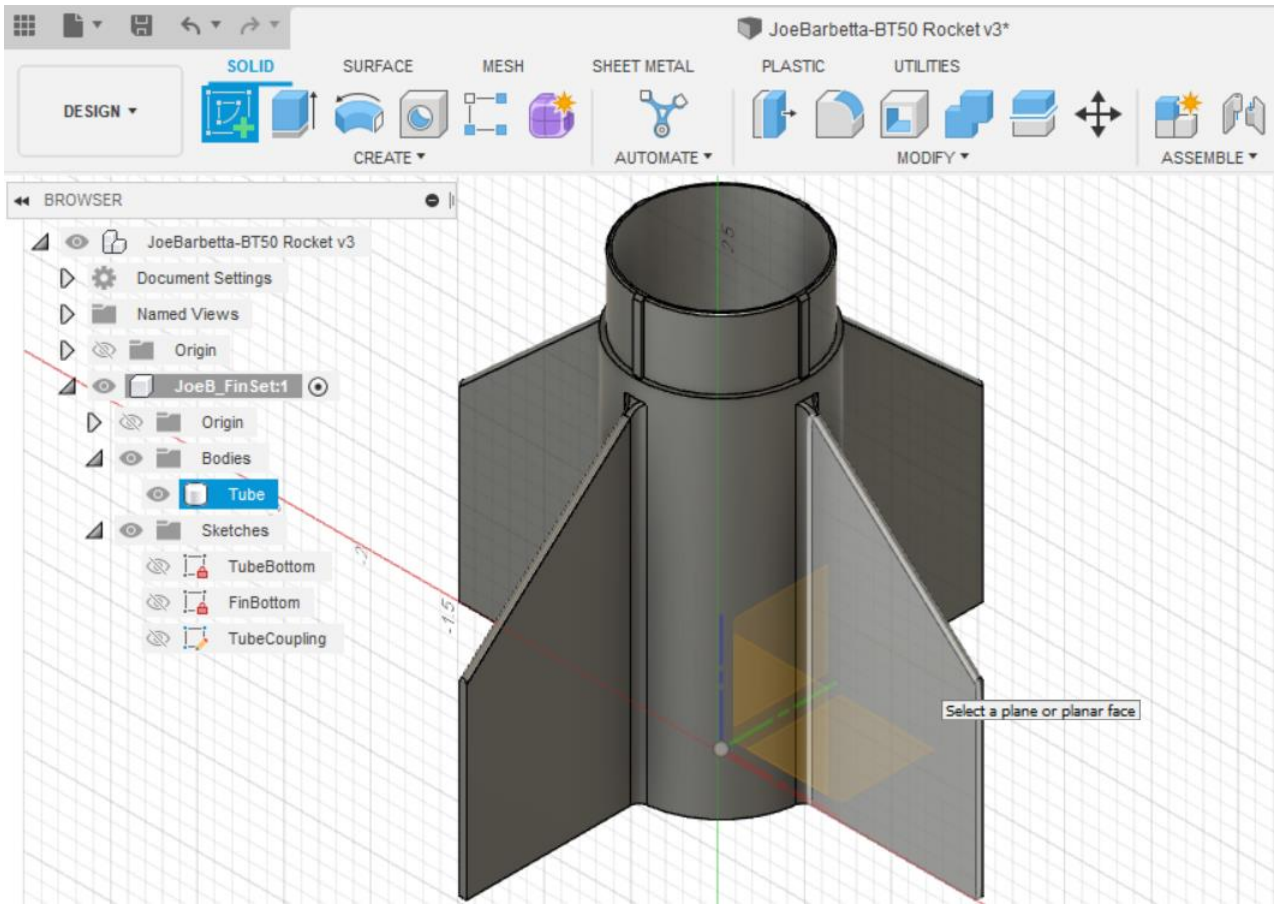
This is our finished fin set with the Parameters window open. Next, we will add text.



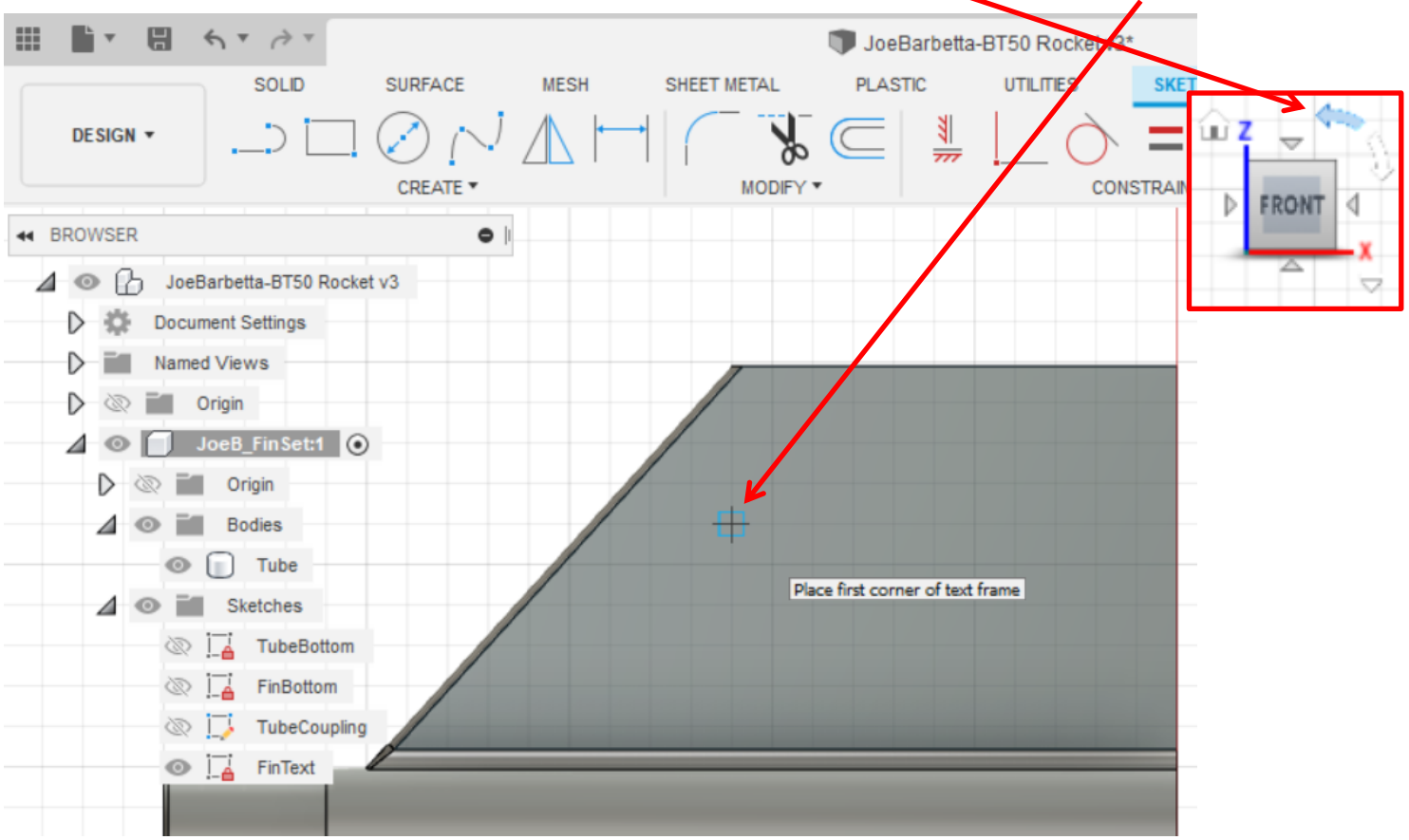
Adding Text to a Flat Surface

It's a good idea to add text to your rockets to easily identify their specifications. For example, to indicate if the warhead is conventional or nuclear and its yield in pounds or kilotons.

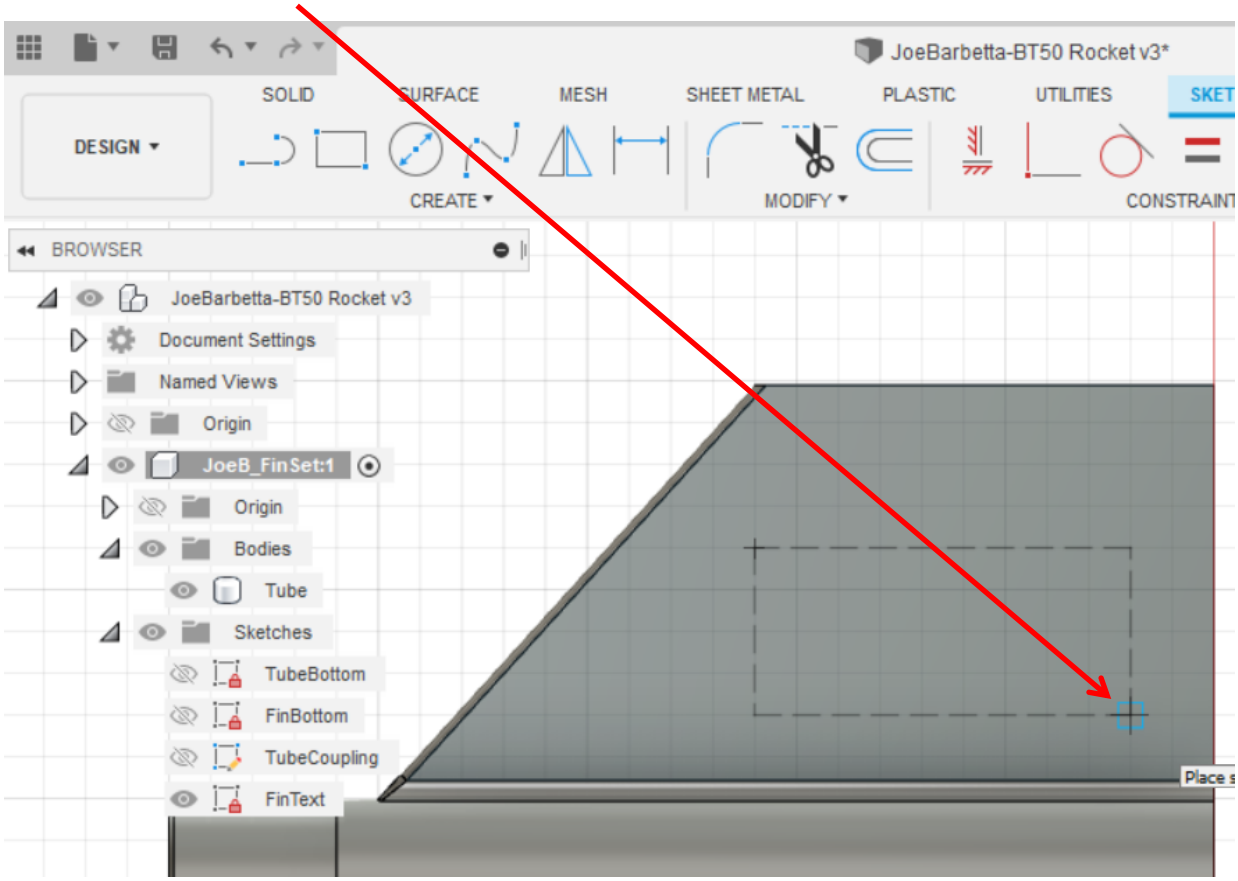
- select **Create Sketch** and click on the fin surface, which should turn light gray



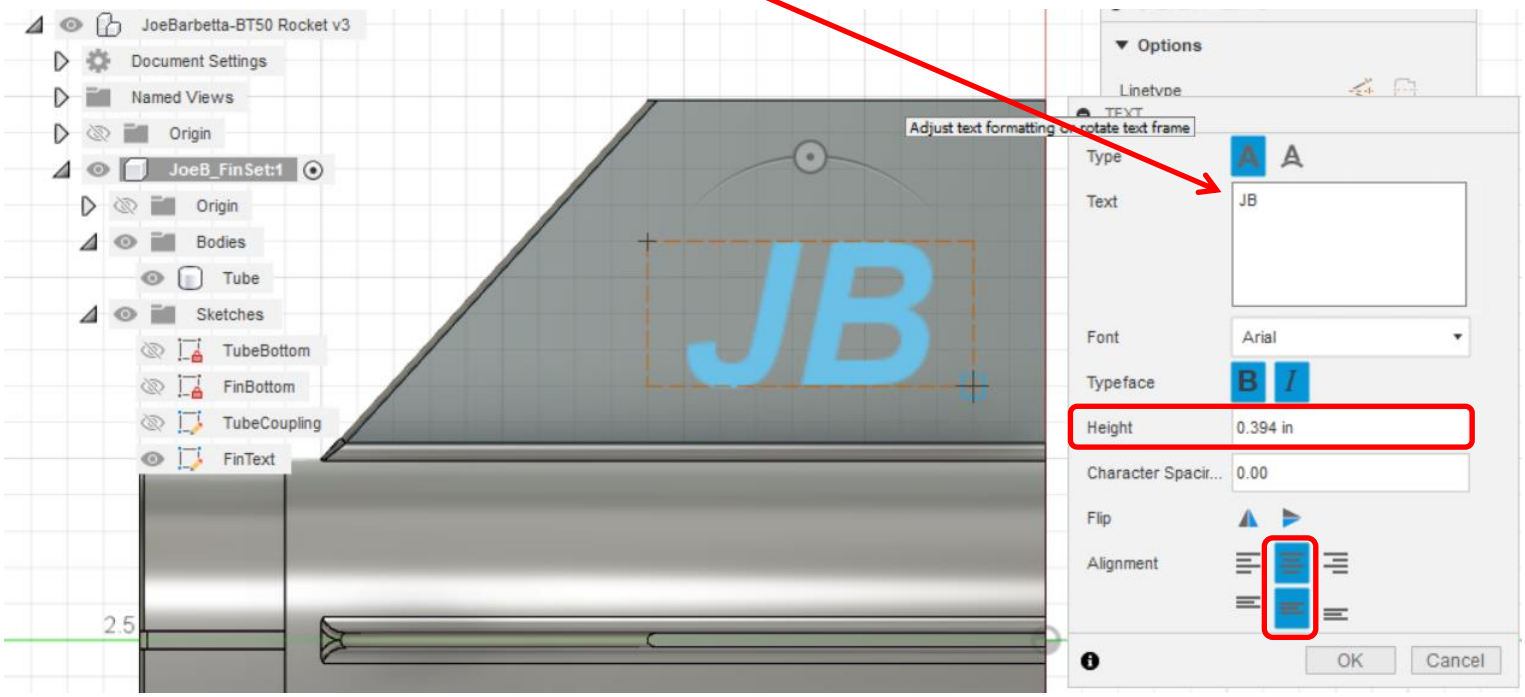
- zoom in to the fin. When hovering over the View Cube two curved arrows will show. Clicking on one will rotate the view.
- in the **CREATE** menu select **Text** and start defining a text region rectangle by clicking where the top left corner should be.



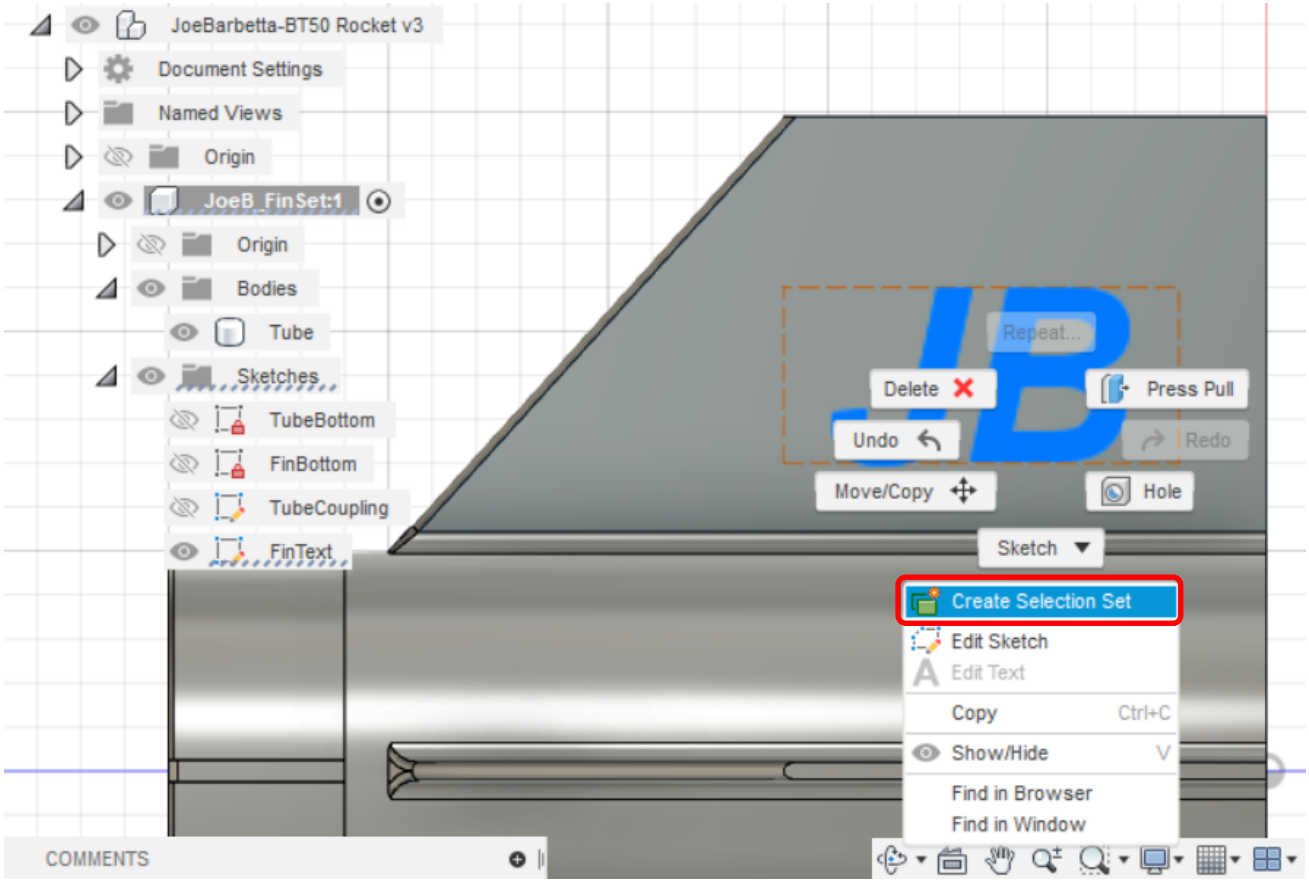
- click where the bottom right corner should be.



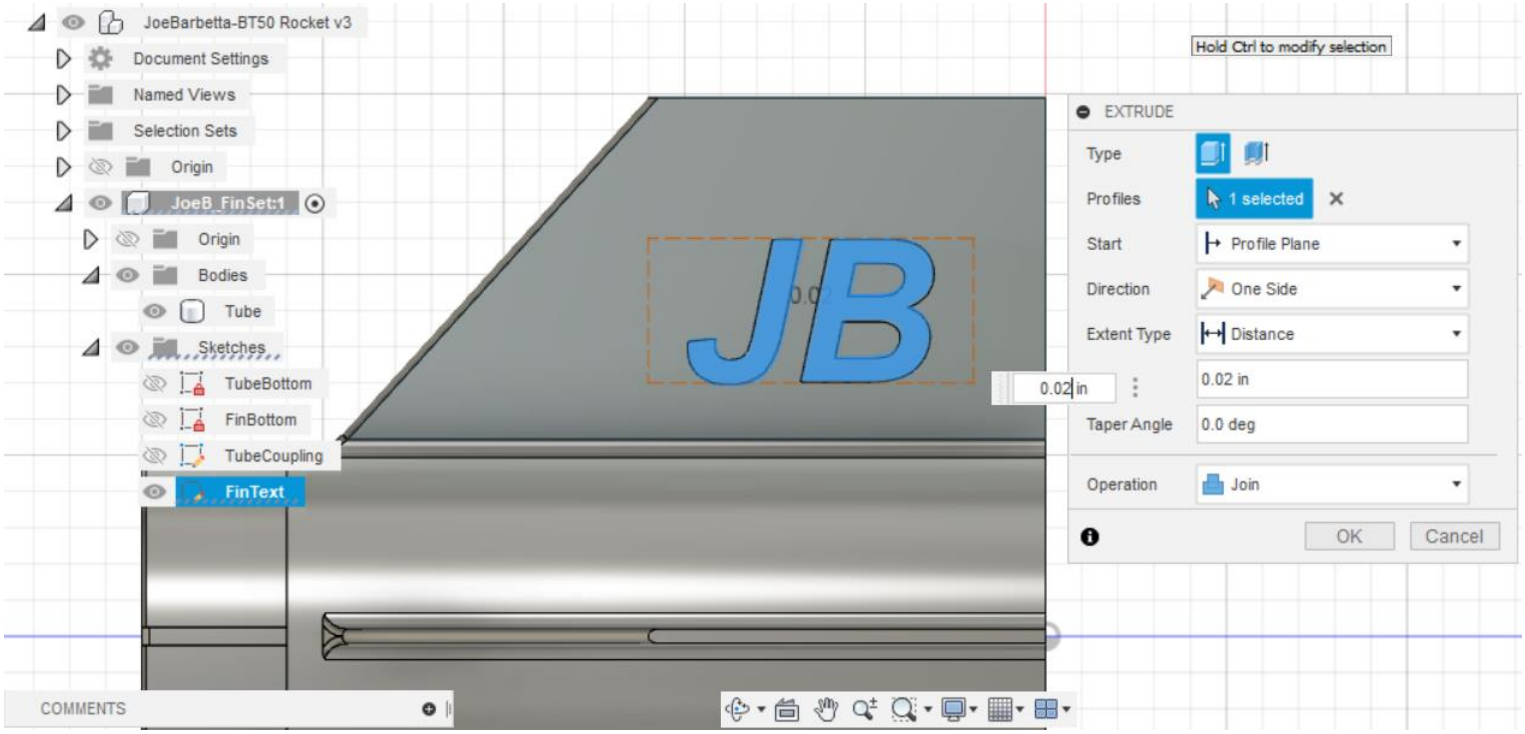
- enter the desired text in **Text**. If the text is too large or too small change the Height until the desired size is attained.
- one can also change the Font and make the **Typeface bold or italic**
- one can also click on **Alignment** options to center the text horizontally and vertically
- the text can also be mirrored by using a **Flip** option
- click on **OK** and then **Finish Sketch**



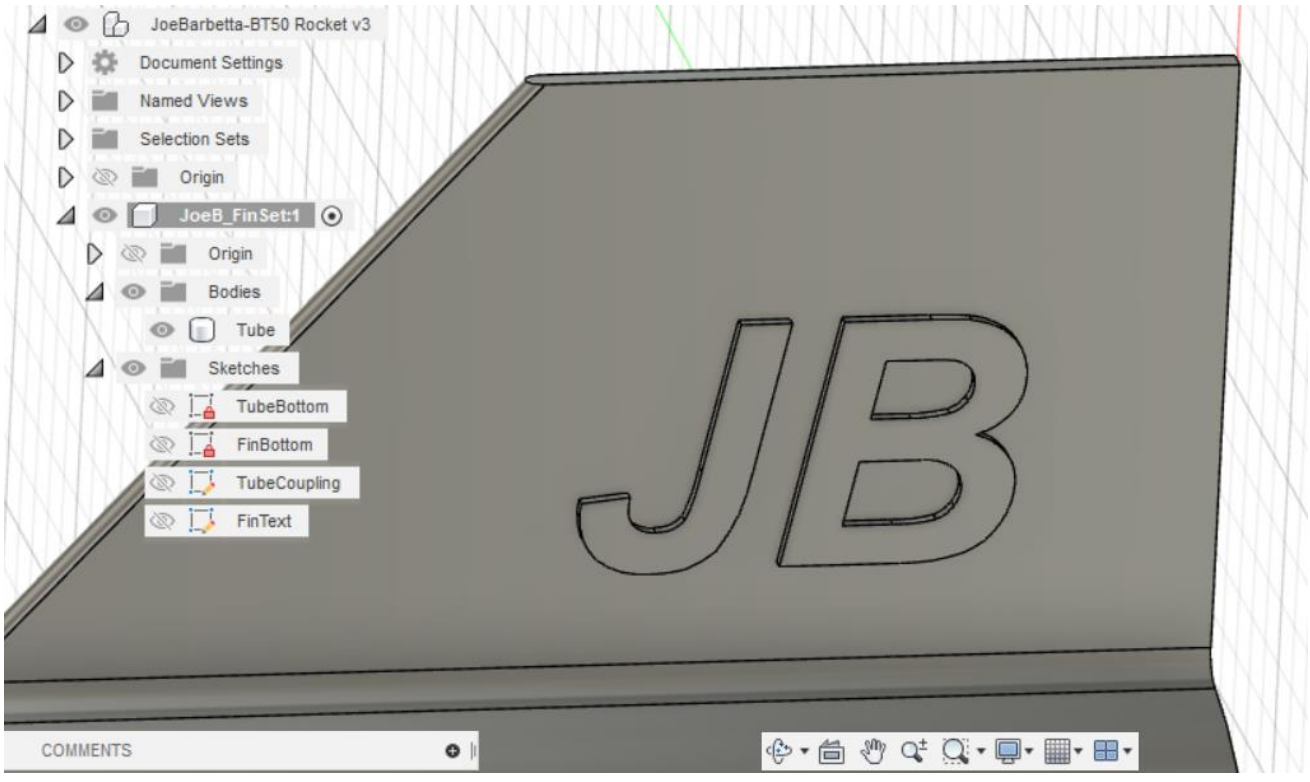
- right-click on the text and select **Create Selection Set**



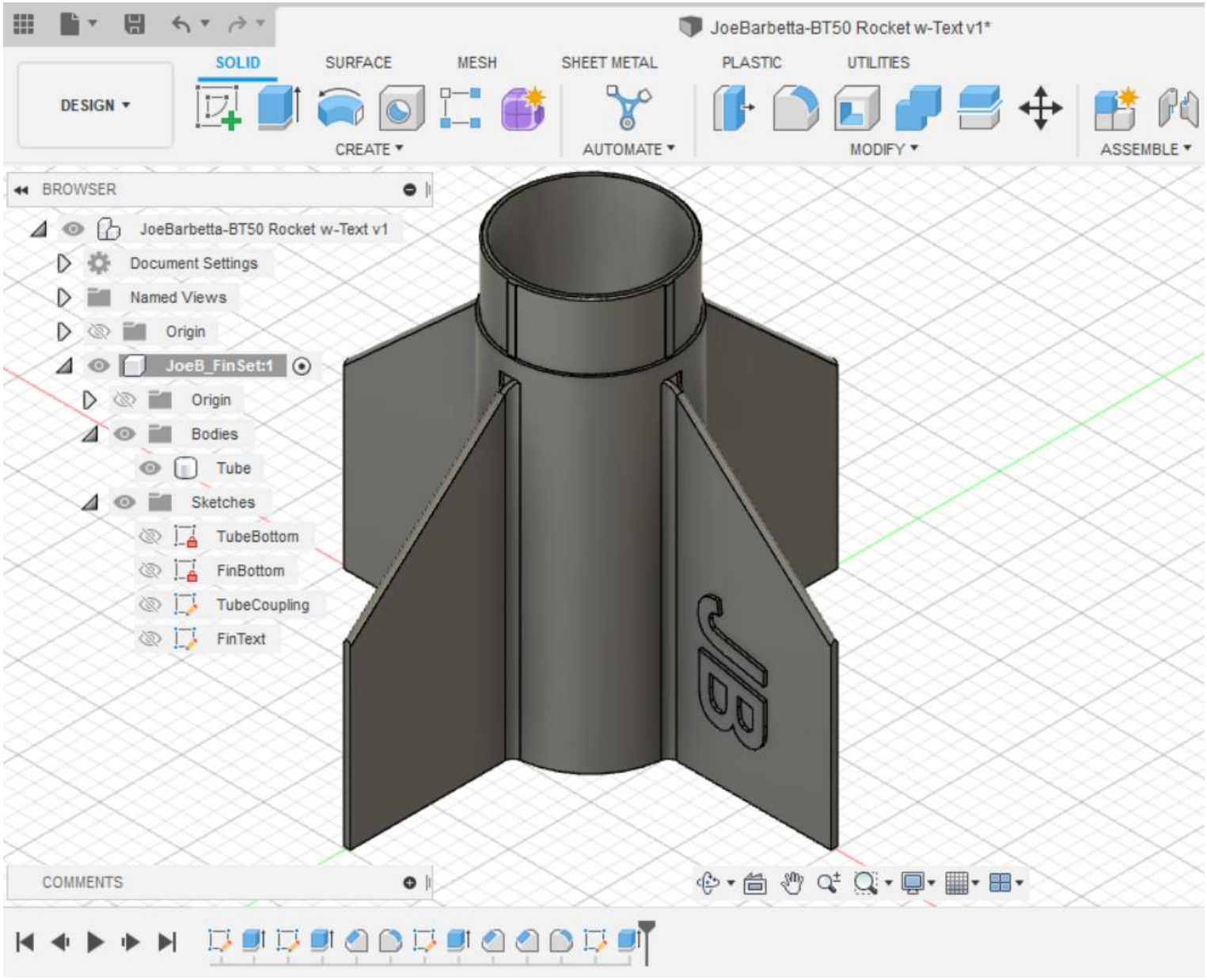
- select the **Extrude** tool, type **0.02**, and click **OK**



Rotating the view should show that the text is extruded.

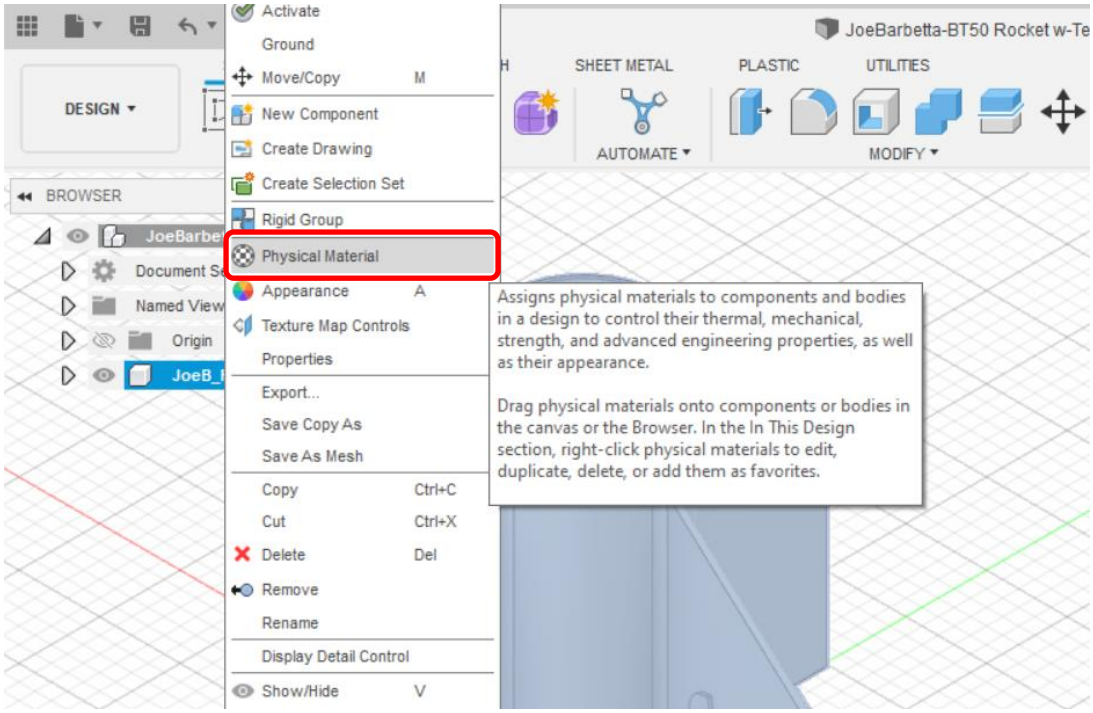


Here is your finished Fin Set.

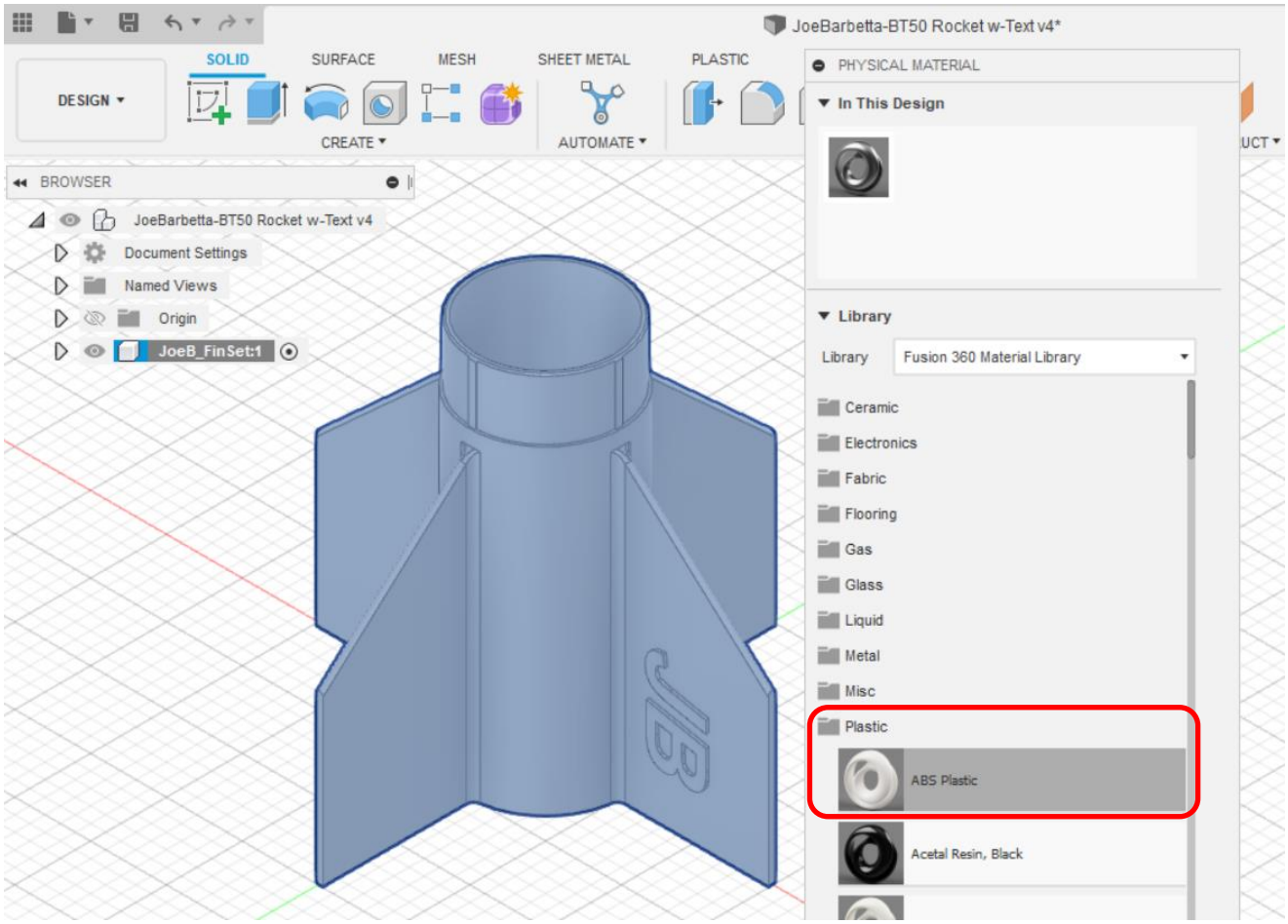


Setting Materials and Colors

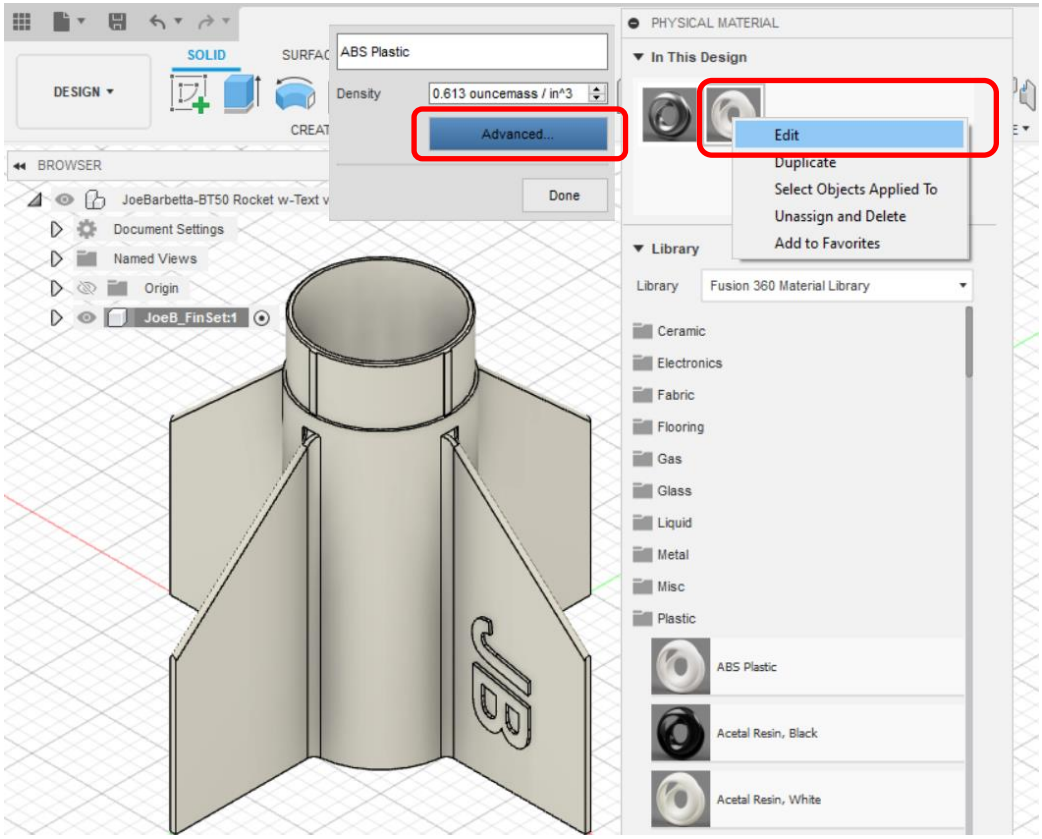
- right-click on the *Component* name and select **Physical Material**



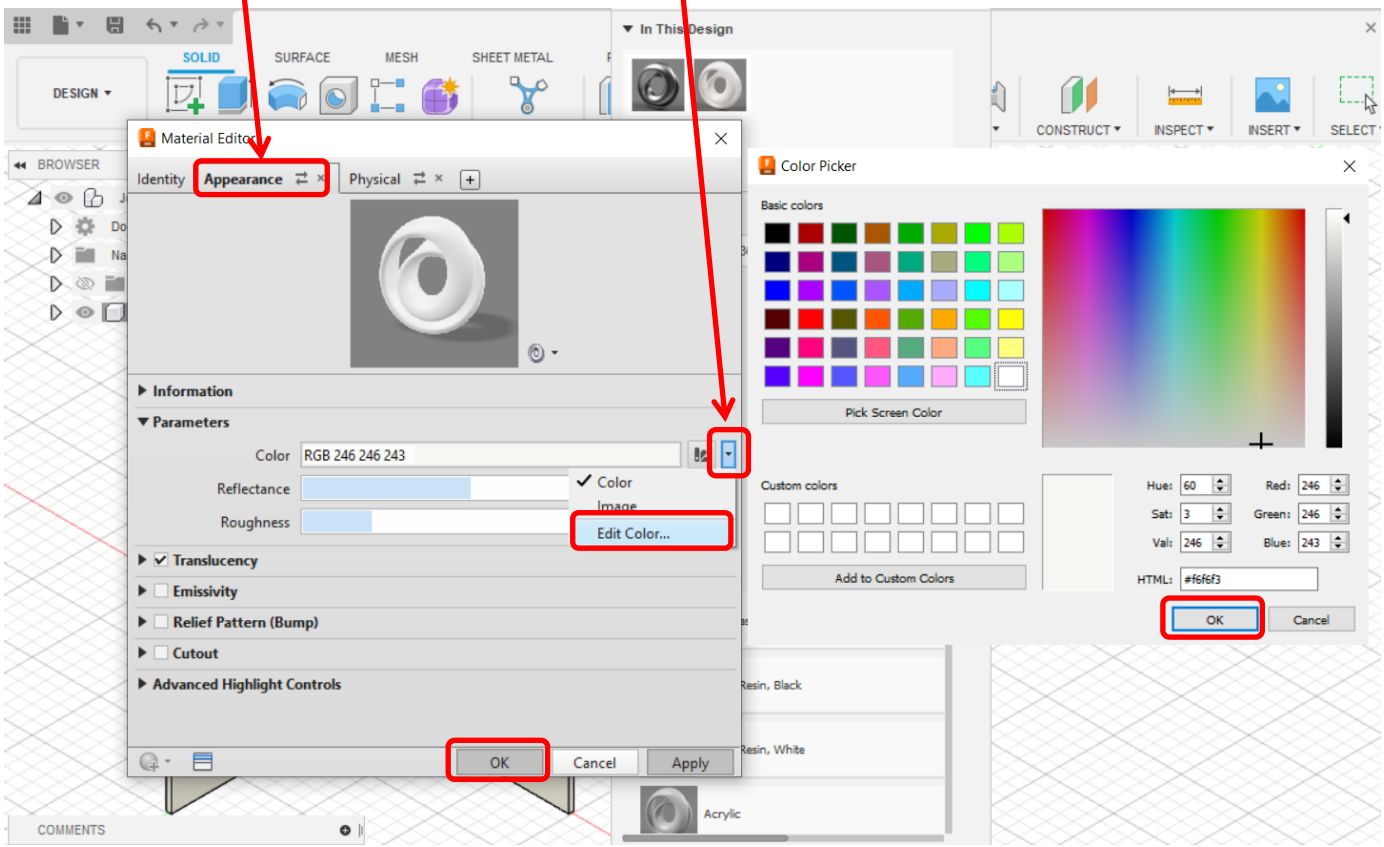
- in the **Library** section open the **Plastic** folder. You may have to scroll to find it.
- click on the **ABS Plastic** icon and **drag it** onto the Fin set. The Fin Set should turn white.



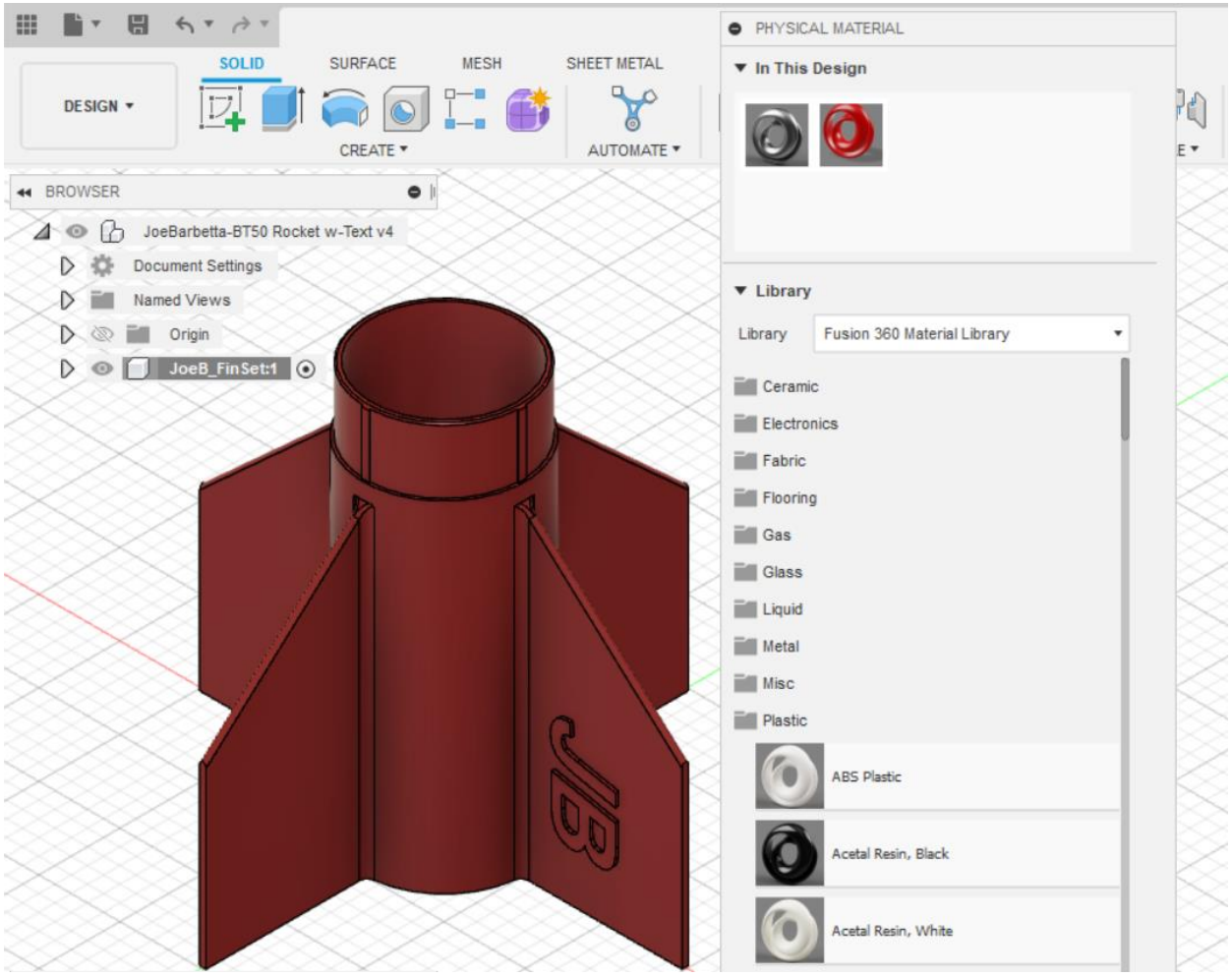
- right-click on the new white icon that appeared in the top "*In This Design*" section and select **Edit**
- click **Advanced..** button in the pop-up window



- select top **Appearance** tab
- in the **Parameters** section click on the small down arrow all the way to the right of *Color* and select **Edit Color...**
- select from the **Basic colors** on the left or **color selector** on the right and click **OK**
- click **OK** on the *Material Editor* window and click **Close** on the *PHYSICAL MATERIAL* window



If there are components in a project, wherein different colors are desired, the white ABS plastic icon in the Plastic folder can be used again and then the new white icon in the top section can be edited to a different color.
It should be noted that the **color of 3D Printed components is determined by the filament** loaded into the printer.

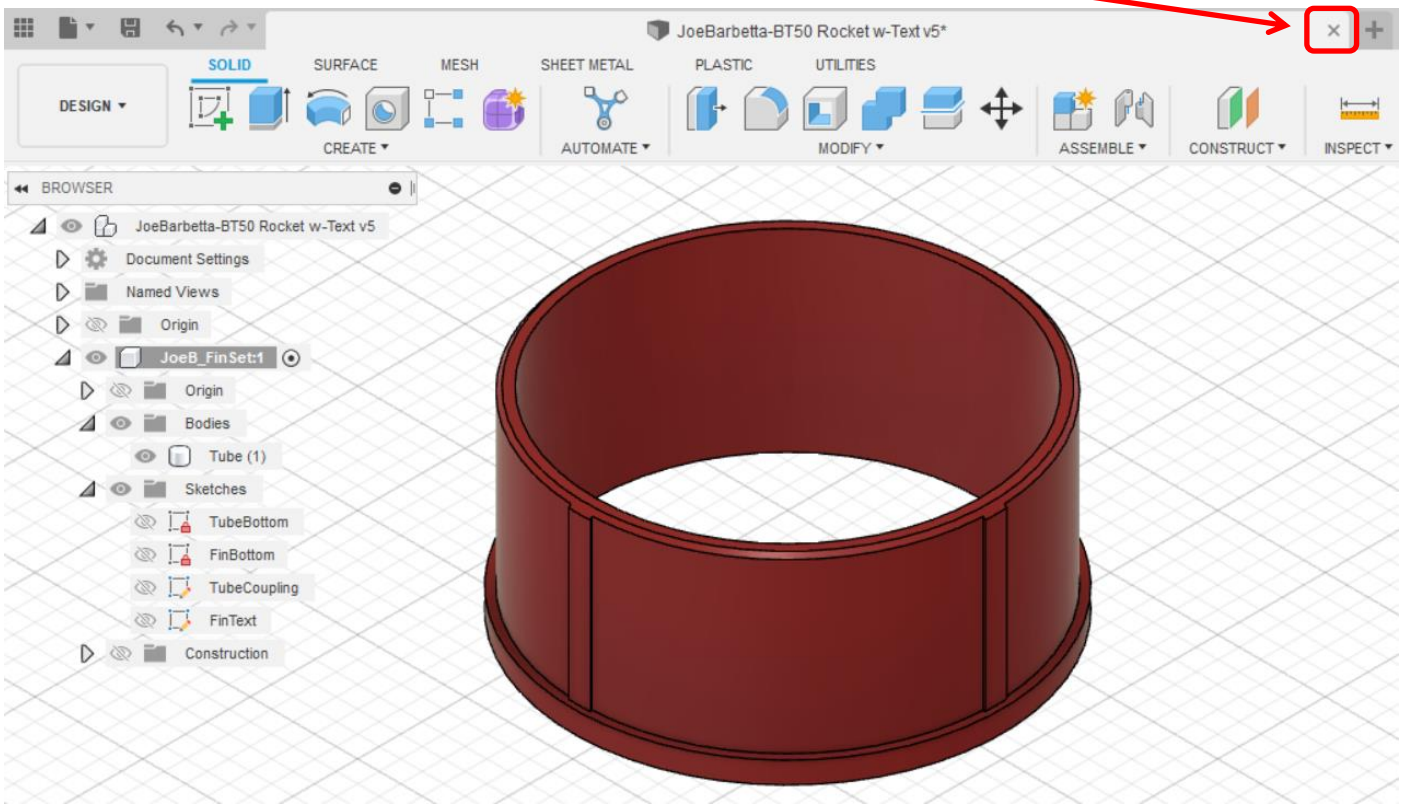


Creating a Test Print

It's a good idea to first print "test sections" of a design for features with critical dimensions. For example, the coupling section with ribs must fit securely in the body tube. Printing the entire component and then finding a problem results in the wasted time and material. One can often cut away large sections of the component resulting in a small test print.

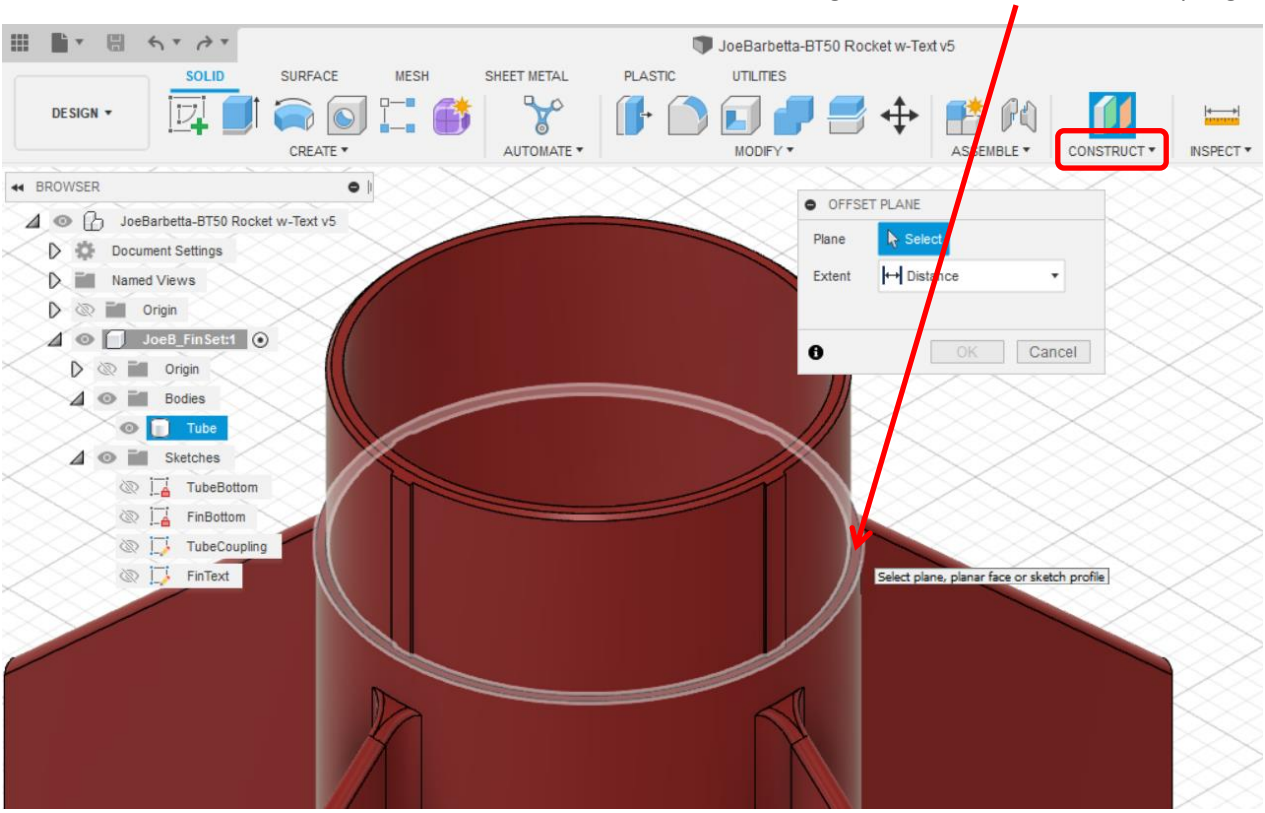
It is a good idea to **save your project now**. Then cut away sections to leave the "test section" and then close the file **without saving**. The project can then be reopened.

- here is the "test print" section to check the fit in the body tube.

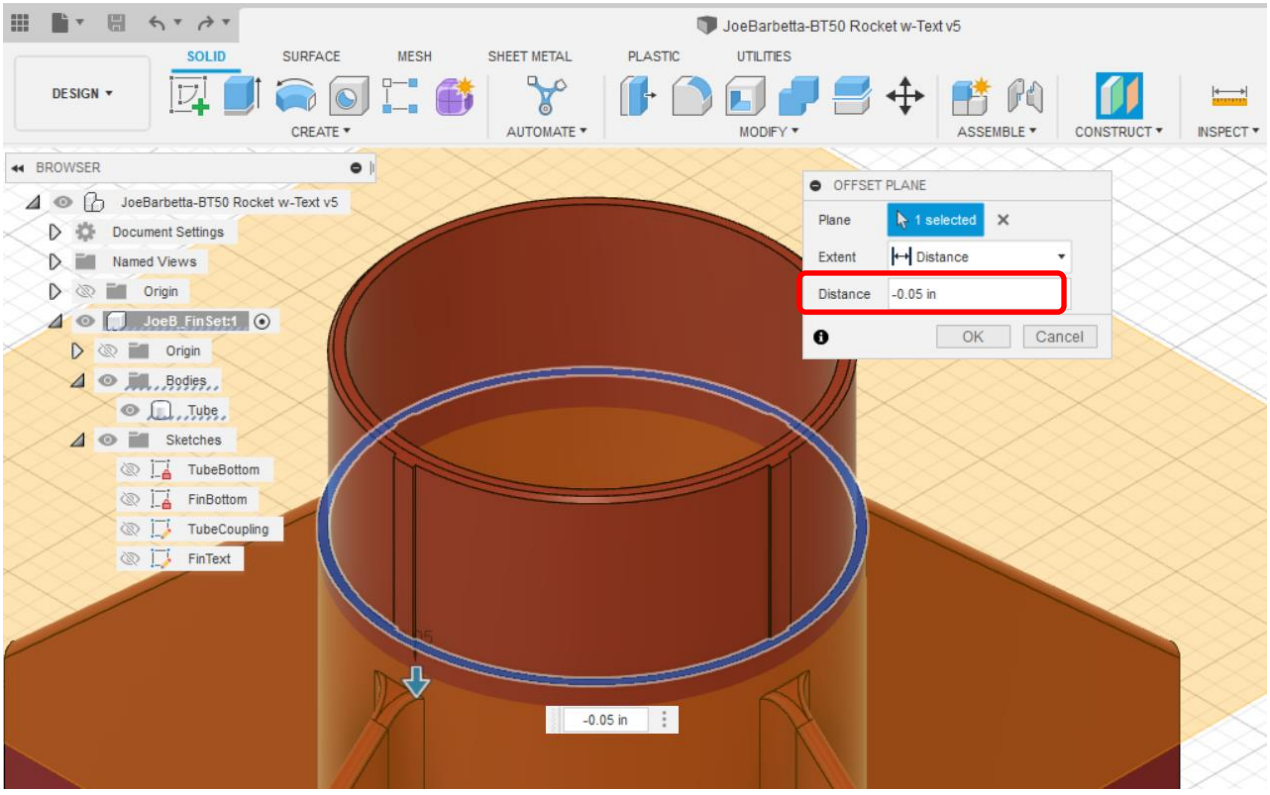


Using a Construction Plane

- within the **CONSTRUCT** menu select **Offset Plane** and click on the region at the bottom of the coupling section.

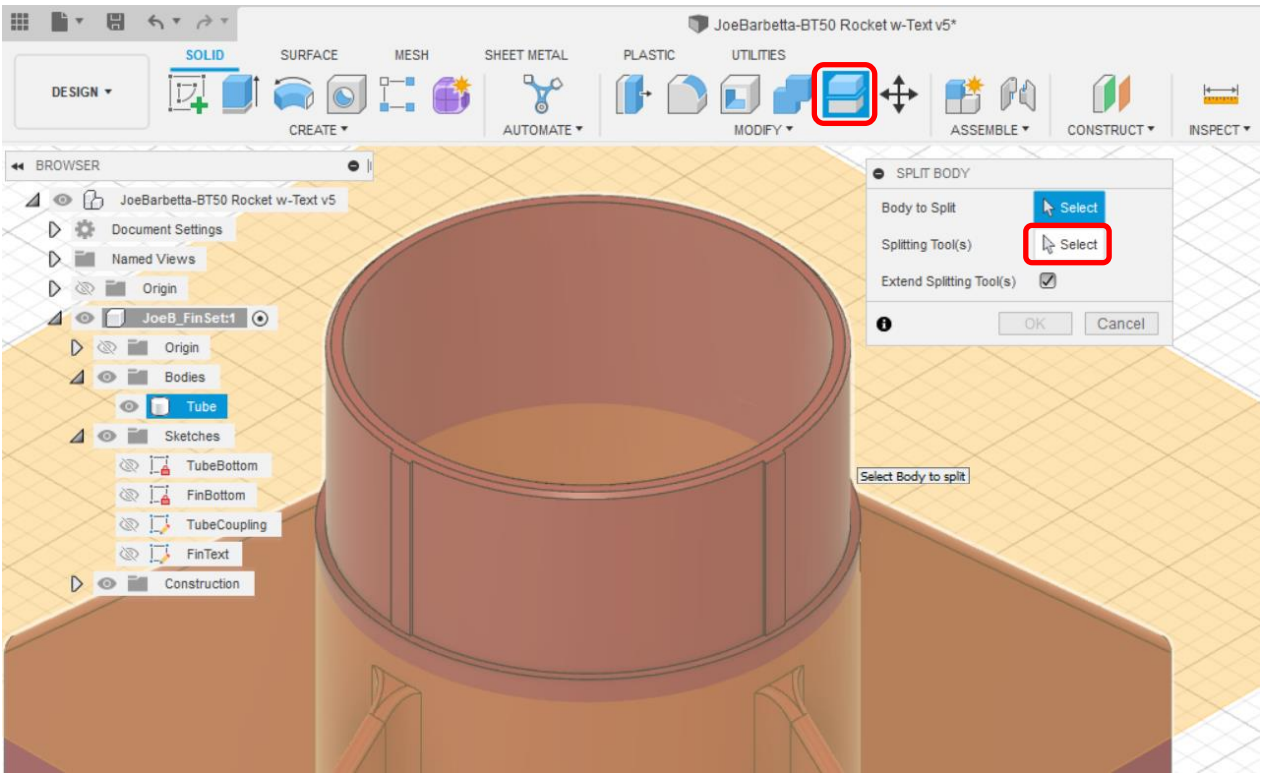


- enter **-0.05** (insure you enter the negative value) for **Distance**, which will shift the plane down slightly.



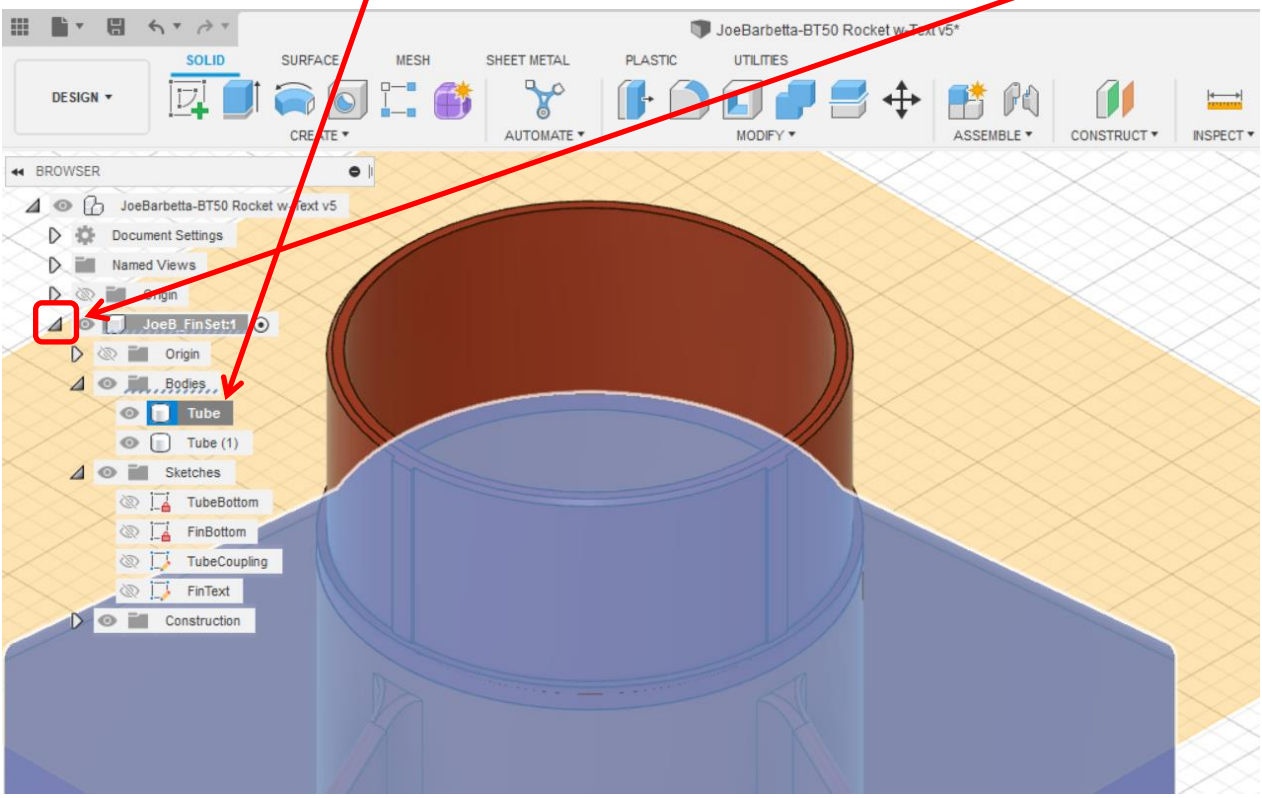
Using the Split Body tube

- select the **Split Body** tool, then click on the FinSet Component.
- click on **Select** to the right of **Splitting Tool(s)** and then click on the yellow Plane.



Note that now there is a now a new Body. When the mouse point hovers over each body, the corresponding body should lighten.

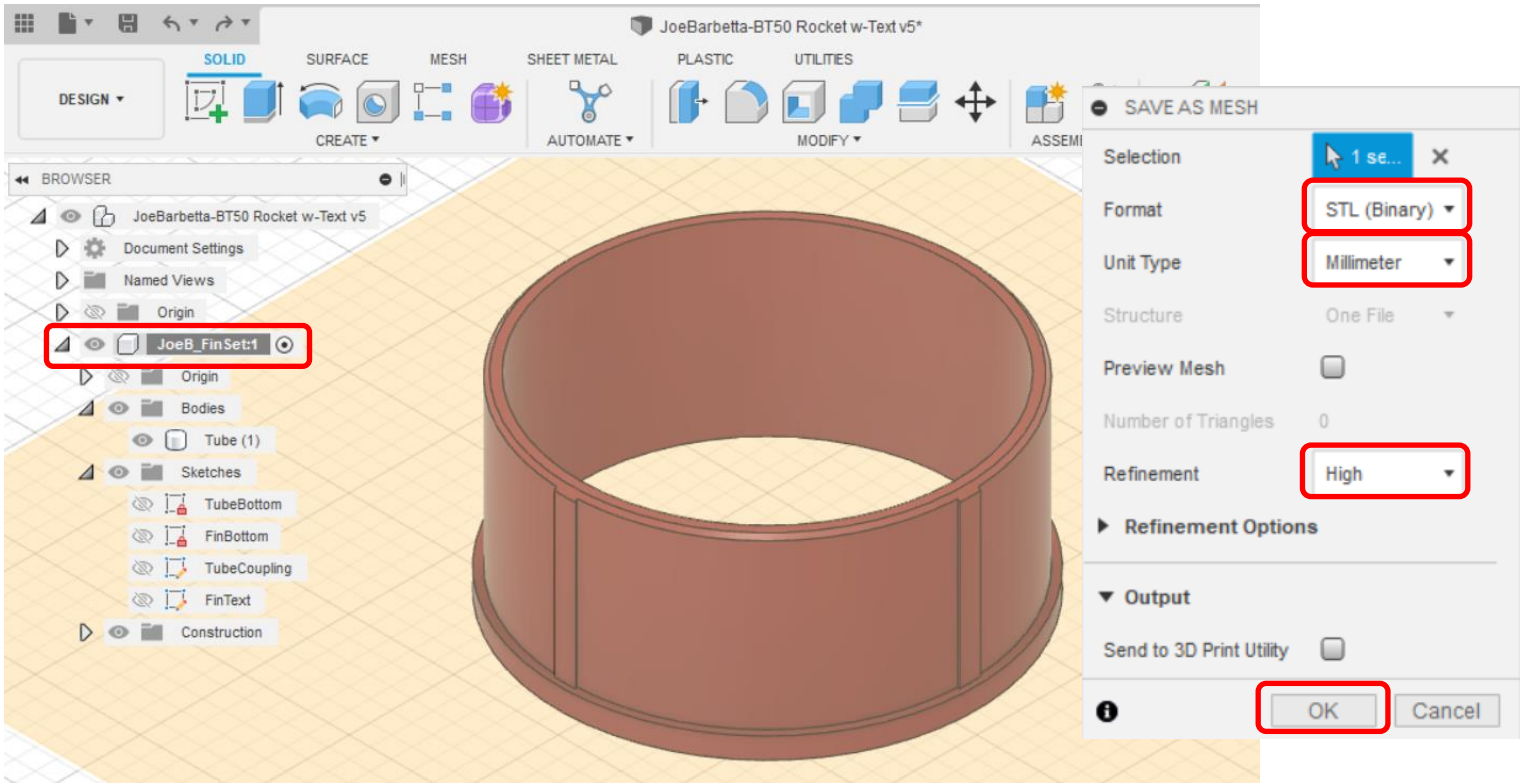
- right-click on the original “Tube” body and select **Remove**. This should delete the fin section. If the wrong body is deleted, click on **Undo** and then remove the other body. If you don’t see the Bodies, use the **arrows** to open these categories.



Exporting a STL file for printing

- right-click on the Component name and select **Save as Mesh**, which is about halfway down the menu list.
- insure **Format** is **STL (Binary), Unit Type** is **Millimeter**, and **Refinement** is **High**. Click **OK**.

- the **Save STL** window will show a file name matching the Component name. Add **Test** to the file name so it will be identified as a test print. **Note the save location of the file, which can be changed if desired.**



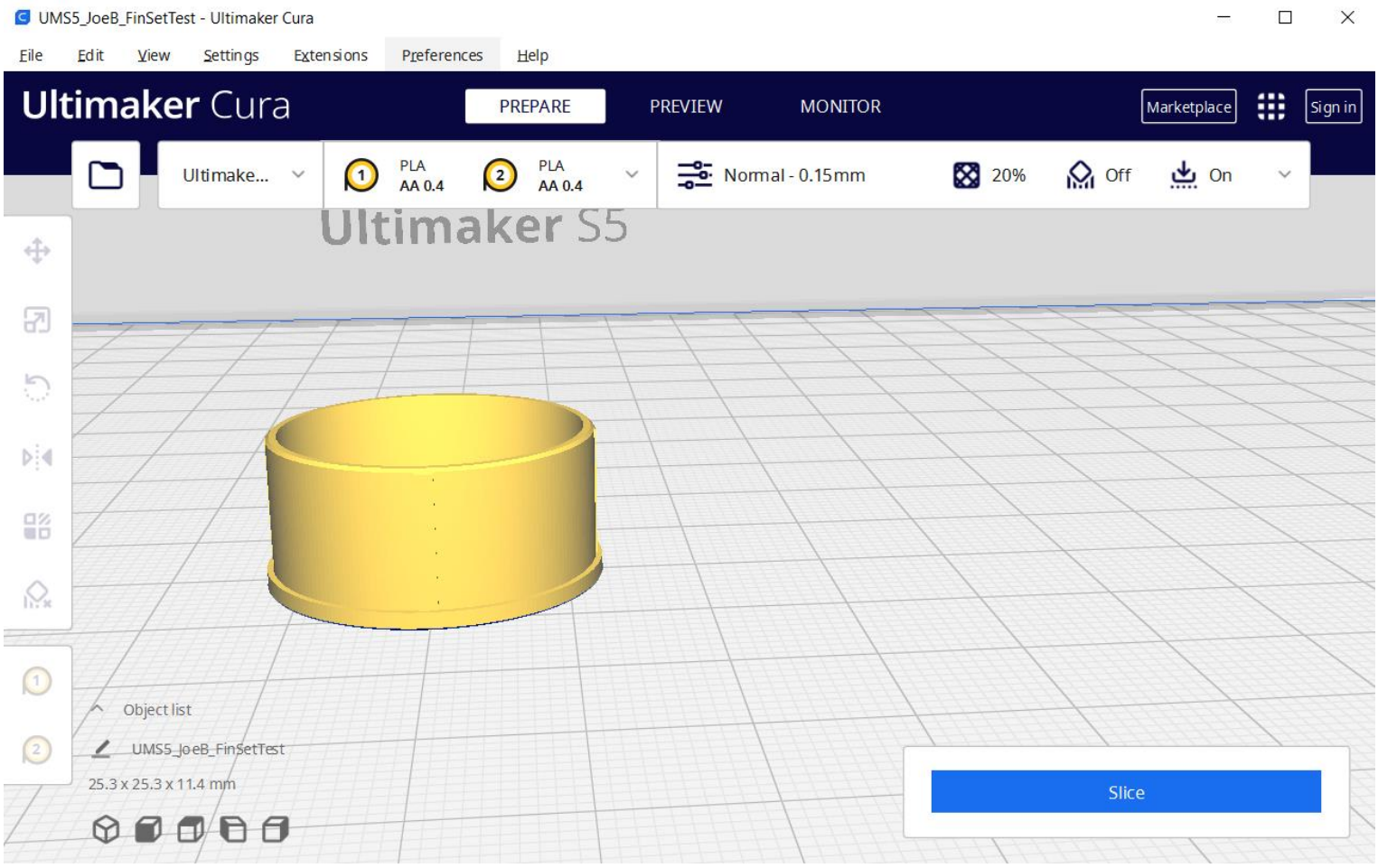
Using the Cura Slicing Program

If this is the first time you are using Cura on this computer, use the **Cura Slicer Program** document on Schoology to get started.

If the **Cura** icon is not on the Desktop, click on the Windows **Search** tool, which can a magnifying glass icon.
Type Cura in the search box and you should find it.
If Cura prompts you to sign in or create an account, **chose not to**. If it is the first time Cura is used on that computer, it may prompt you to select a printer. Select "**Non Networked Printer**" and then **Ultimaker 5S**, which may be the top default printer. You may also have to click **Next** a few times.

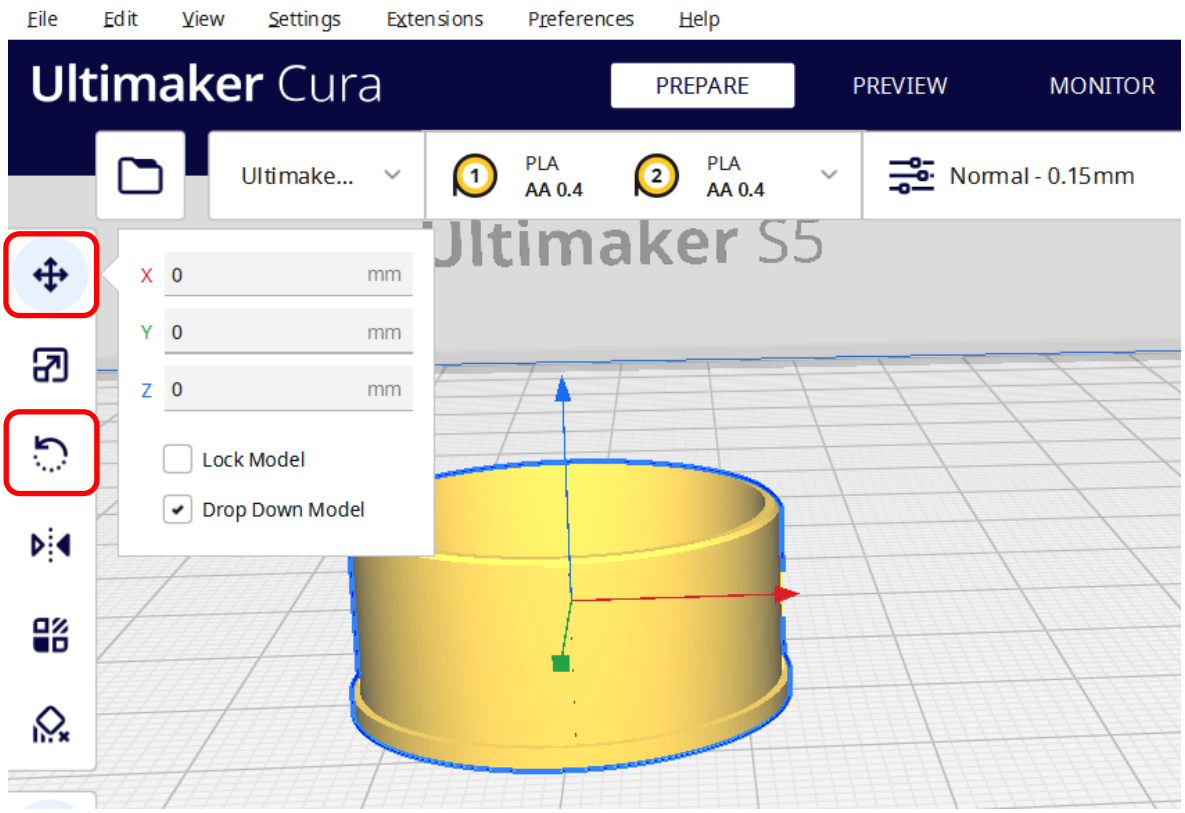
- from the top **File** menu select **Open File(s)**. The top navigation bar of the Open file(s) can be used to navigate to the **.stl file location**. Select the **.stl file** and click **Open**.

Below is the view of the test print of the FinSet coupling after zooming in.

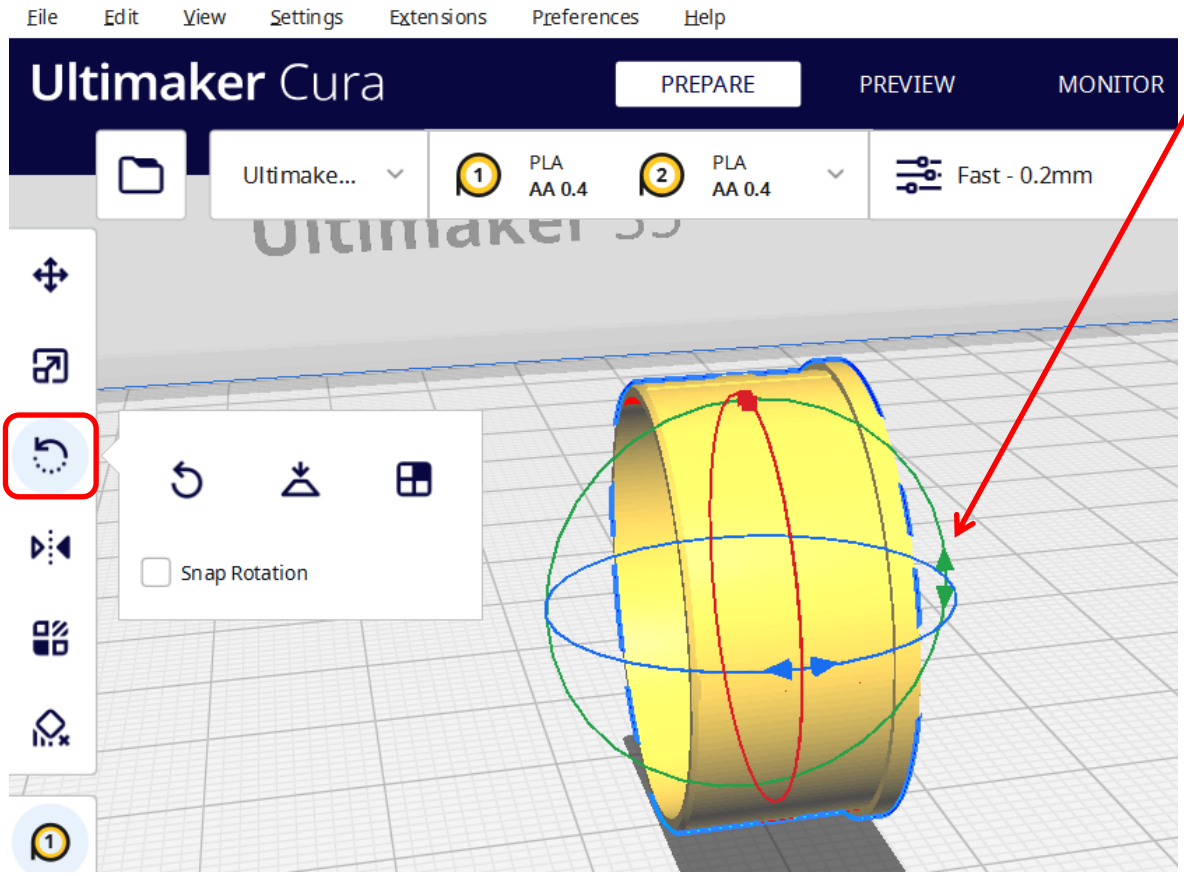


This program is annoying because it uses different methods to change the object view.
- Turning the mouse wheel zooms in and out. Holding the mouse wheel down allows panning.
- Holding the right mouse button changes the view angle.
- The **View Cube** options at the screen bottom left allows jumping to different views.

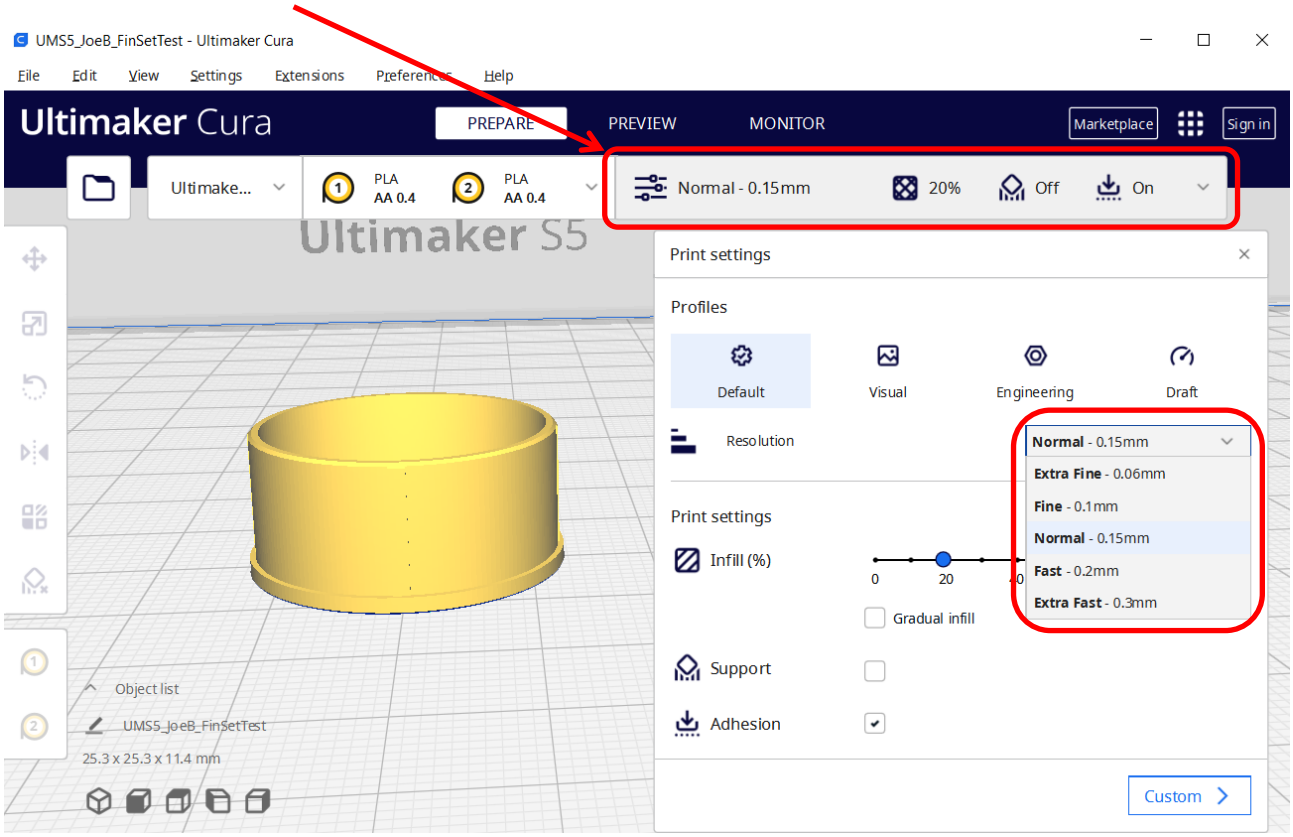
- Click on the object to select it. This will enable options on the left side of the screen.
Often, the object to be printed will be located in the center of the build plate and no manipulation is needed.
Here one can see that the object is properly oriented on the build plate.



If an object is opened and is not in the proper orientation, the Rotate tool can be selected and the rotate arrows for the desired axis can be clicked on and moved. The angle will jump in 15 degree increments to make it easy to rotate an object.

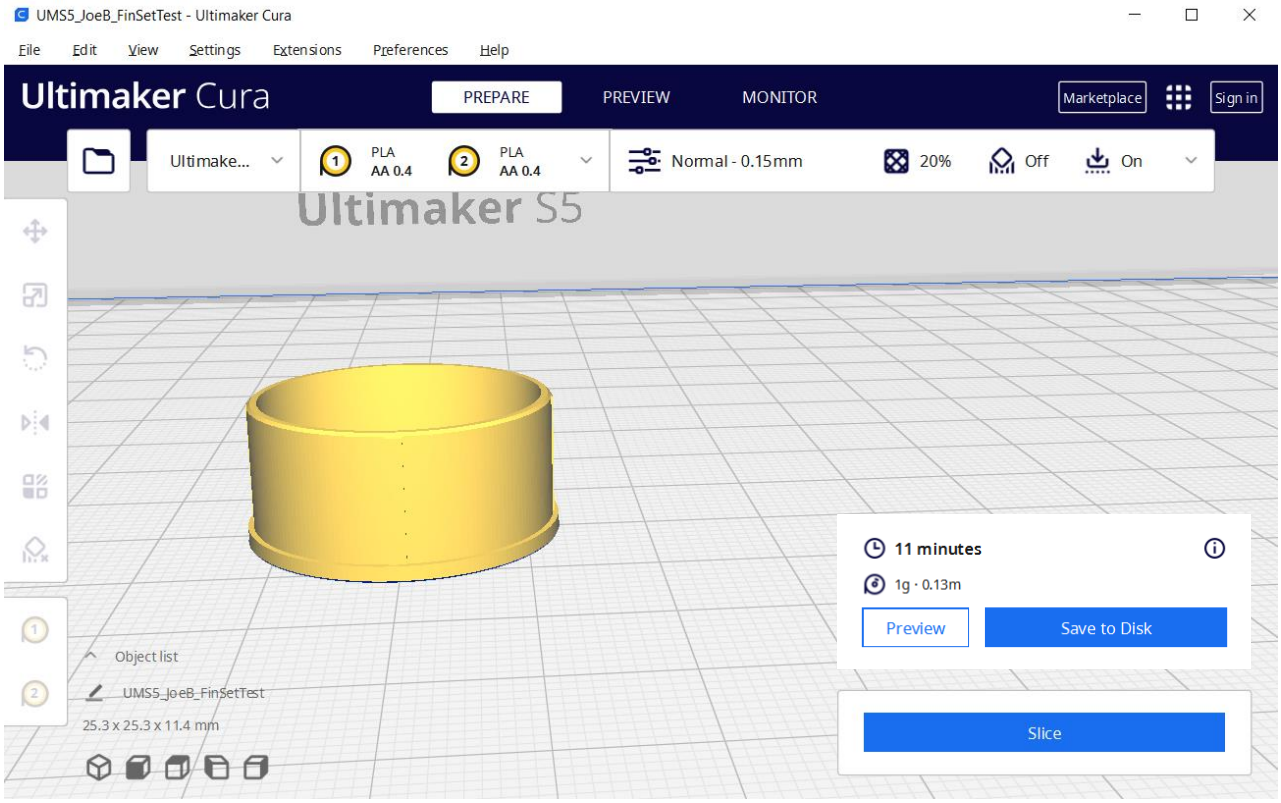


- click on the **settings bar** to open **Print settings**. The most common setting to change is the **Resolution**.



The Resolution setting is a compromise between quality and print speed. A thinner layer will result in better accuracy for top surface features. For example, the top of any angled or rounded surfaces will appear smoother with smaller resolutions because each layer is thinner. When prioritizing printing speed, a larger resolution for thicker layers can be selected.

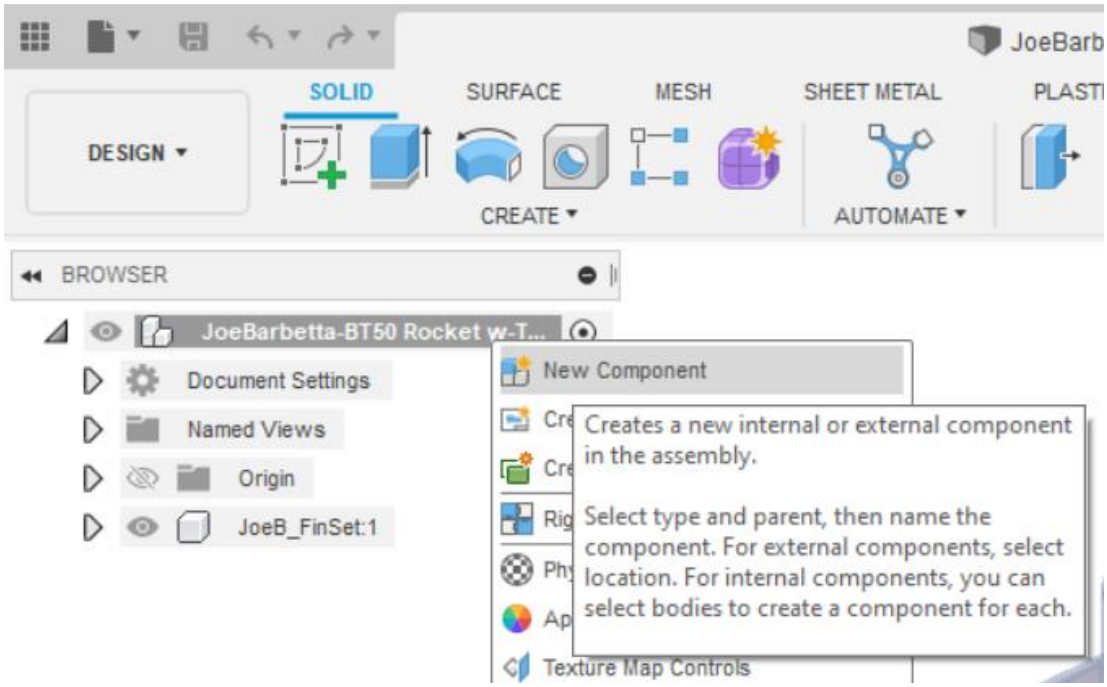
- click on the **Slice** button. A new window will pop up, which will show how long the print will take. Clicking on **Save to Disk** will allow the **.ufp** to be saved. This is the file that will be loaded onto the 3D Printer.



Creating a New Component for the Nose Cone

- right-click on the top line in the **Browser** and select **New Component**. This will insure it is a "top level" component and not as part of another *Component*.

- enter the component name: **<your first name and last name initial> NoseCone** e.g. **JoeB_NoseCone**



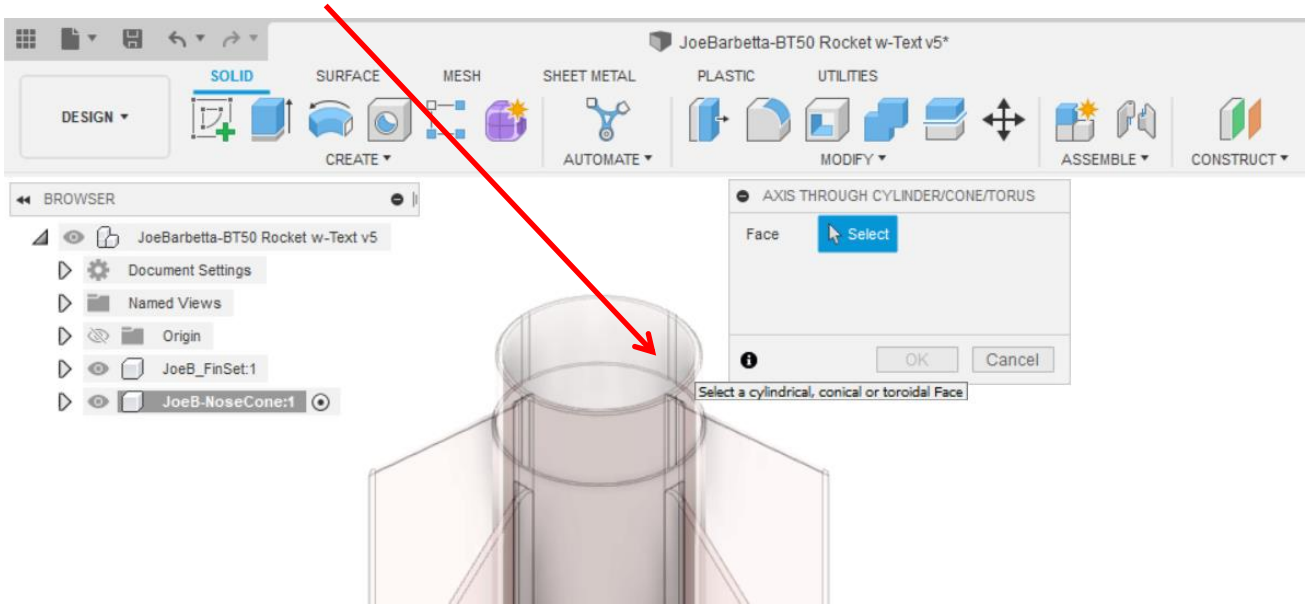
- next to **User Parameters** click on **+** and enter the **Name** and **Expression** for each item in the below list. Note that you will need to click **+** for each item. You may be able to copy and paste Expressions, but **if it shows as red you may have to type the expression.**

- **Don't forget to click OK when done!**

Name	**Expression**
Nose_Length	3.000
Nose_Thickness	0.040
Nose_OD	0.997
Nose_CouplingOD	0.950
Nose_CouplingLength	0.400
Nose_CouplingRibOD	0.960 in
Nose_CouplingRibPos	Nose_CouplingOD / 2 - 0.01 in
Nose_CouplingChamfer	(Nose_OD - Nose_CouplingOD) / 2

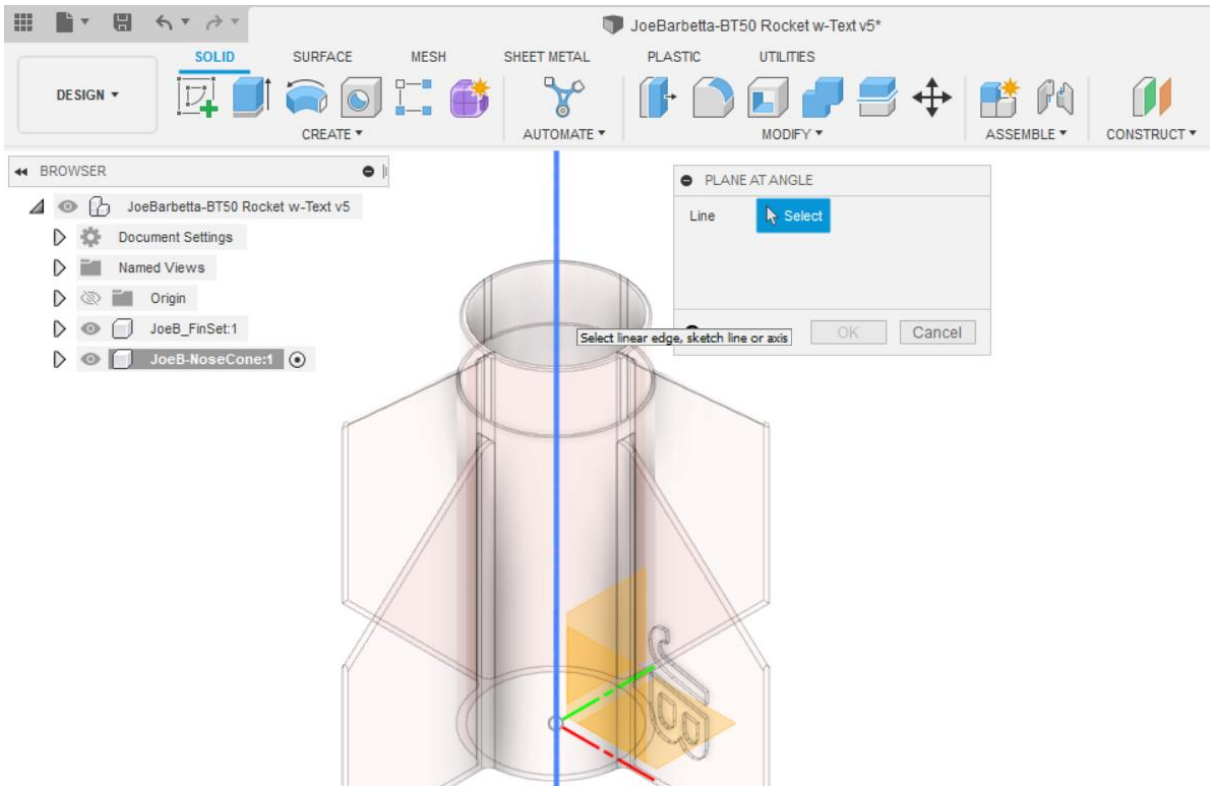
Creating a Construction Axis

- within the **CONSTRUCT** menu select **Axis through Cylinder/Cone/Torus**
- click on the **inside of the FinSet tube** and click **OK**. A blue "center line" with alternating long and short dashes should appear.

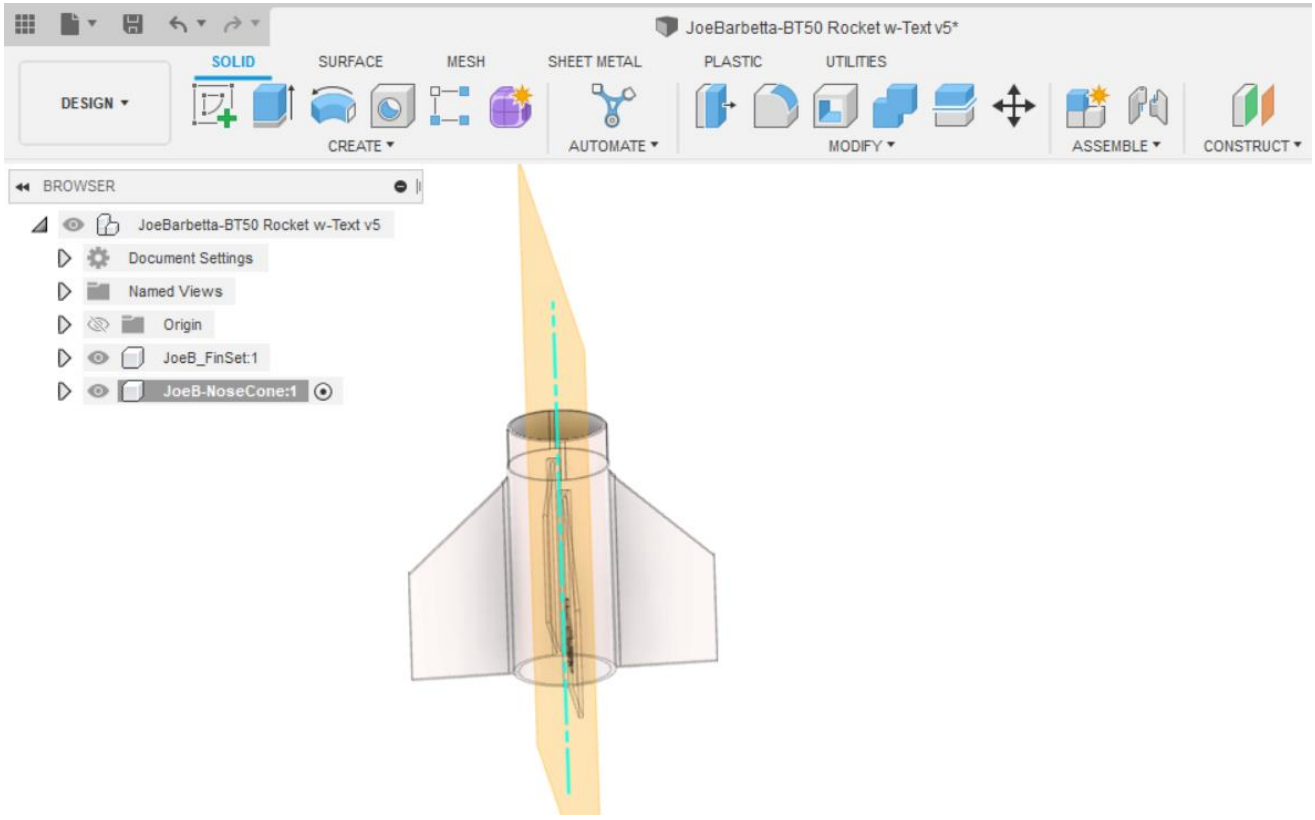


Creating a Construction Plane

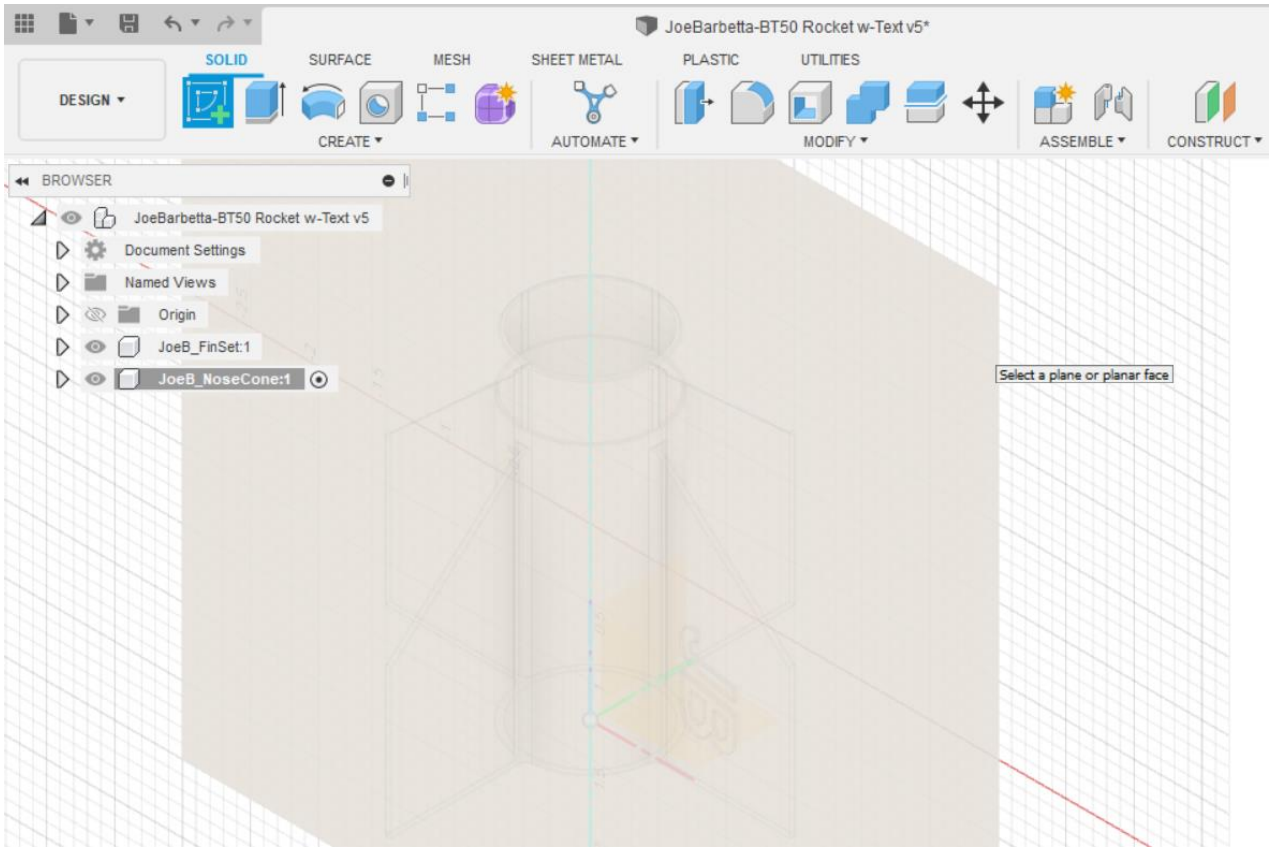
- within the **CONSTRUCT** menu select **Plane at Angle**
- click on the **Construction Axis** and click **OK**. Do Not enter an Angle. It will be kept at the default of 0.



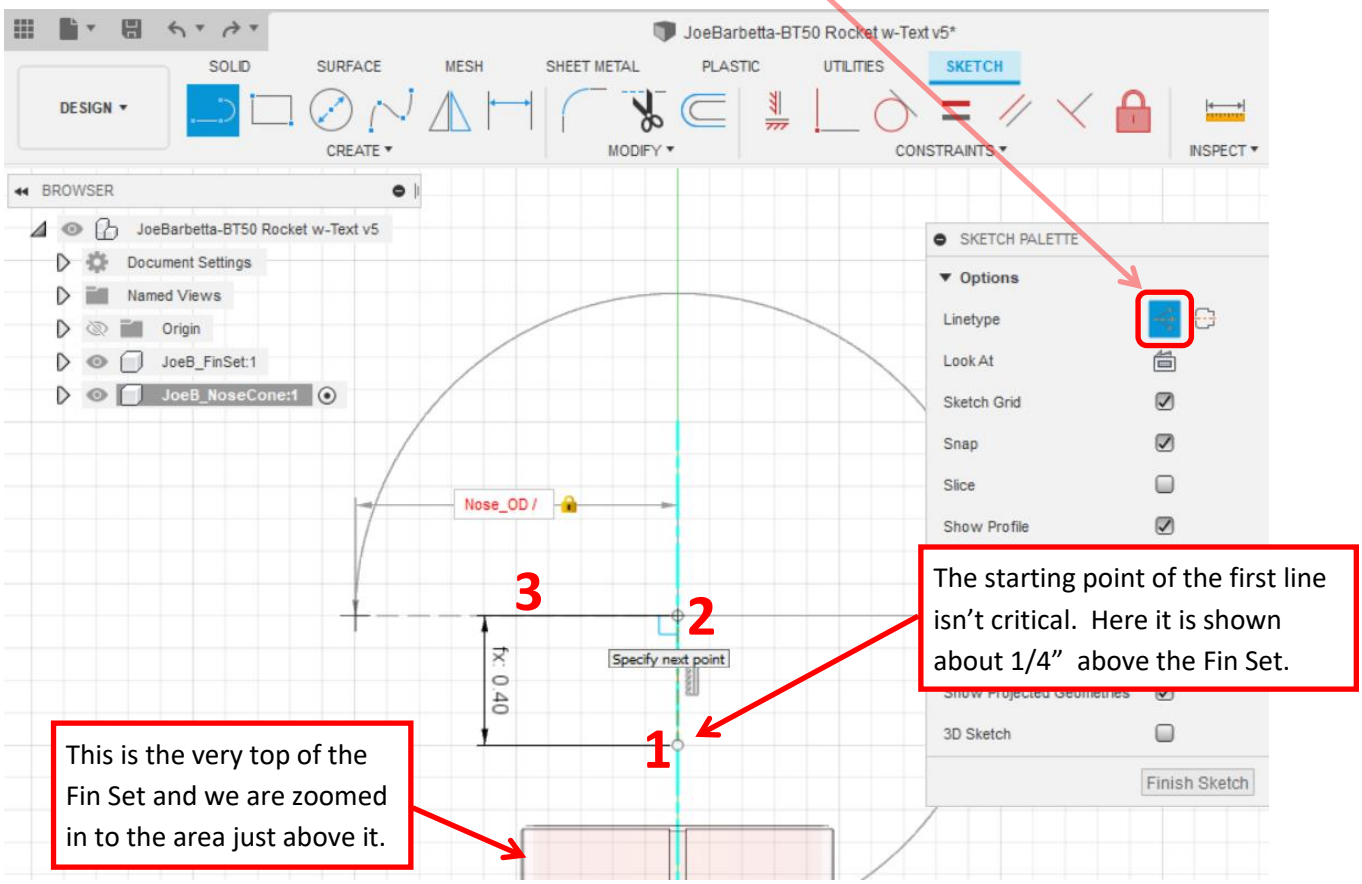
Here the view was changed to show this new vertical plane through the center of the Fin Set. One needs to create a *Construction Plane* if needed for a new sketch that can't be started on an existing plane or object face.



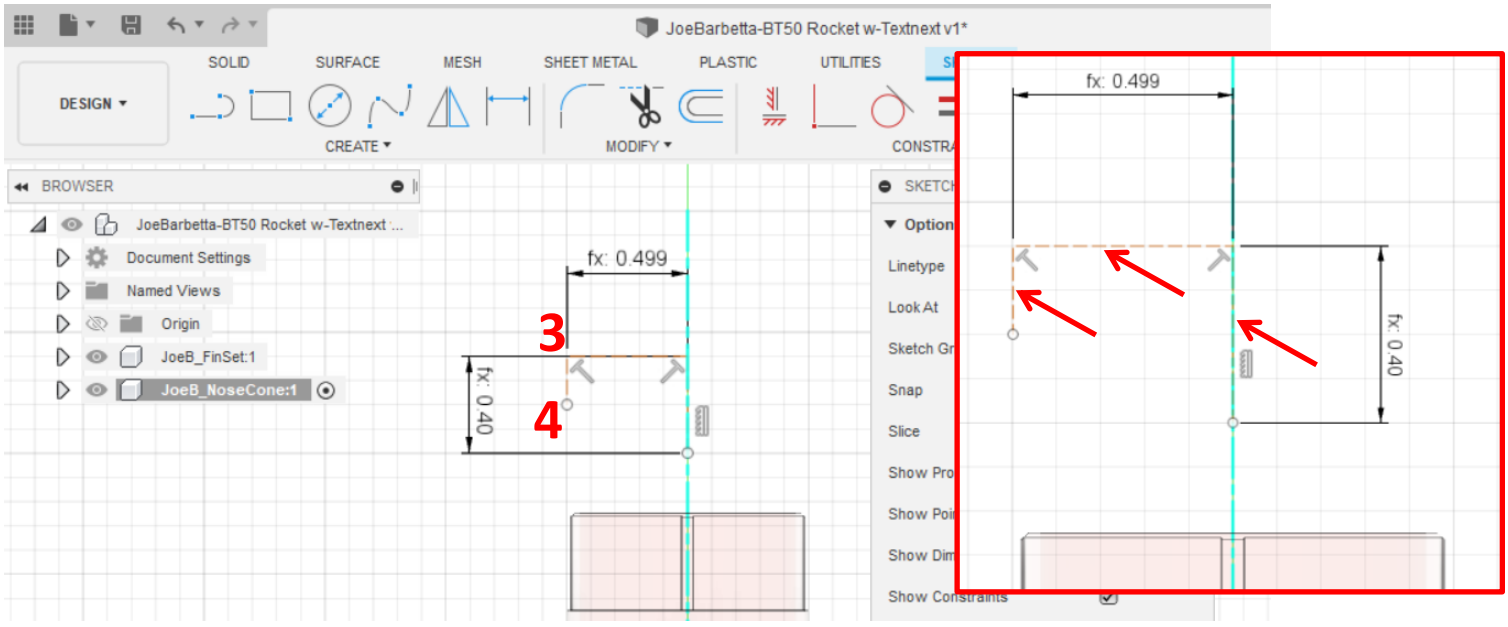
- click on the top **Sketch** tool and click on the **Construction Plane** that you just created



- **zoom in** so that you can sketch **just above the FinSet**
- click on the **Construction icon next to LineType** to turn it blue
- create a line starting at **1** (on the centerline just above the FinSet) and upward. Type **n** and select **Nose_CouplingLength**
We are using **n** now because this will list the parameters starting with those starting with **n**.
- create another line from **2** (the end of the previous line) to the left. Type **n** and select **Nose_OD** and then use the right arrow key to allow adding **/ 2** after **Nose_OD**. This is an example of combining a parameter with an operation for a dimension and should result with the end of that line near **3**.

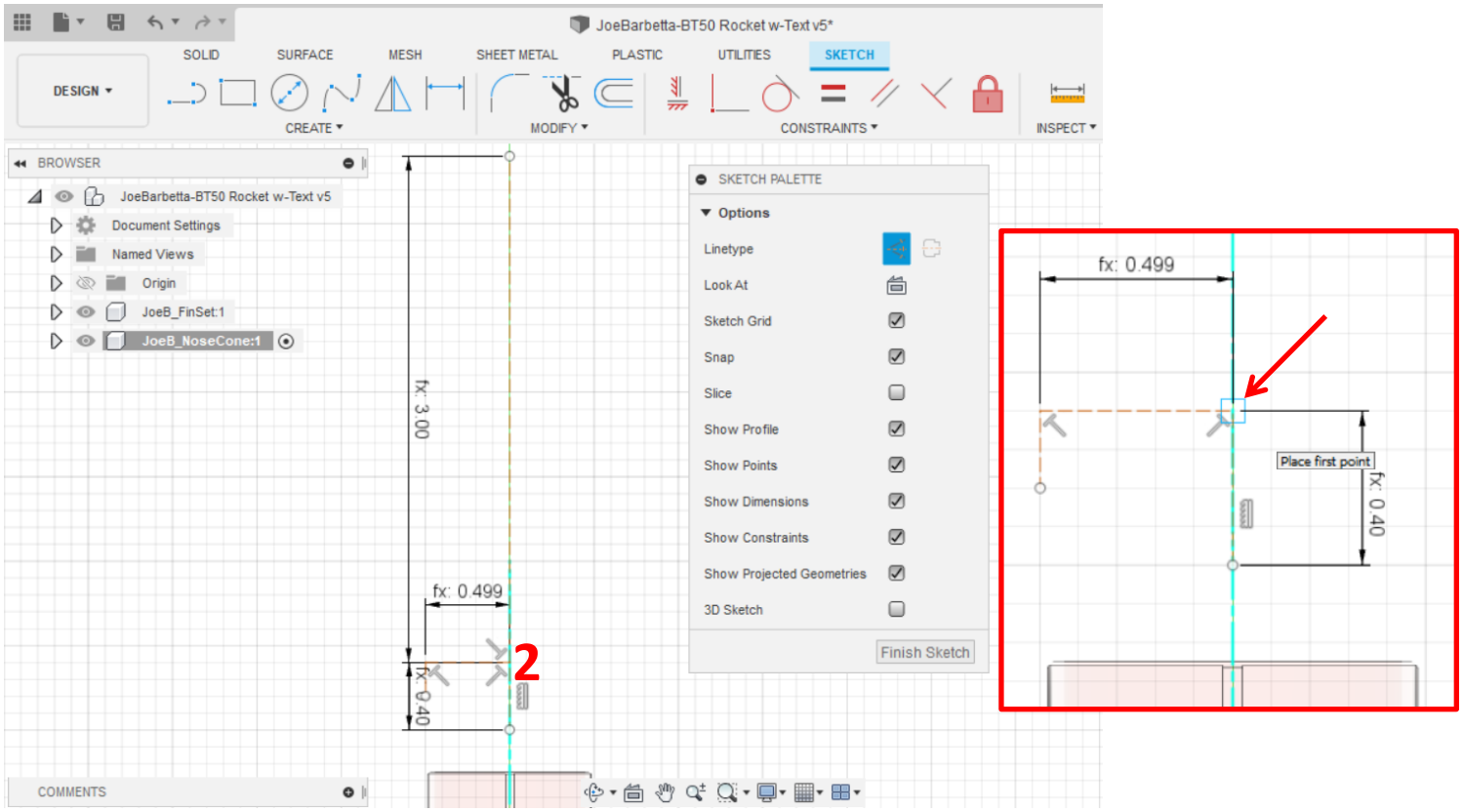


- create a line from **3** down **0.2** in to near **4**. Click on the fx: 0.040 dimension and move it to the right and a close-up should look as that on the right. There should be 3 orange dashed lines with one mixed with the green line.



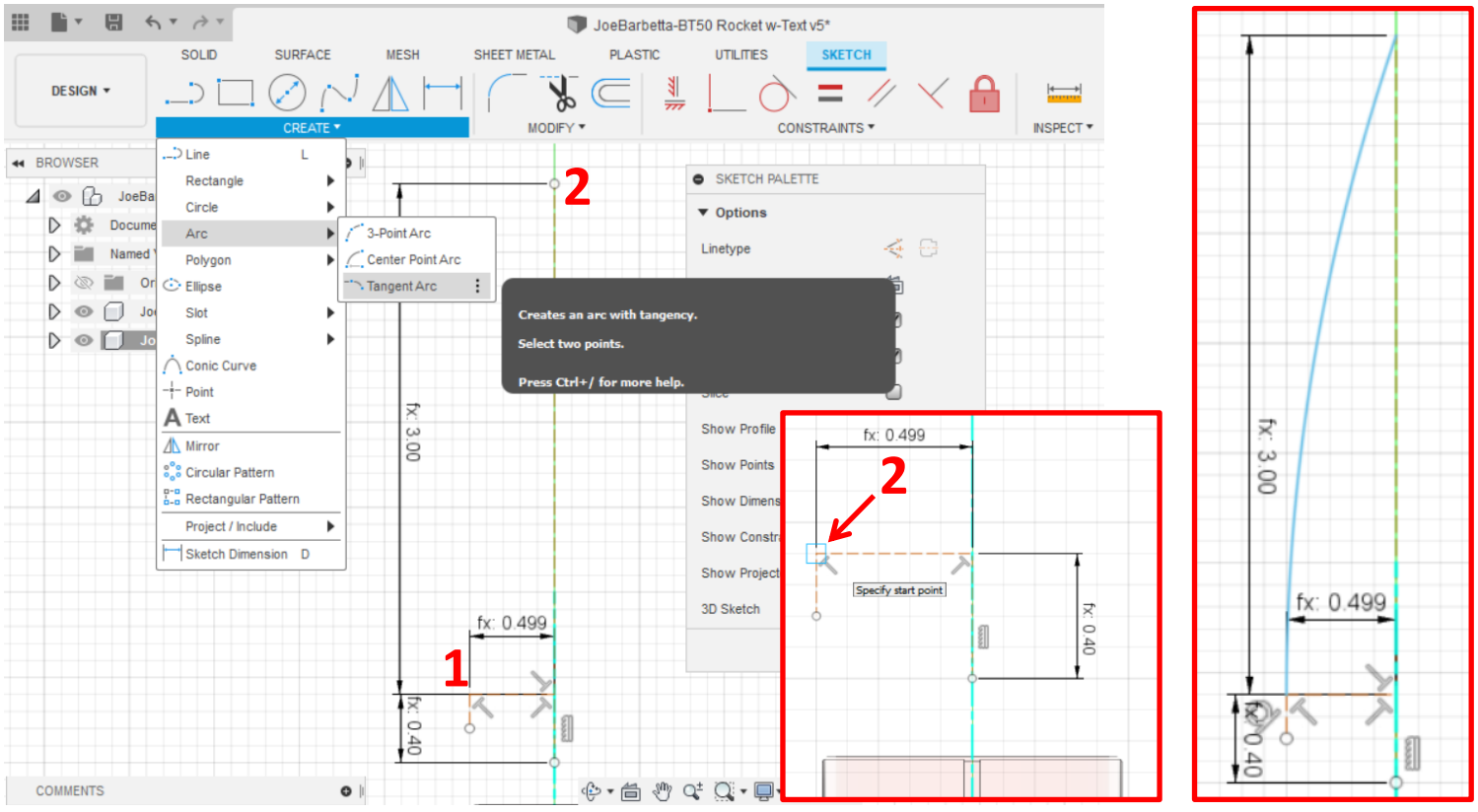
- zoom out and create a line starting at **2** and upward. Type **n** and select **Nose_Length**.

Remember not to worry if the dimension lines look different. In fact, one can click on a dimension line and move it without affecting the design. The close-up picture shows where this line should start from.

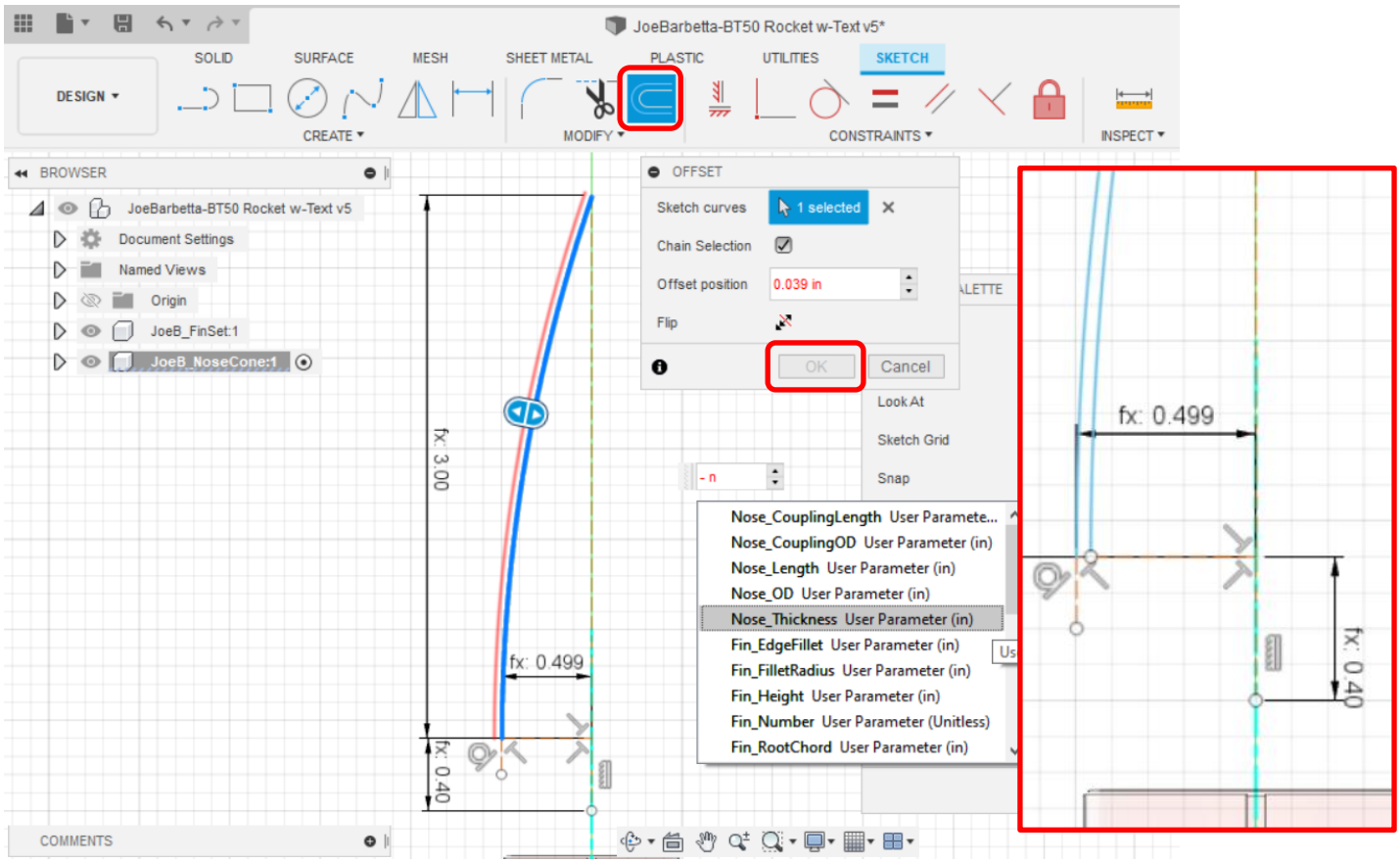


- click on the Construction **icon next to LineType** to turn off the blue highlighting to draw "normal" lines.

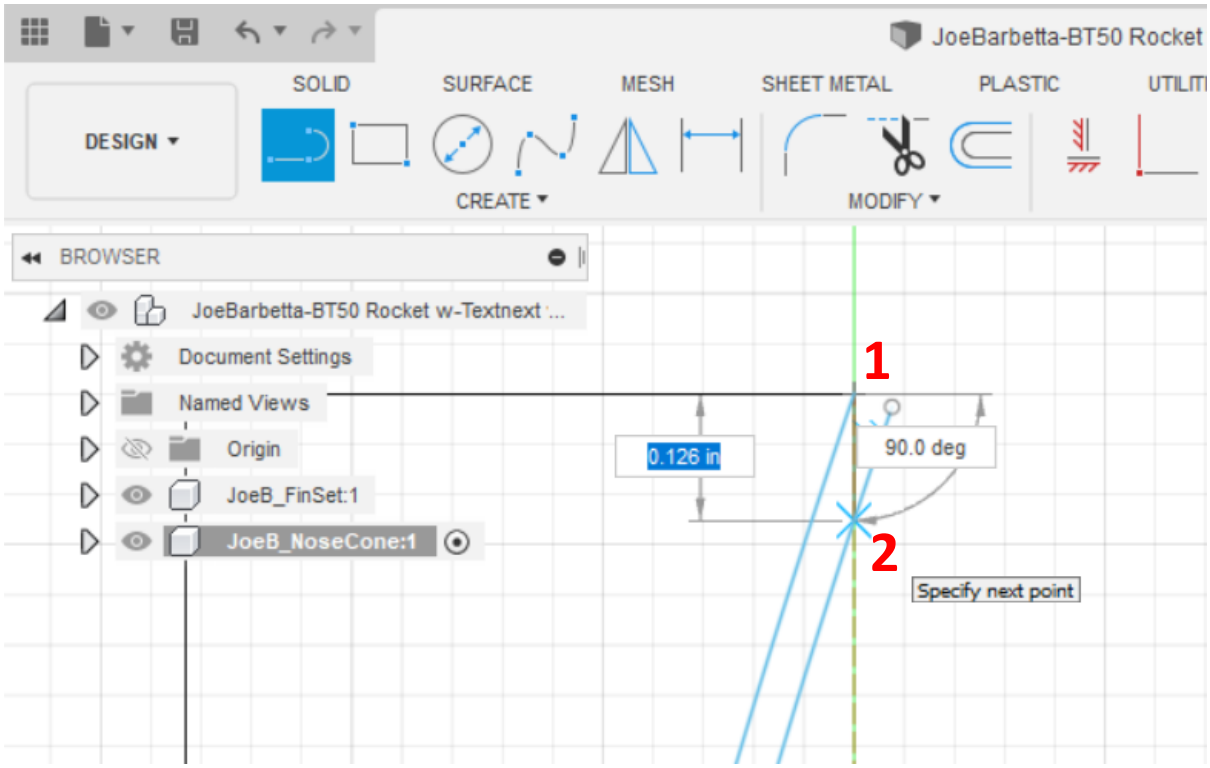
- from the **CREATE** menu select **Arc** and then **Tangent Arc** and click on point **1** (see the close-up for this point) and then extend the arc up to point **2**. The result is shown on the right.



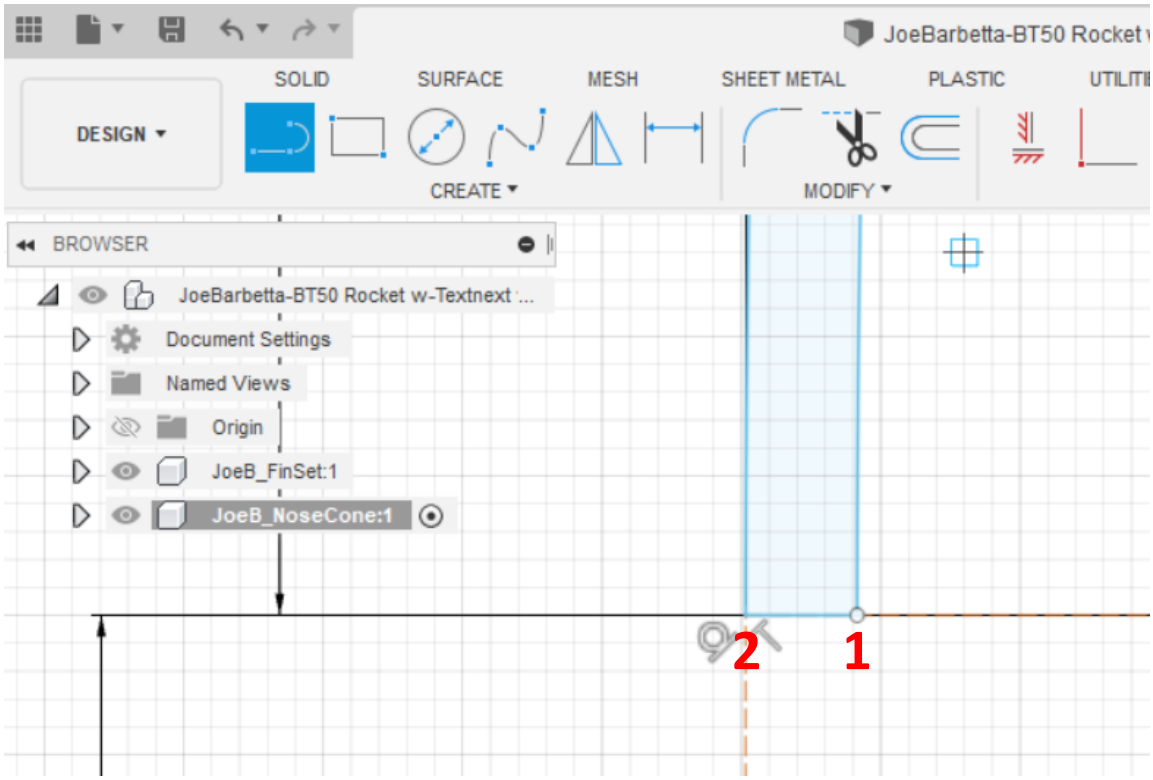
- select the Offset tool and type **- n** (note the negative before n) and select **Nose_Thickness**. The negative is needed to force the red arc to the right of the blue arc.
- click **OK** in the **OFFSET** window. The close-up picture shows what the bottom area of the Sketch should look like now.



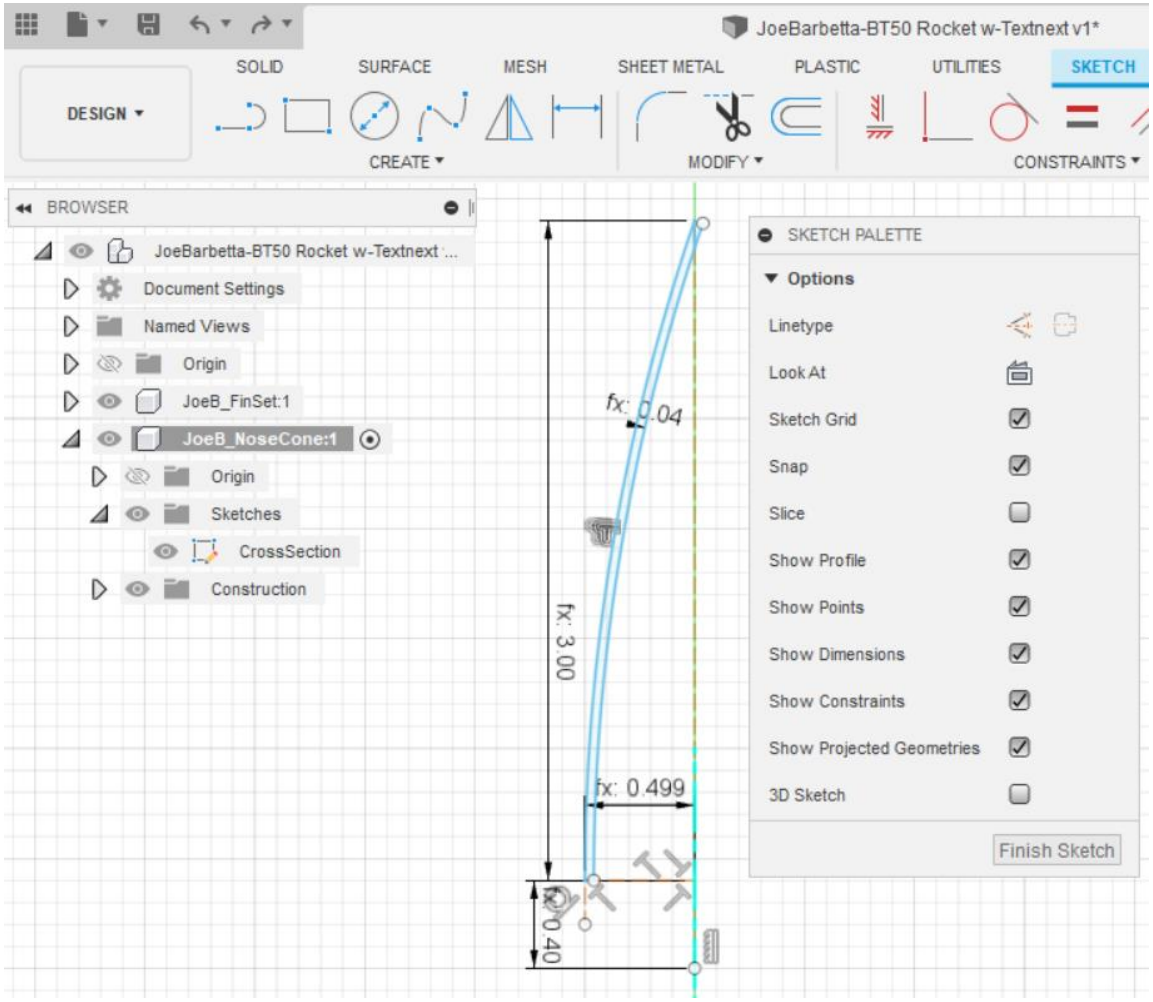
- zoom in to the top and create a line from the top point at **1** and down until it meets the lower arc at **2**.



- zoom in to the bottom and create a line from point **1** to point **2** to close the bottom. The profile interior should turn blue.

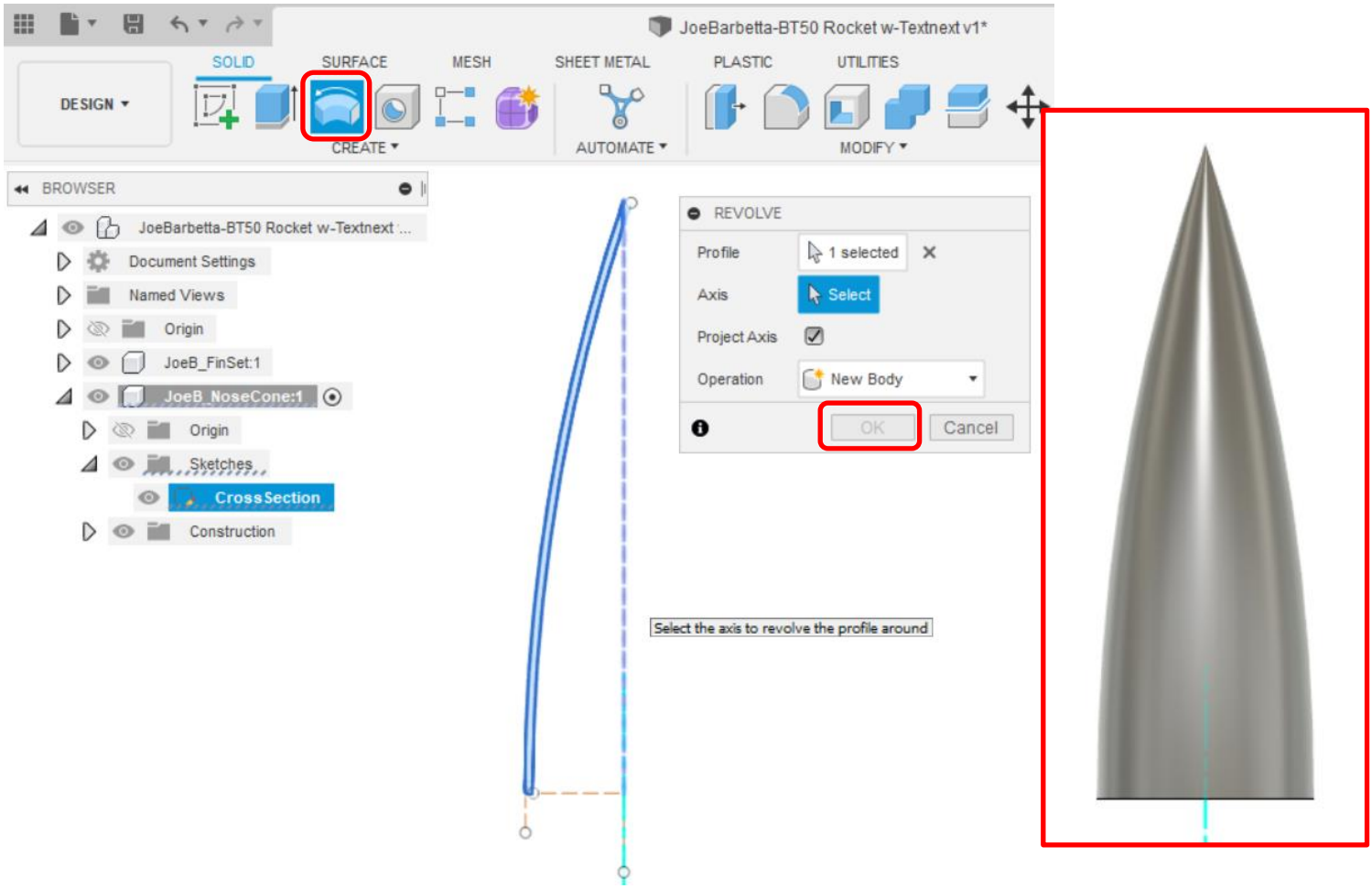


- when zoomed out it should look like below. Again, note that the black dimension lines may look different.
- click **Finish Sketch**

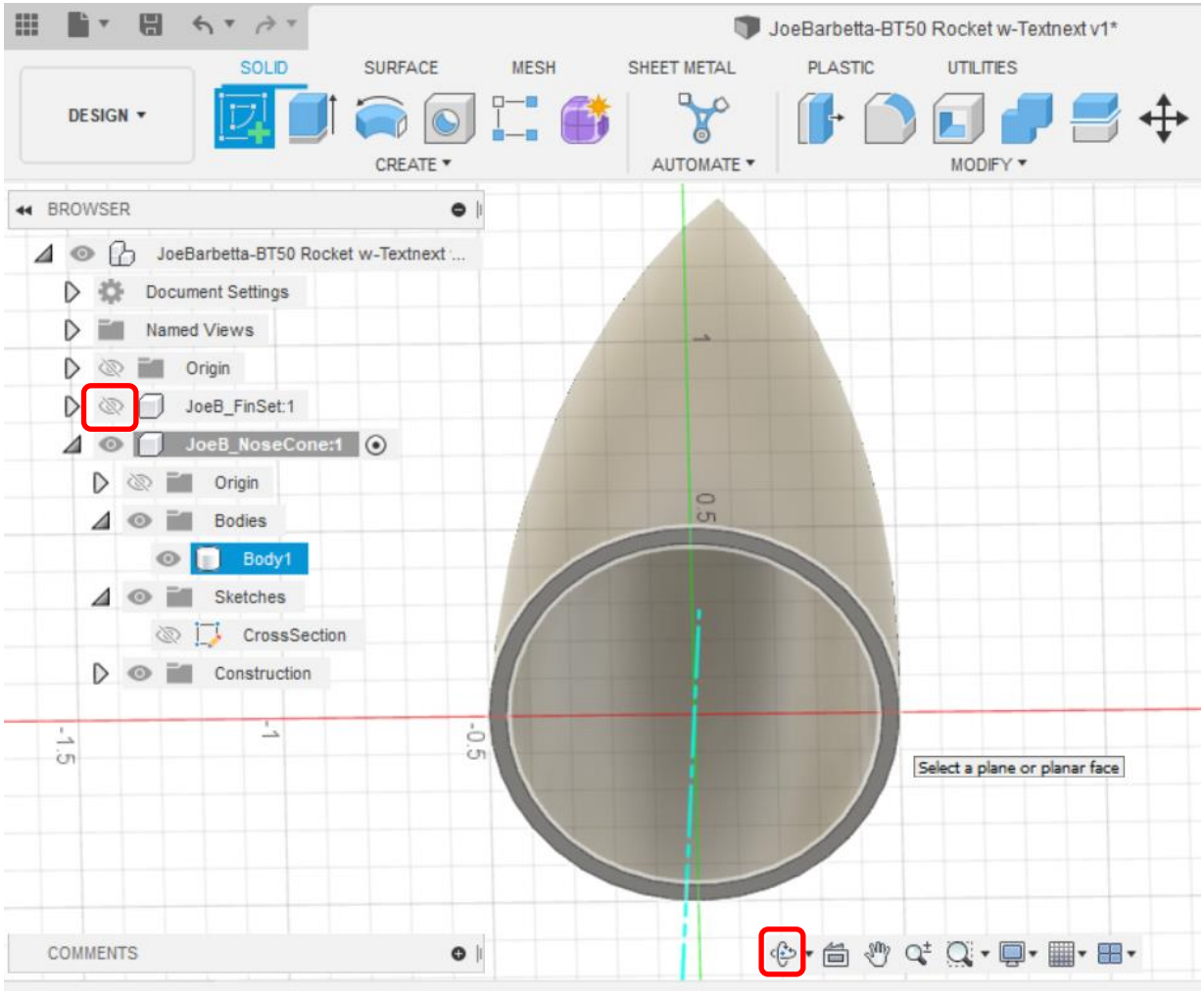


Using the Revolve Tool

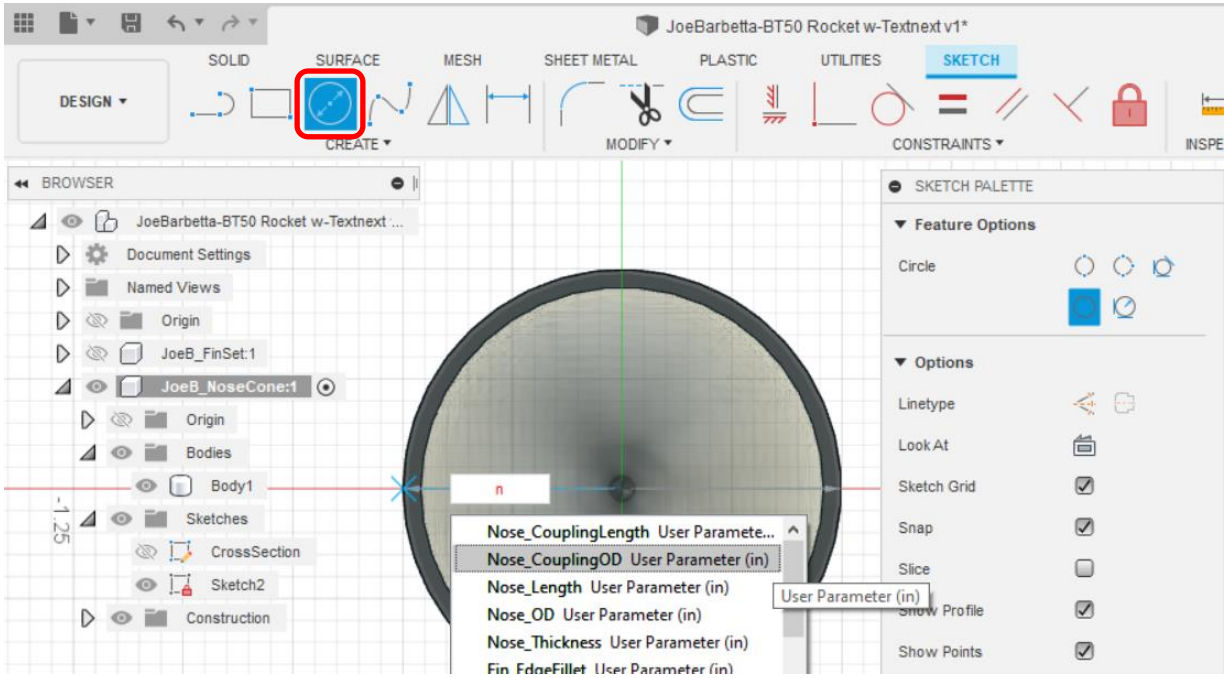
- select the top **Revolve** icon or select **Revolve** from the **CREATE** menu. The profile should turn darker blue.
- click on **Select** next to **Axis** and click on the center line. You should see the resulting closed cone.
- click **OK** in the REVOLVE window



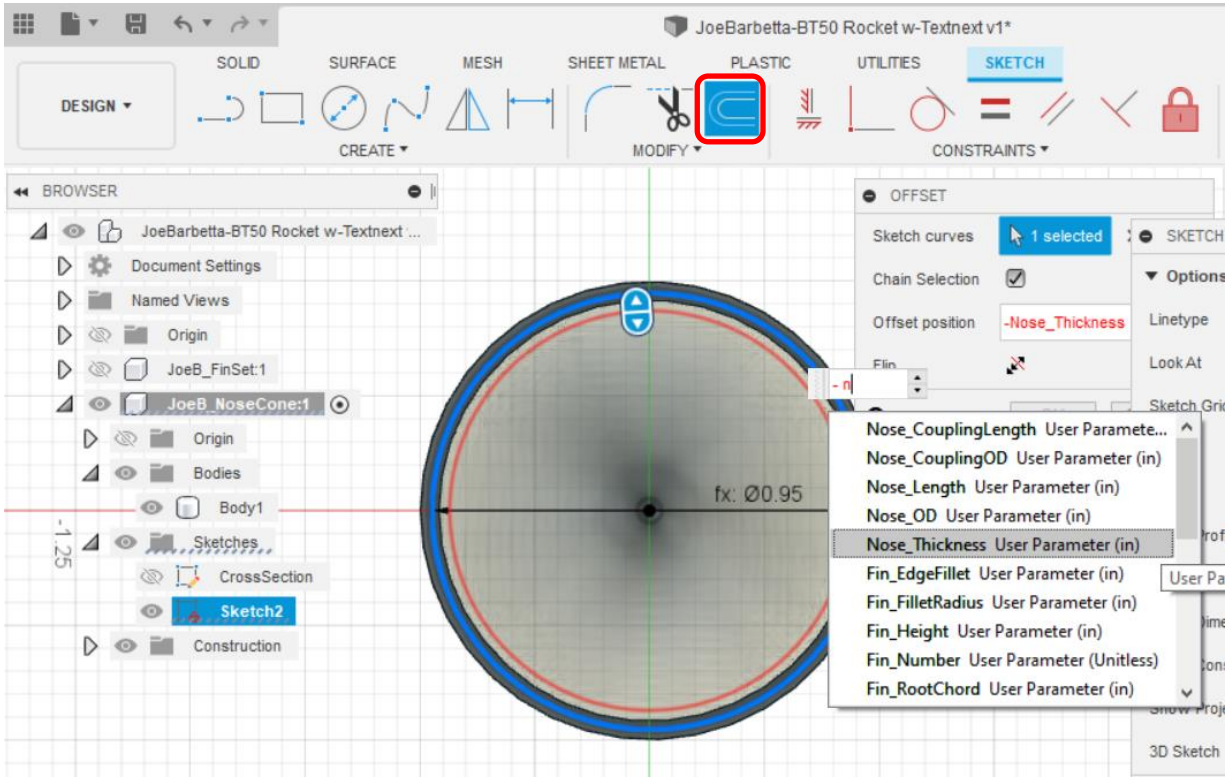
- hide the FinSet by clicking on the **eye** icon for the FinSet. One can use the eye icons to hide or unhide items in the BROWSER.
- rotate the view using either the **View Cube** or the bottom **Orbit** tool
- select the **Sketch** tool and click on the **bottom of the Nose cone**



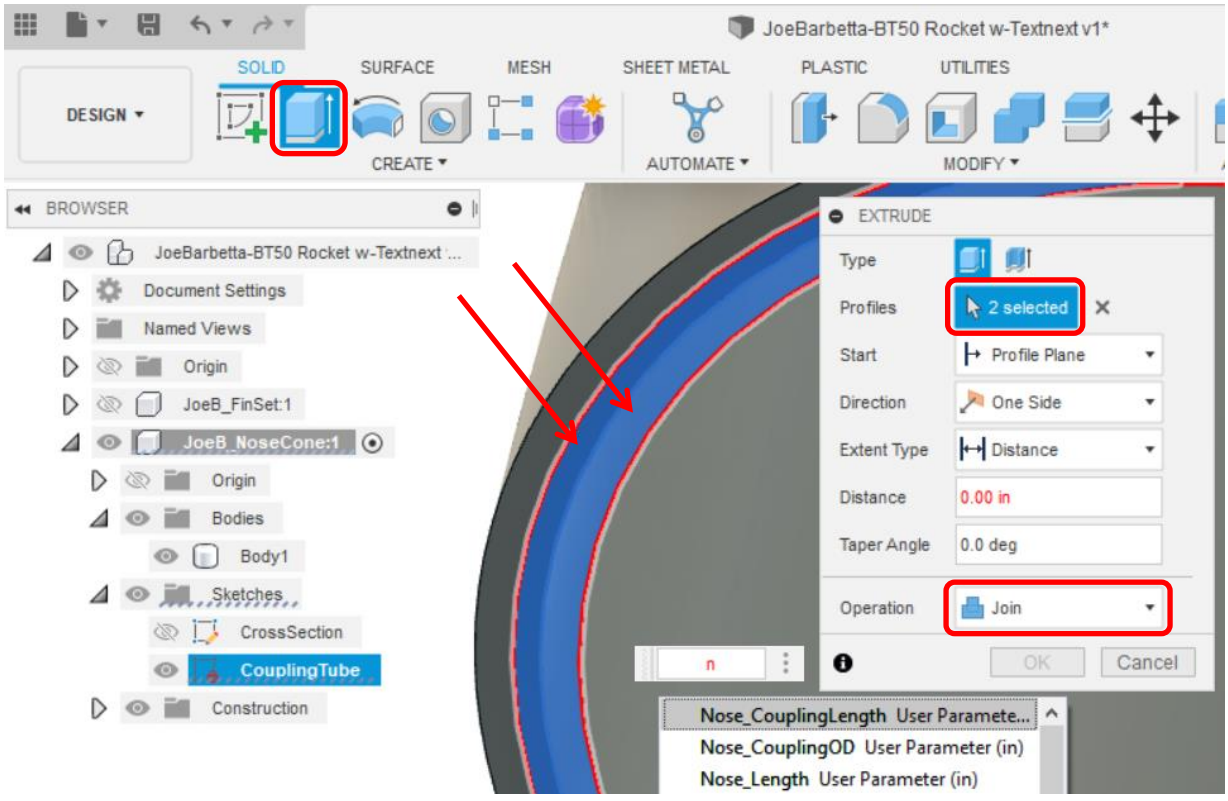
- select the **Circle** tool to create a circle from the center point. Type **n** and select **Nose_CouplingOD**



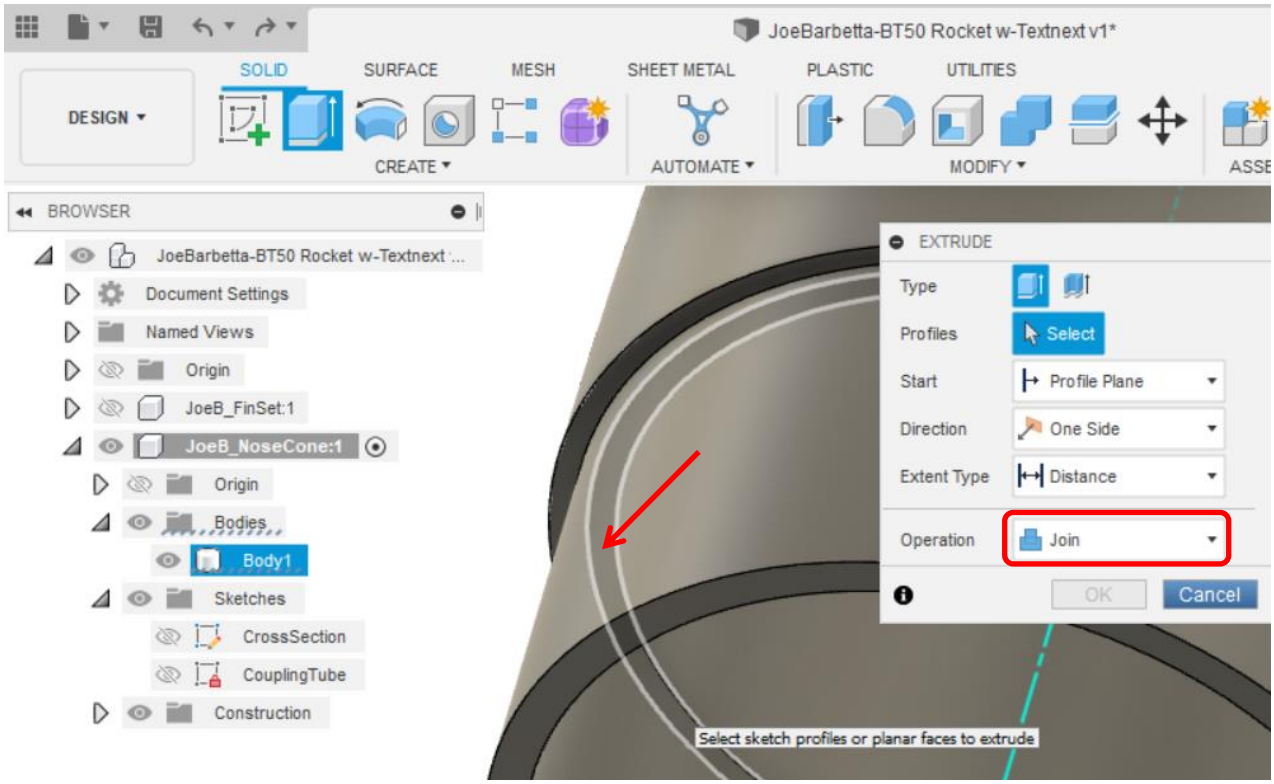
- select the Offset tool and type **- n** (note the negative before n) and select **Nose_Thickness**. The negative is needed to force the red arc to the inside of the blue circle.
- click **OK** in the **OFFSET** window and **Finish Sketch**



- select the **Extrude** tool and type **n** and select **Nose_CouplingLength**.
- as indicated by the two red arrows one must click on **both the inner and middle** ring areas.
- ensure **Join** is selected for the **Operation**
- type **n** and select **Nose_CouplingLength** and click **OK**



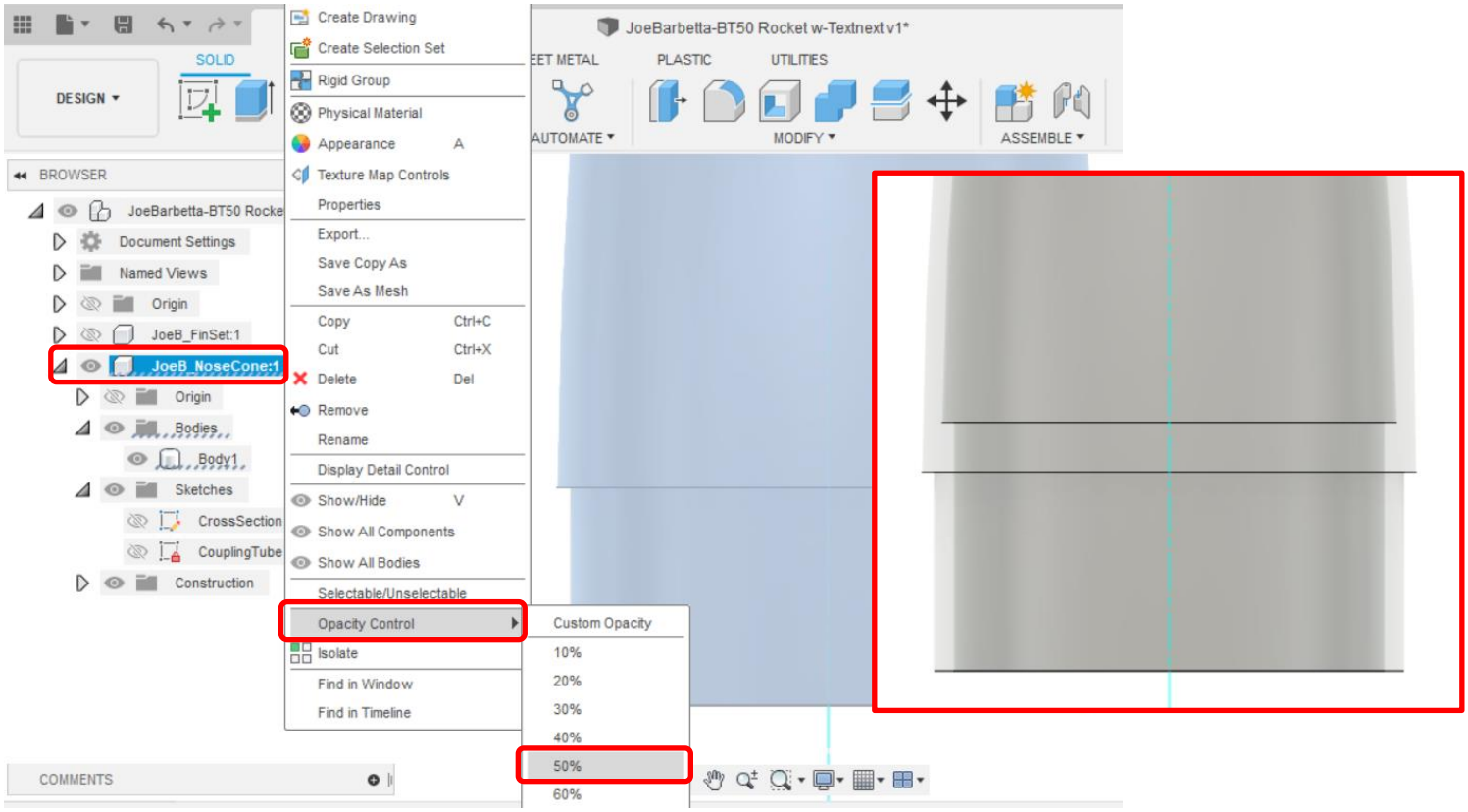
- select the **Extrude** tool again
- **click on the ring area** indicated by the red arrow to select the hidden top of the Coupling Tube.
- ensure **Join** is selected for the **Operation**
- type **0.1** and click **OK**



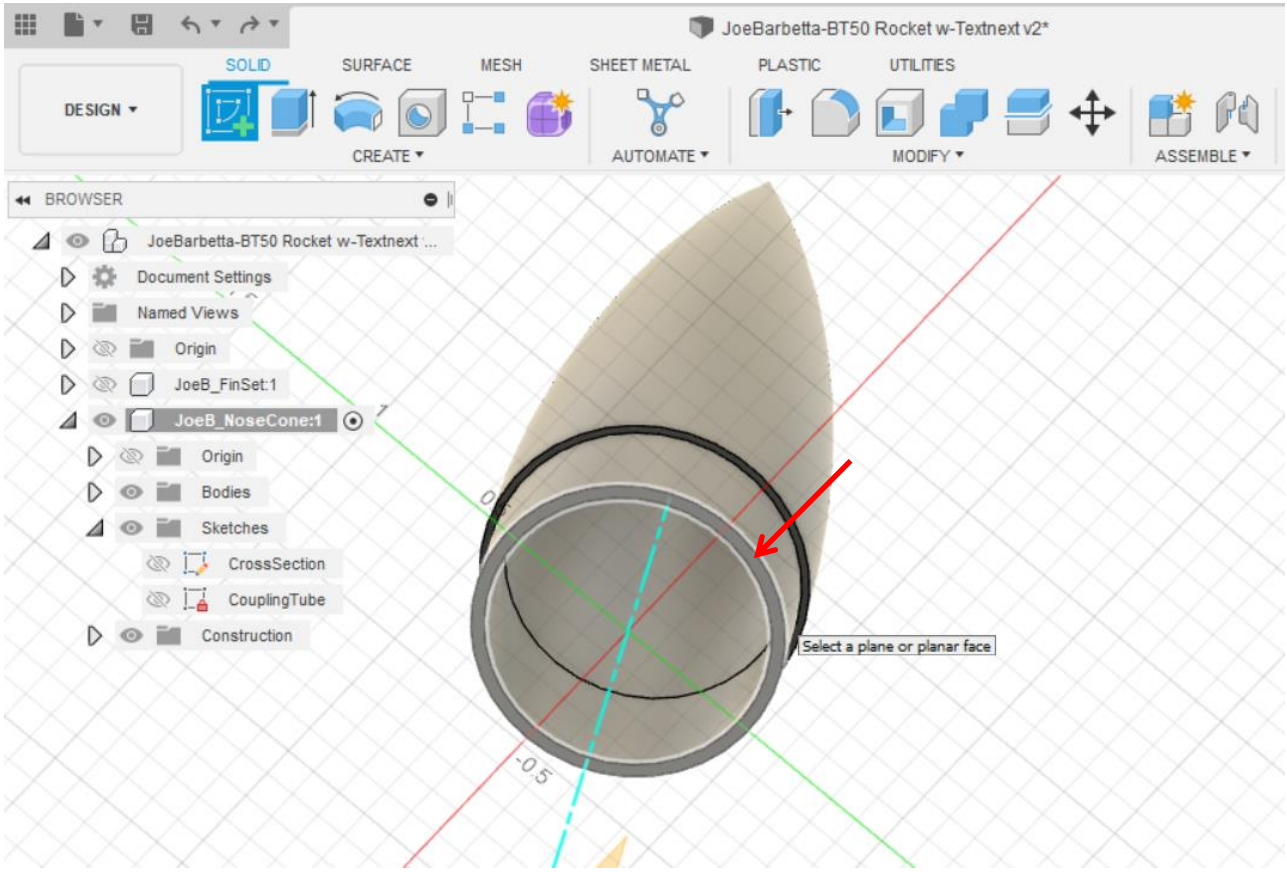
- right-click on the **Component Name** for the Nose Cone and select **Opacity Control** and **50%**

Note that the Nose Cone is now transparent as shown on the right.

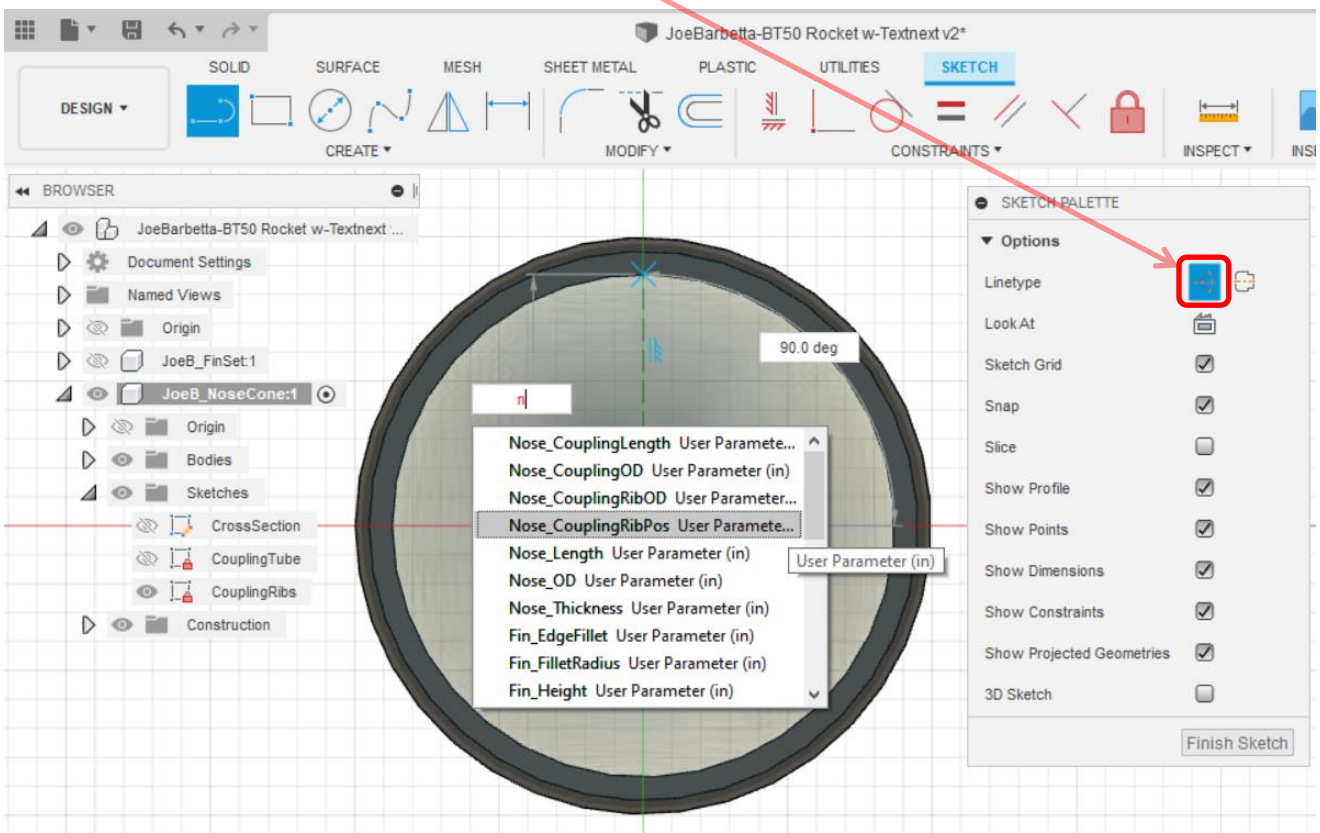
- right-click on the **Component Name** again and change **Opacity Control** back to **100%**



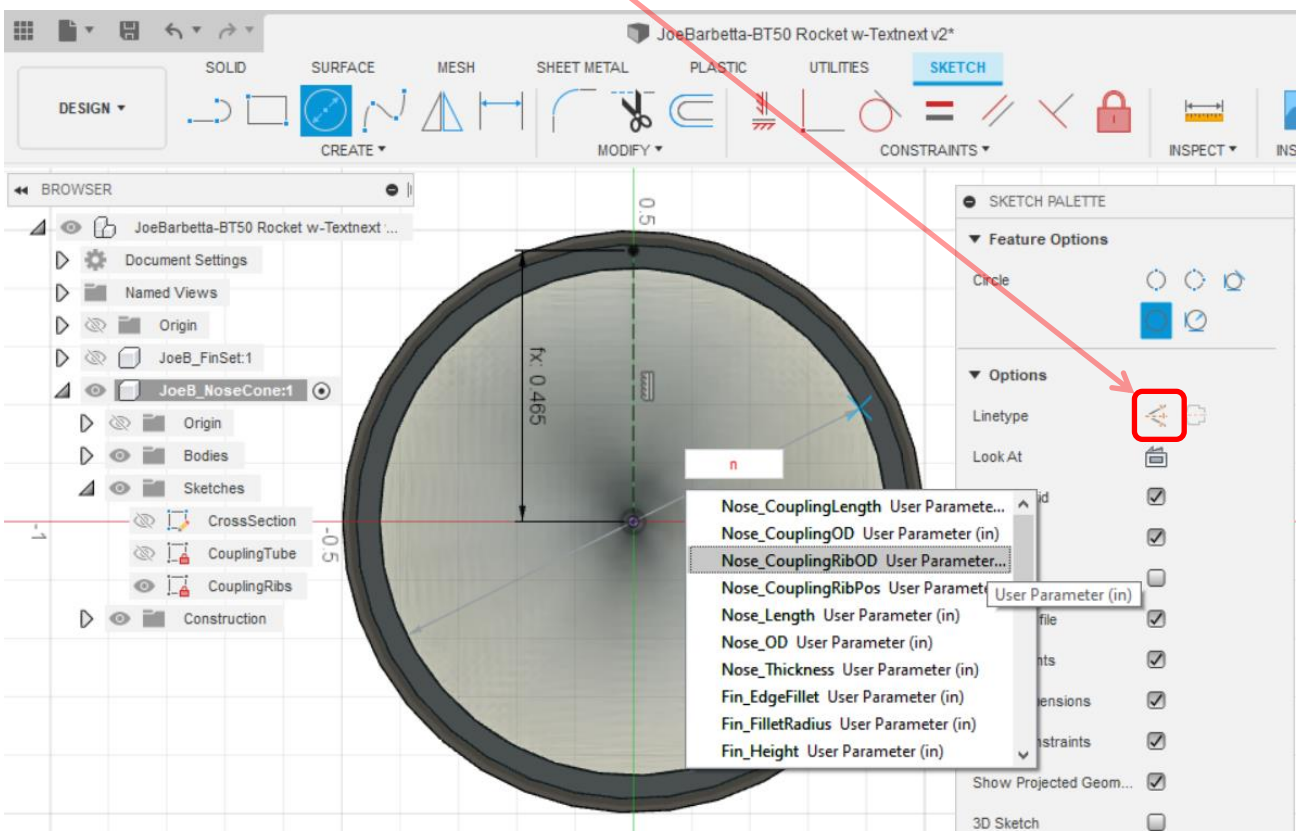
As we did for the Fin Set, we should add ribs that can easily be sanded or scraped to ensure the best fit in the body tube.
- select the Sketch tool and click on the bottom of the coupling.



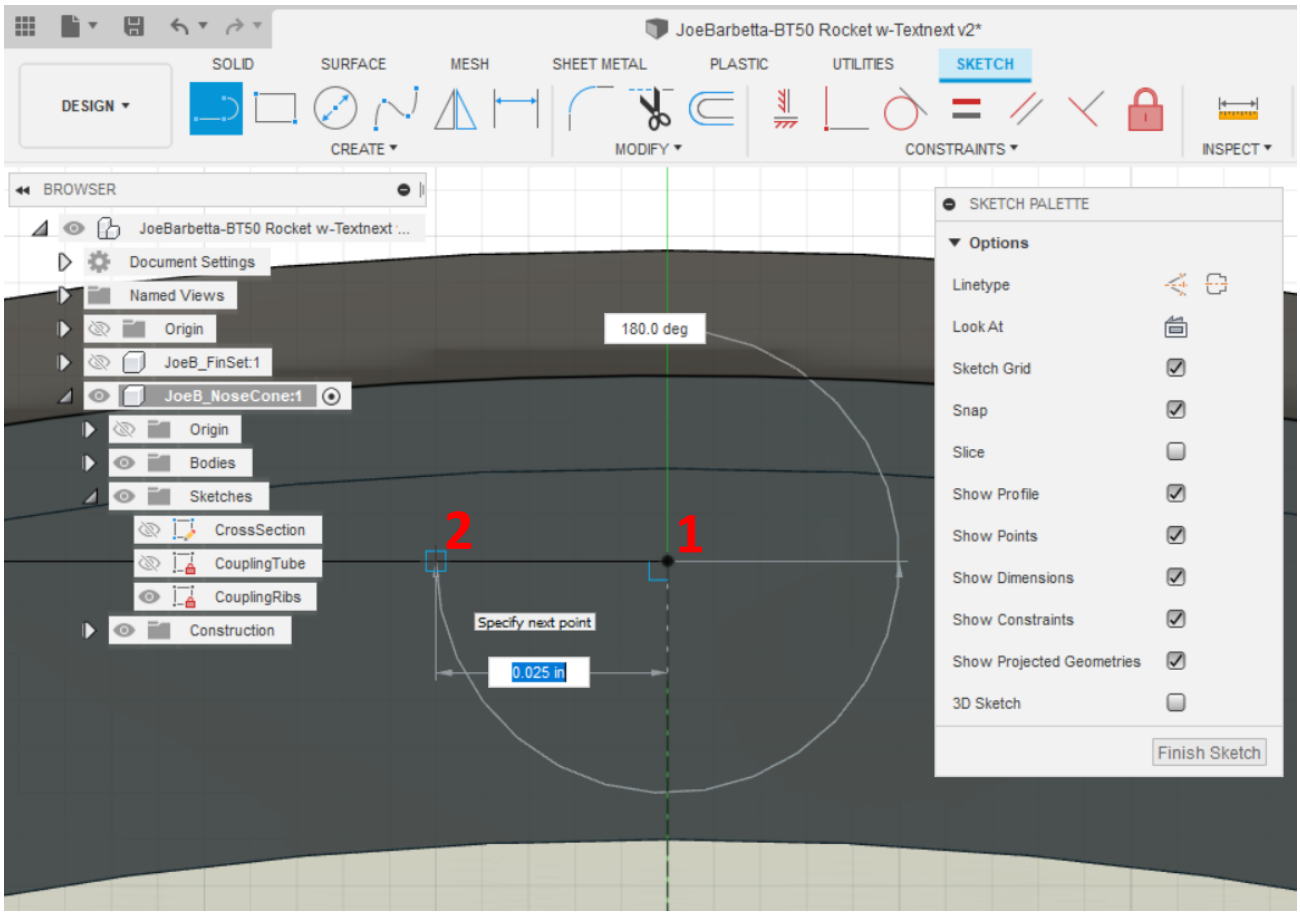
- **zoom in** as shown below
- click on the Construction **icon next to LineType** to turn it blue
- create a line starting at the center and up towards the edge. Type **n** and select **Nose_CouplingRibPos**



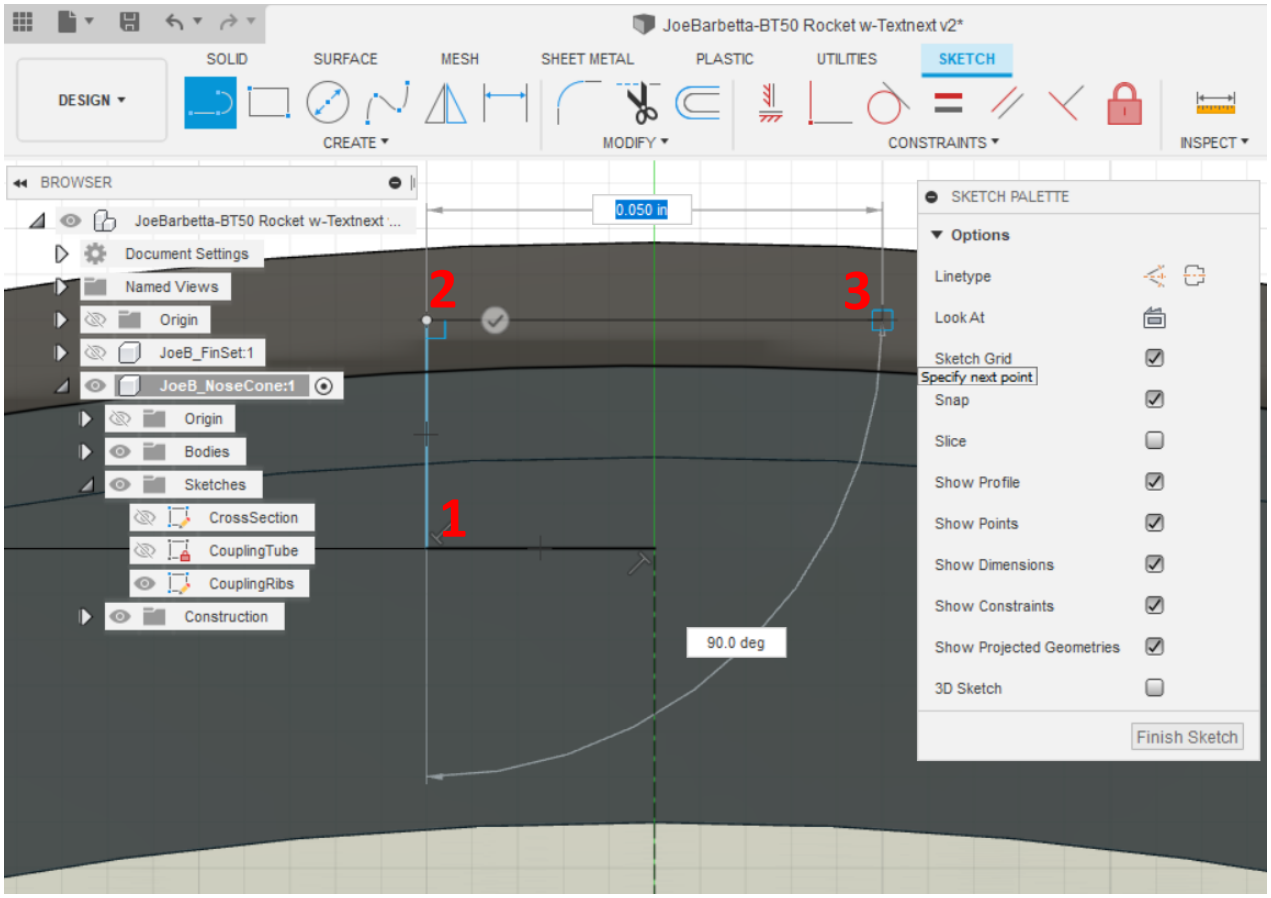
- click on the Construction **icon next to LineType** to remove the blue highlighting to create normal lines
- create a **Circle** from the center and type **n** to select **Nose_CouplingRibOD**



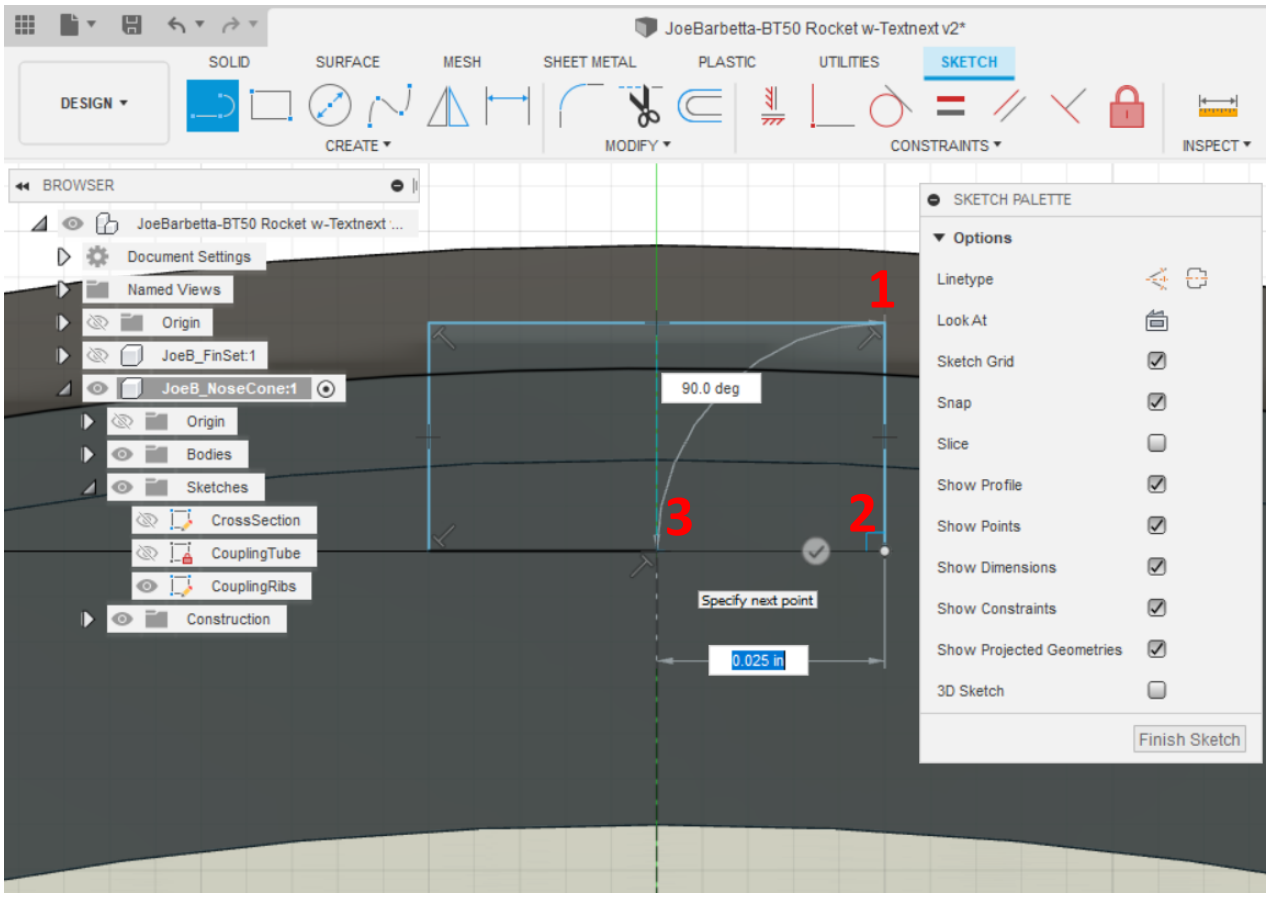
- create a **Line** from the **top of the Construction Line** at **1** to the left near **2** and enter **0.025**



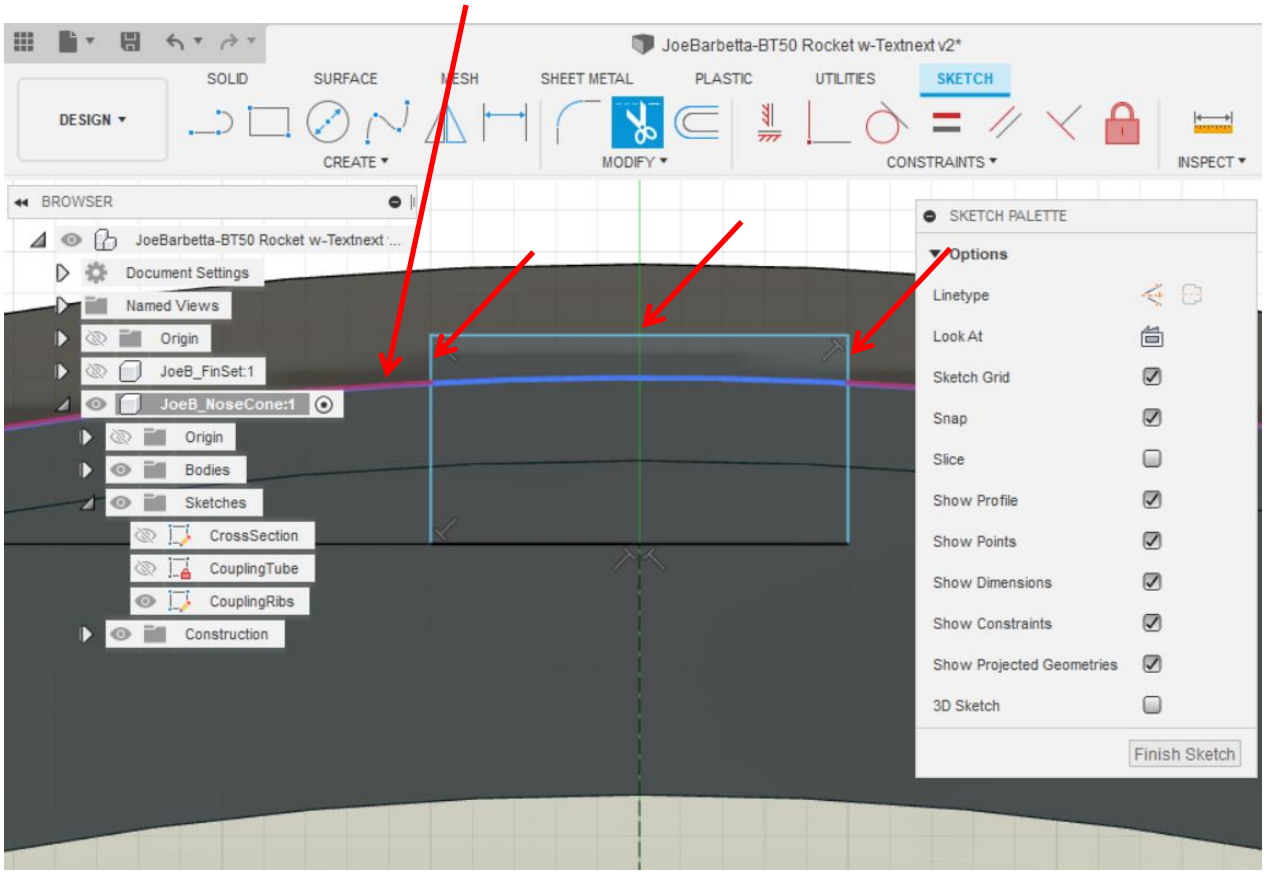
- create a **Line** from the last point at **1** up by **0.025** to **2** and then to the right by **0.050** to end at **3**



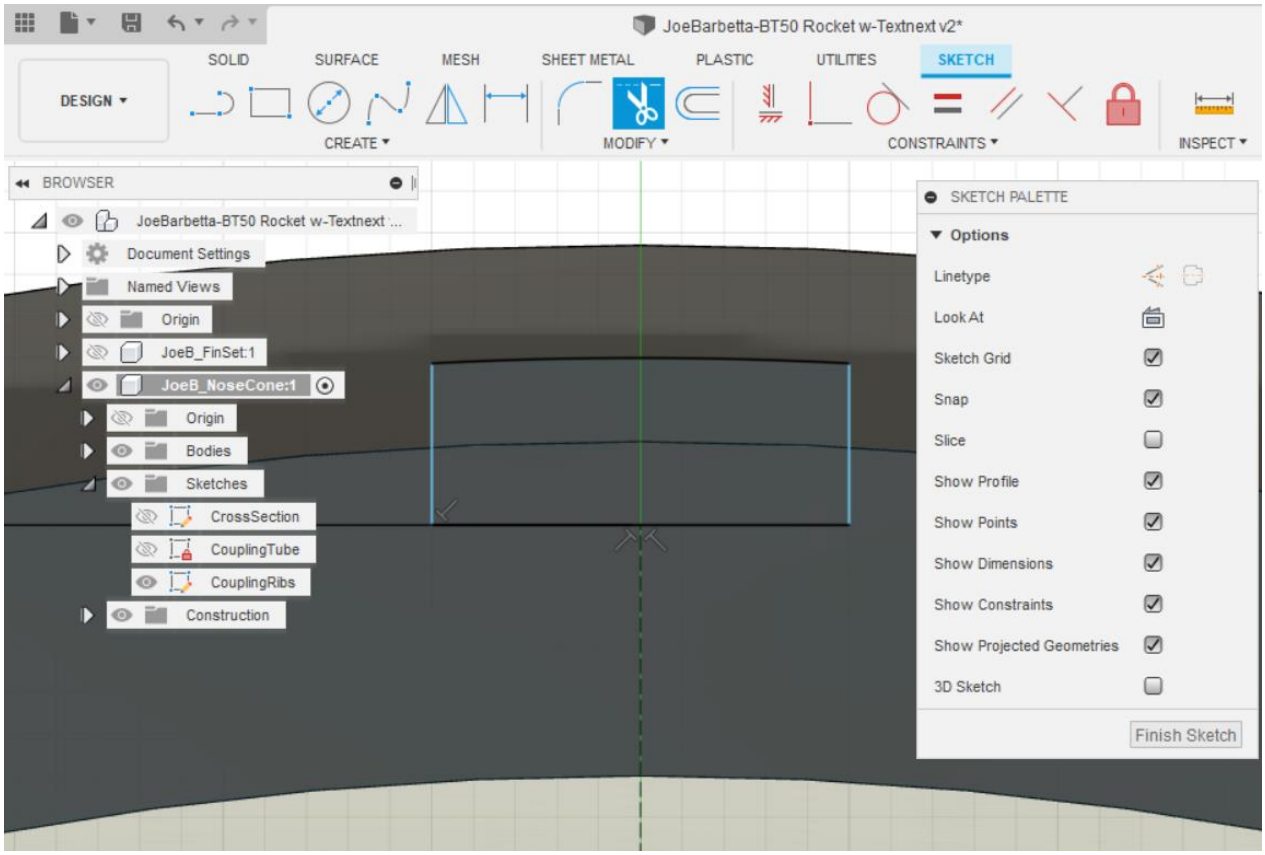
- create a **Line** from the last point at **1** down by **0.025** to **2** and then to the left by **0.050** to end at **3**



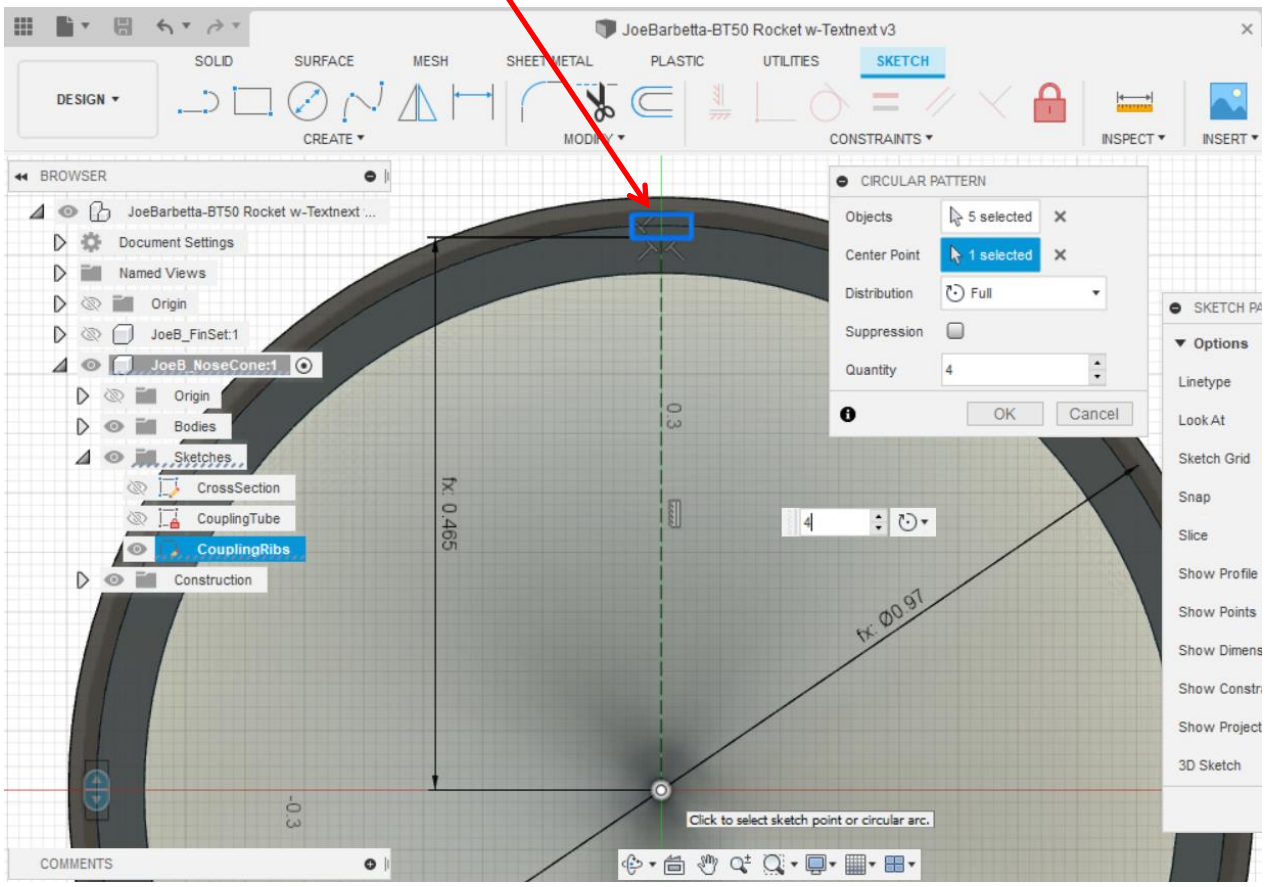
- select the **Trim** tool and click on the **arc** outside the rectangle and the **3 top segments** of the rectangle.



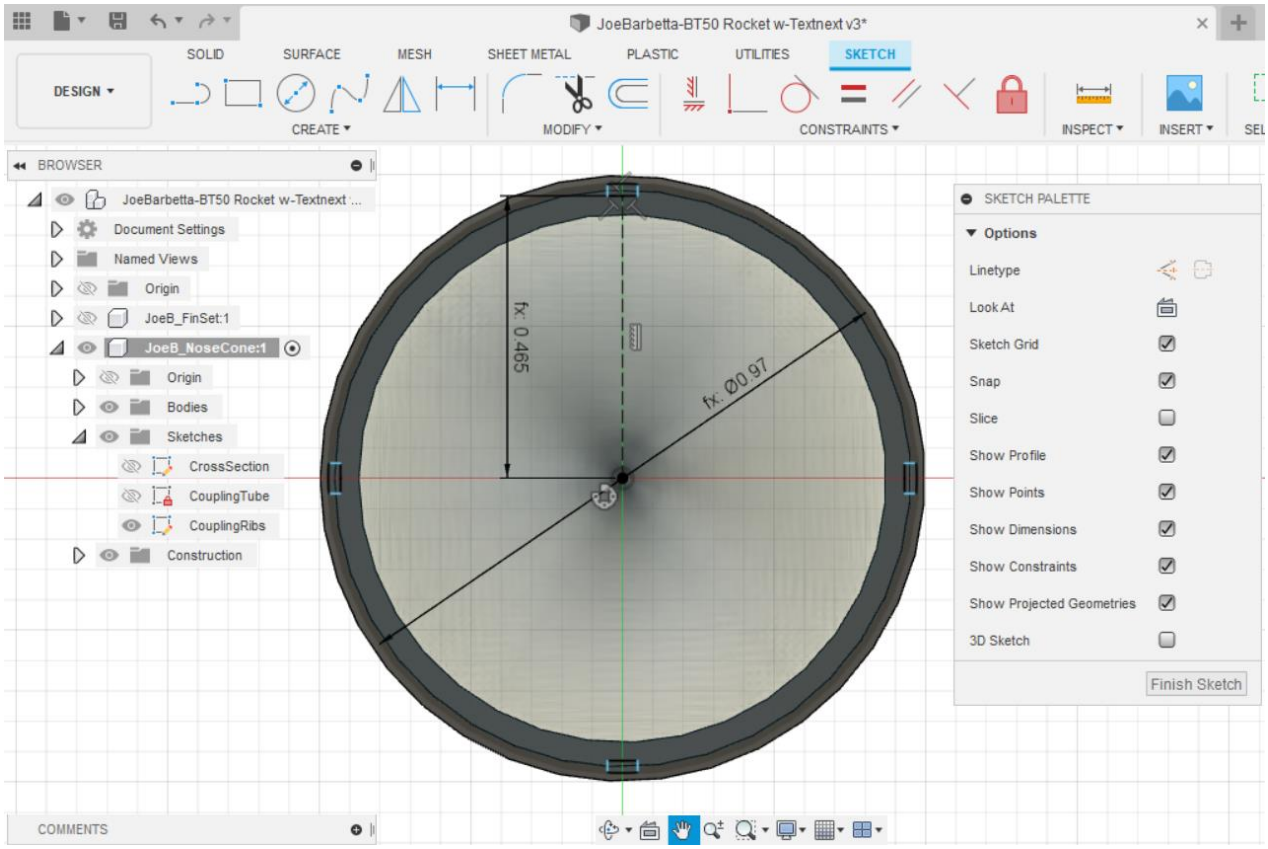
There should be a rectangle with a curved top as shown below.



- **zoom out** and click on **CREATE** and select **Circular Pattern** near the bottom of the list.
- **double-click on any edge of the rectangle**. This should cause each edge to turn blue.
- enter **4** for the Quantity and click **OK**.

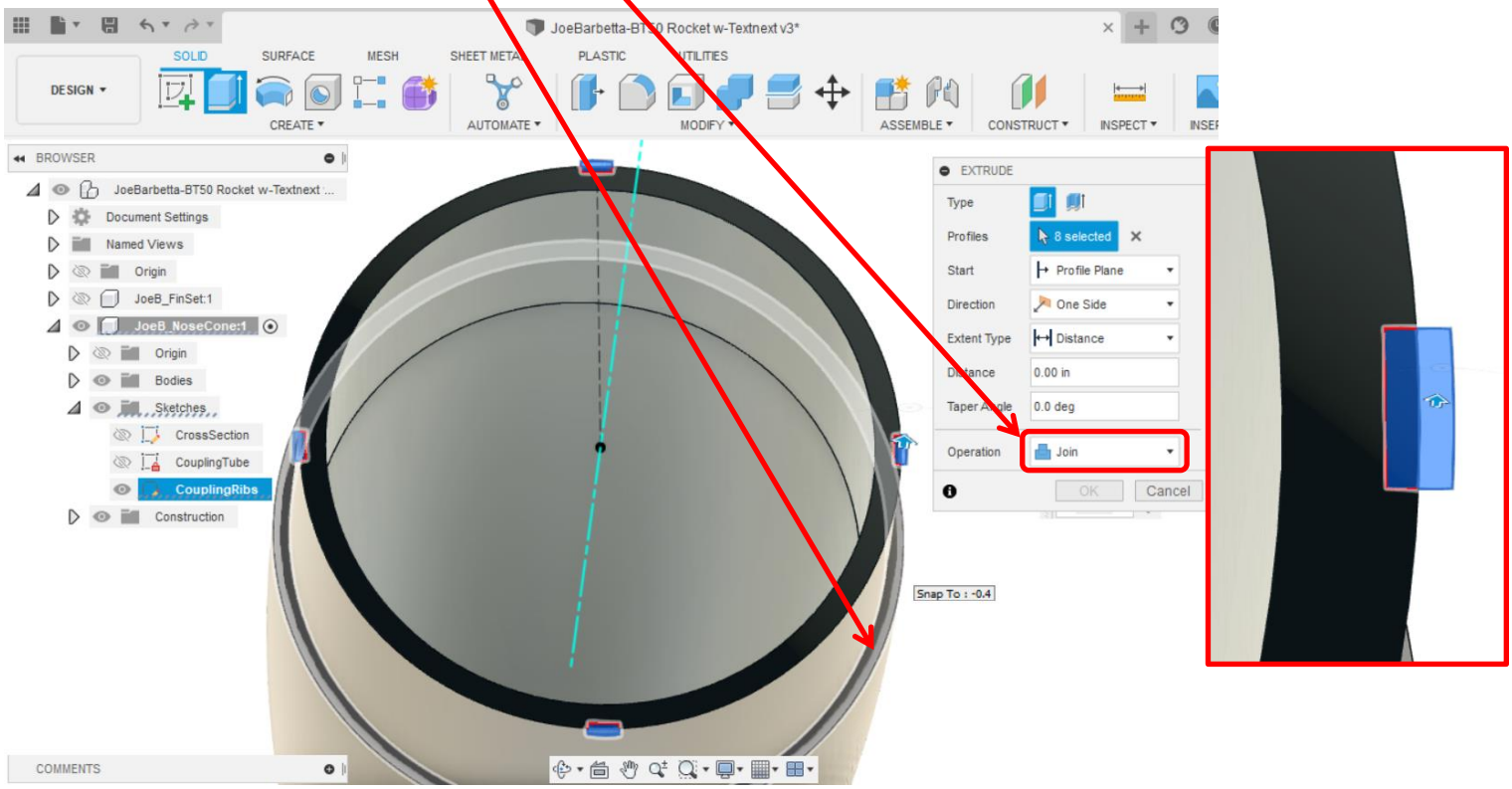


When zoomed out the 4 rectangles should be visible. Click **Finish Sketch**.

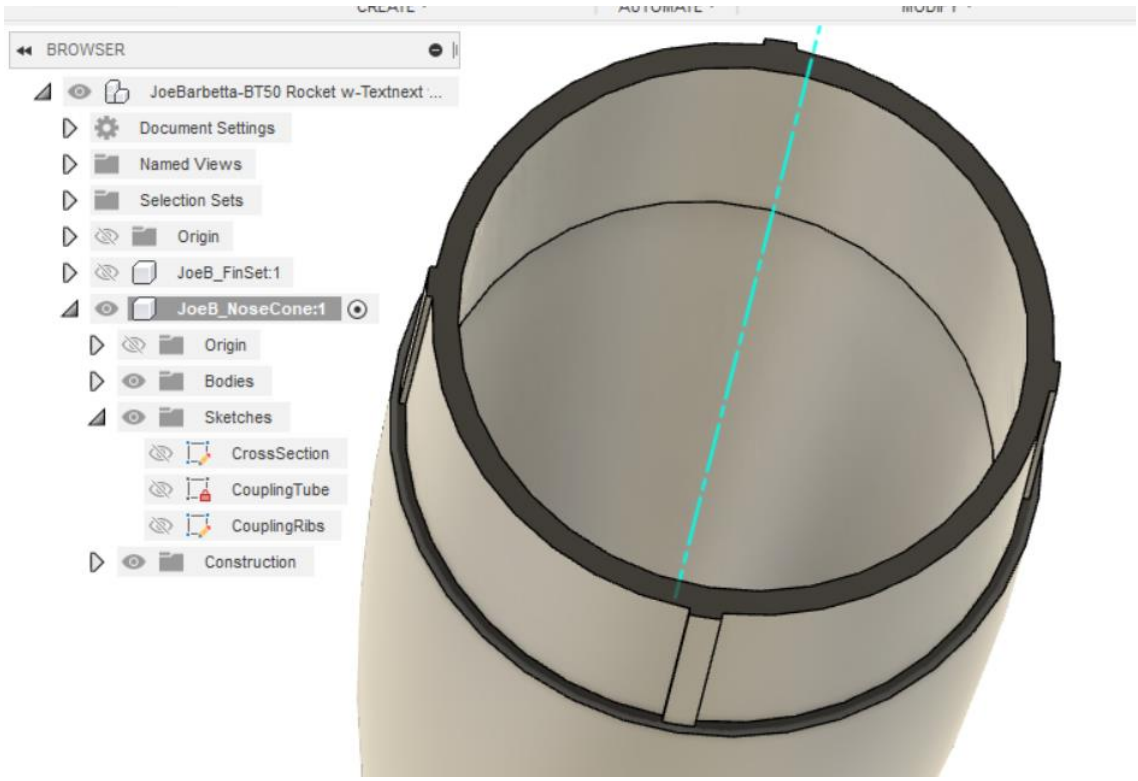


Here is another pain-in-the-butt operation that may require zooming in and out.

- **click on the two regions** making up **each of the 4 rectangles** and thus 8 regions will be clicked on in total. The closeup on the right shows how each rectangle should look.
- ensure that **Join** is selected for the **Operation**.
- click on the somewhere on the **annular face** where the coupling tube meets the nose cone. Ribs will appear along the coupling.

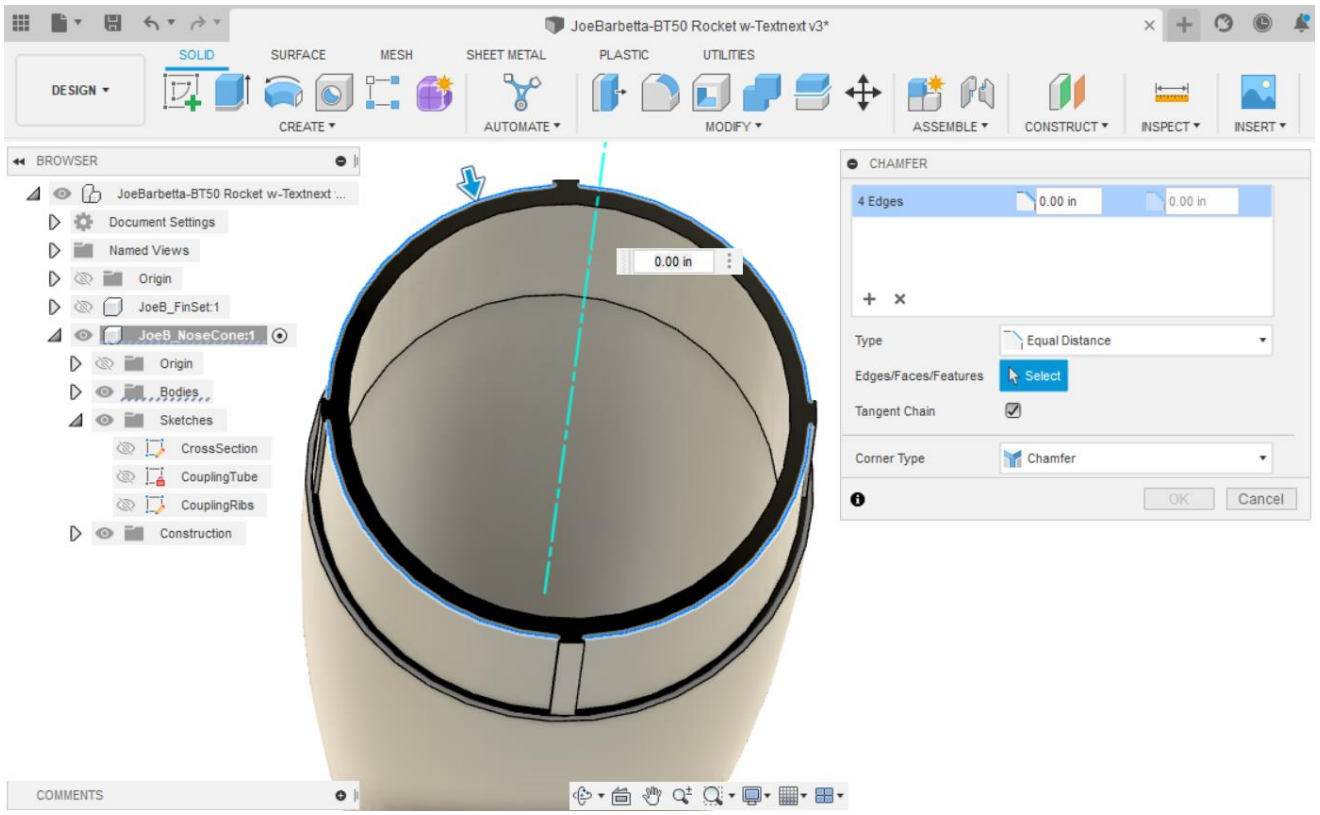


- click OK and the result should appear as below.

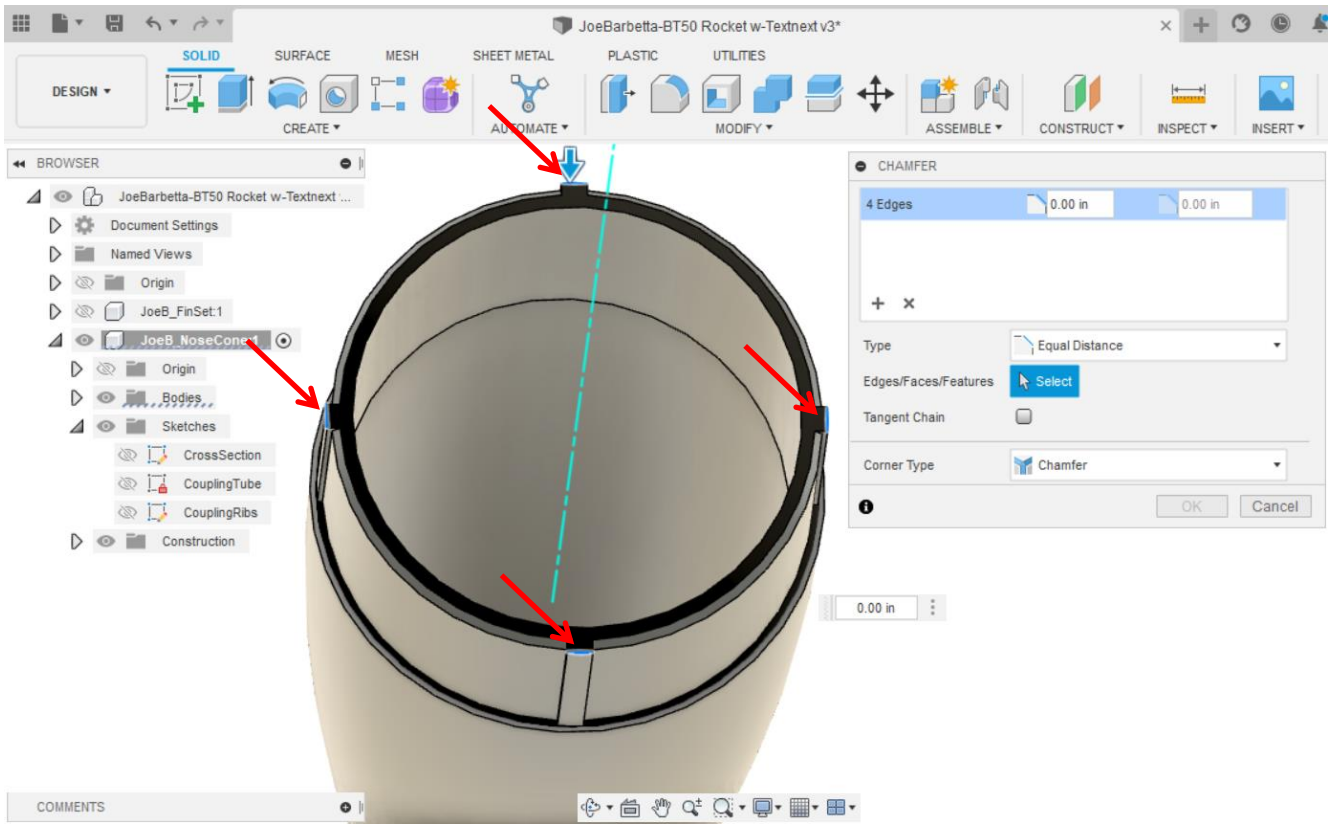


Now some chamfers will be added to make it easier to insert the nose cone in the body tube.

- click on **each of the 4 outer edges**.
- enter **0.015** and click **OK**. If an error appears, try smaller values.

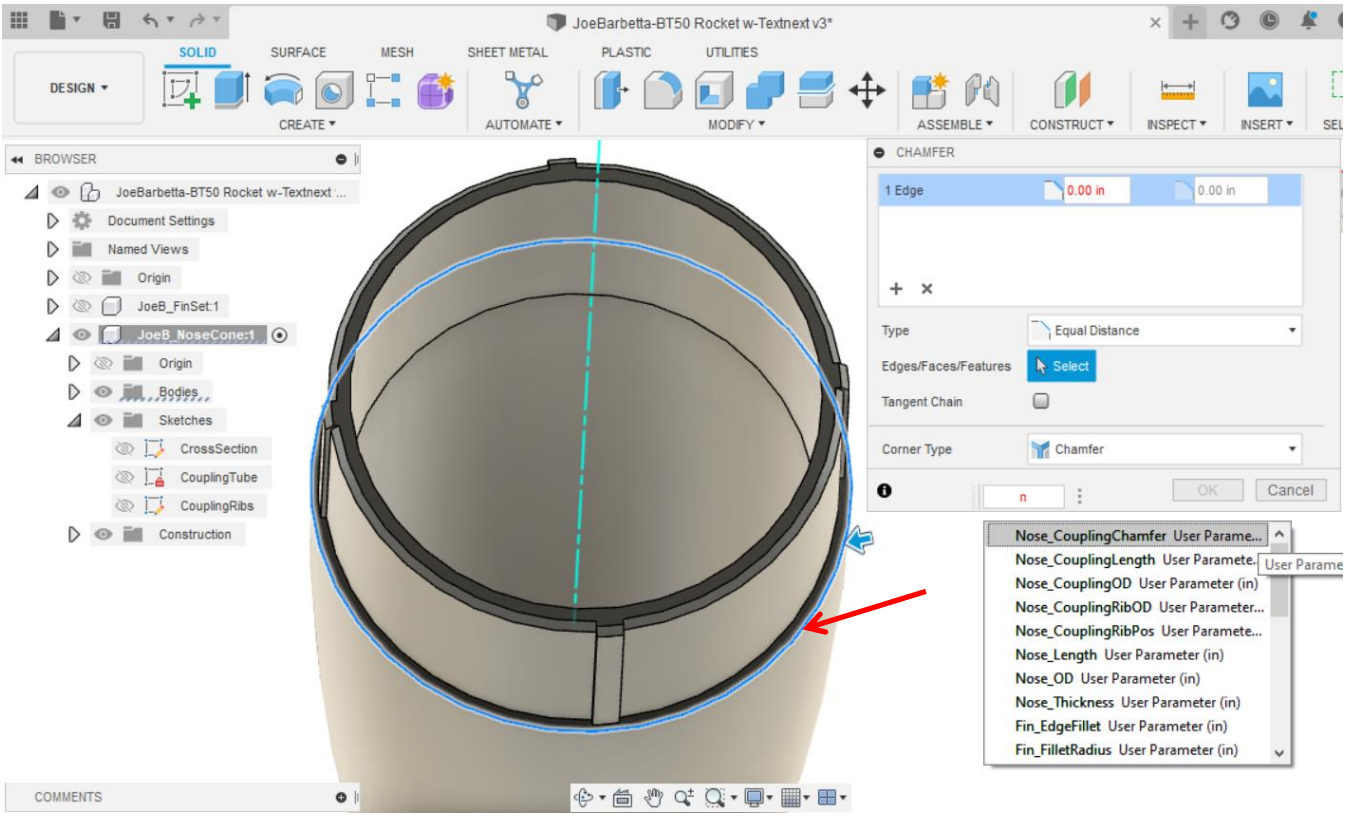


- click on the top **outer edge of the 4 ribs**.
- enter **0.015** and click **OK**. If an error appears, try smaller values.

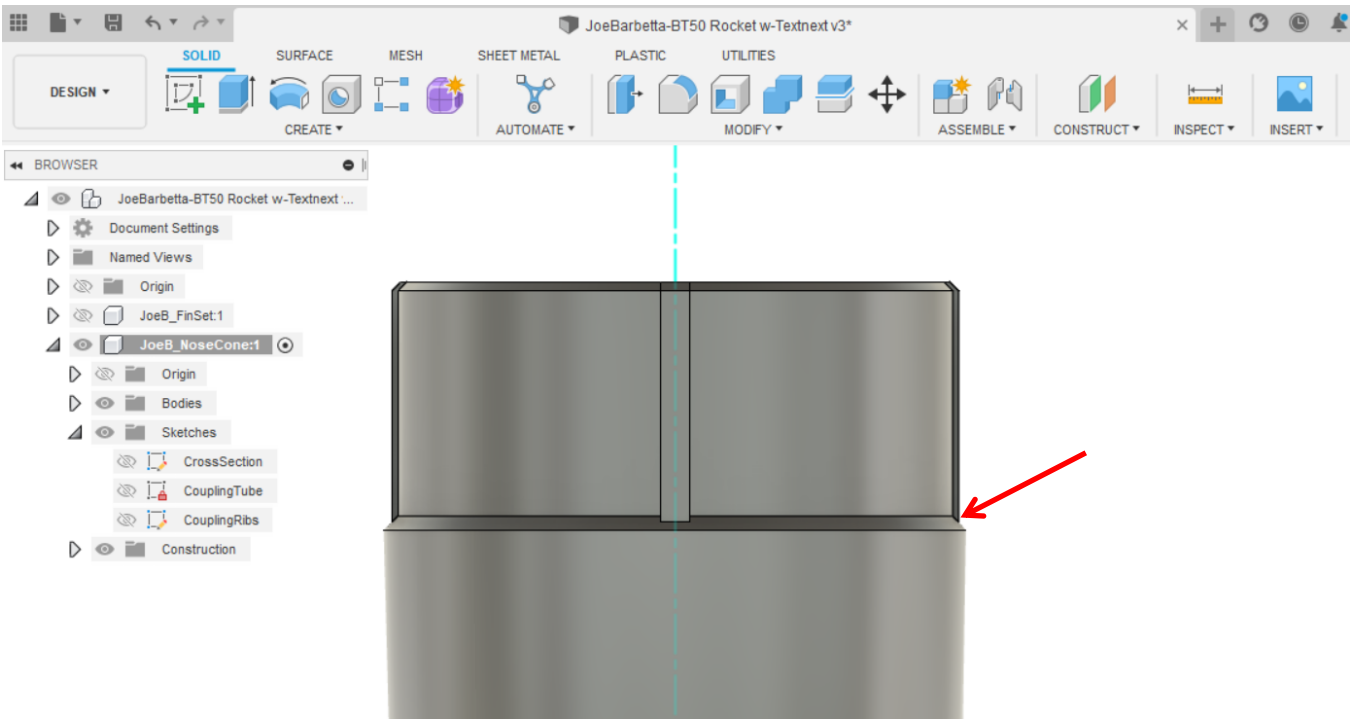


Because the nose cone will be printed when flipped 180 degrees compared to below it will start printing from what is now the top surface. When the print reaches the layer where the coupling tube meets the nose cone it will encounter an overhang. Adding a chamfer will allow adherence to the 45 degree rule.

- click on the **outer edge** as shown. Type **n** and select **Nose_CouplingChamfer** and click **OK**.

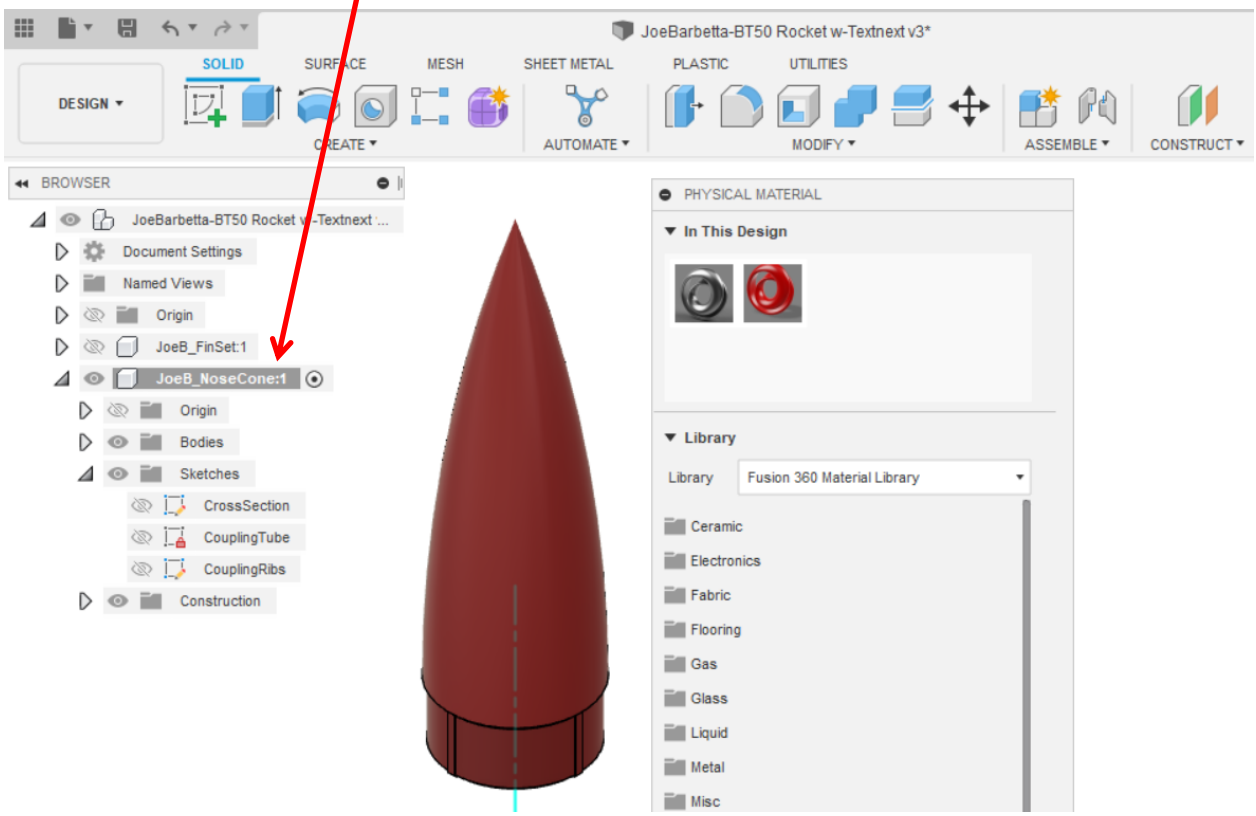


This is now how the side view looks. Note the 45 degree chamfer.



- right-click on the *Component* name and select **Physical Material** and **drag the red icon onto the nose cone**.

If you want a different color, you can first right-click on the icon and select Duplicate and then right-click on the new icon and select Edit. The instructions in "Setting Materials and Colors" can be followed to pick a new color. This new icon can then be dragged onto the nose cone.

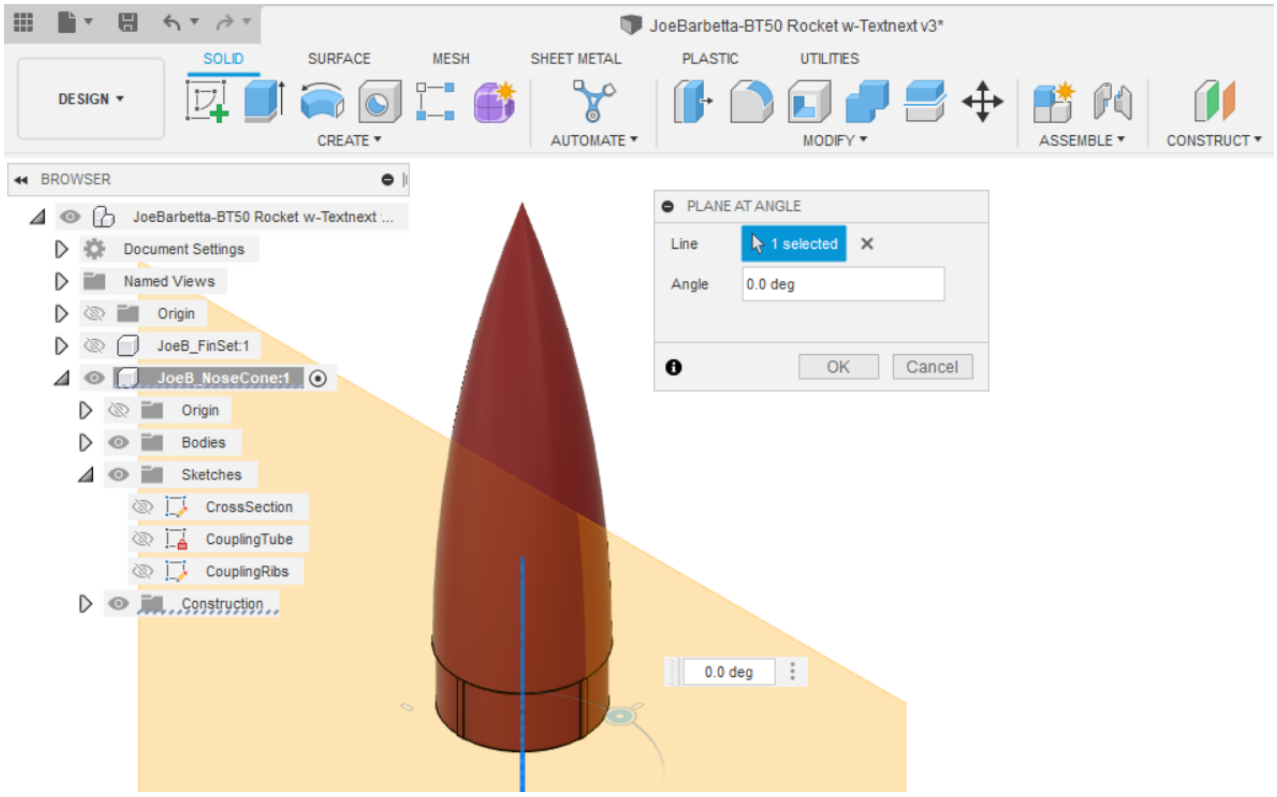


Adding Text to a Curved Surface

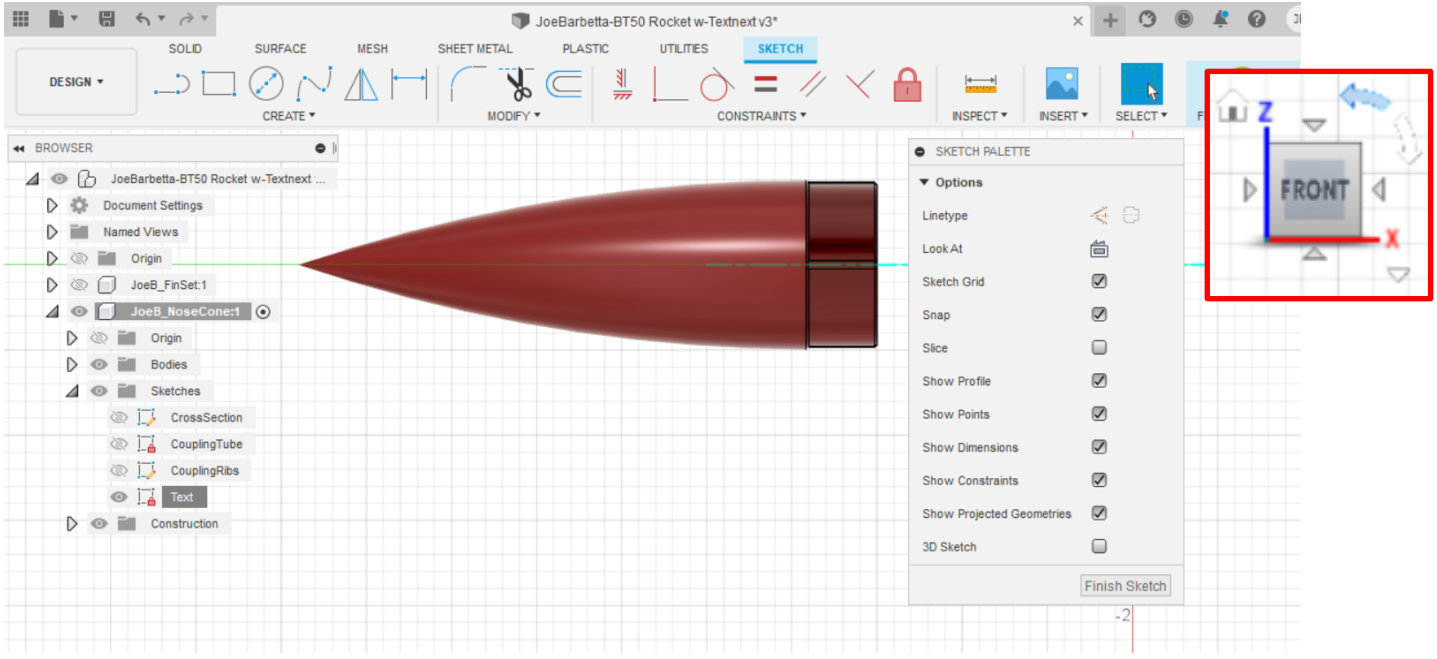
One can only select a flat surface of an object to start a Sketch and if it is desired to add text to a curved surface a new *Construction Plane* must be created.

Our nose cone already has an axis, but if it didn't one would use the Axis Through Cylinder/Cone/Torus under CONSTRUCT to create one. We will use the one we have already.

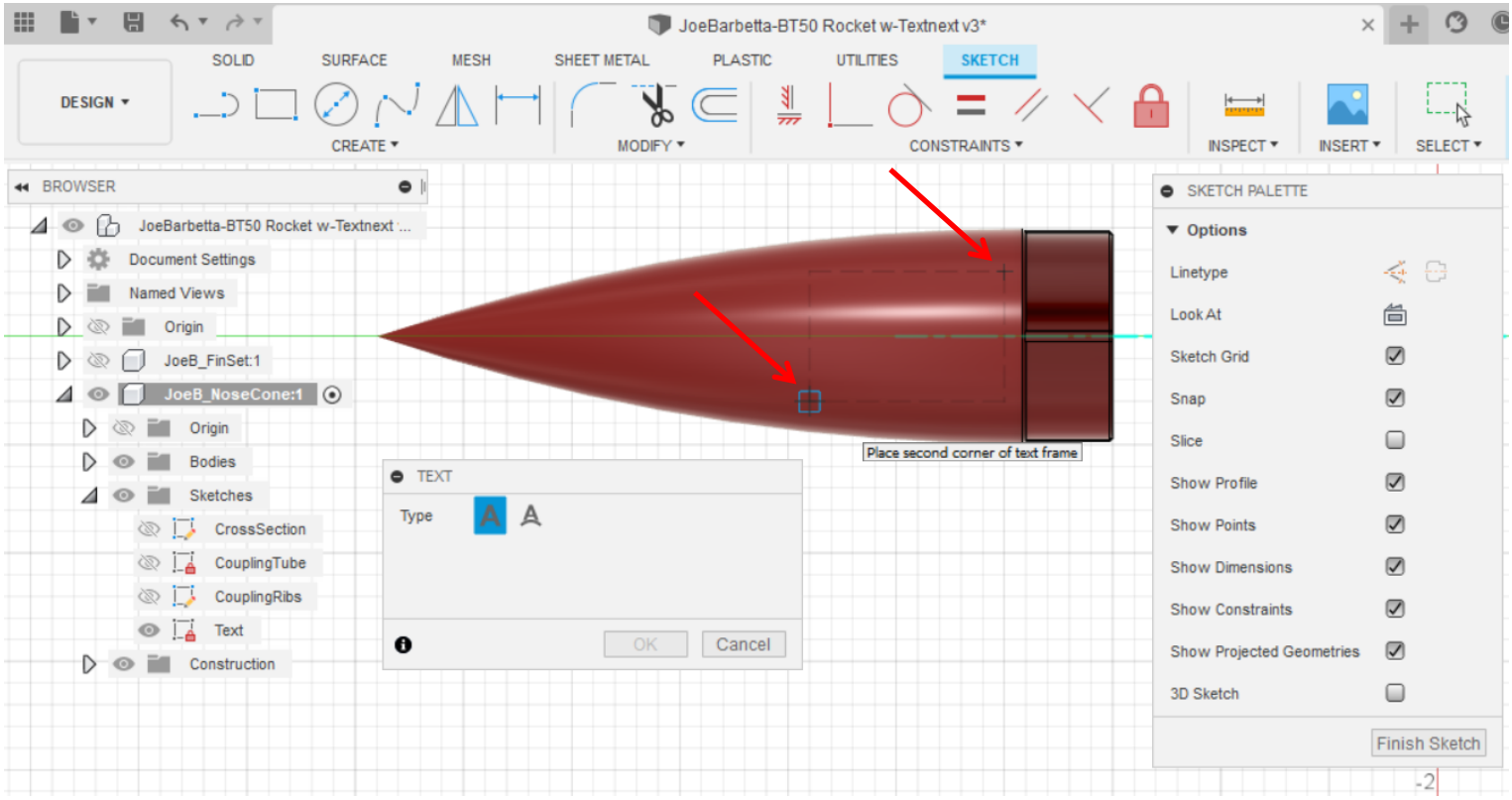
- select **Plane at Angle** under **CONSTRUCT** and click on the **blue axis**. We can keep the default value of 0.0 deg and click **OK**. A yellow *Plane* should appear as below.
- select **Sketch** and click on the **yellow Plane**



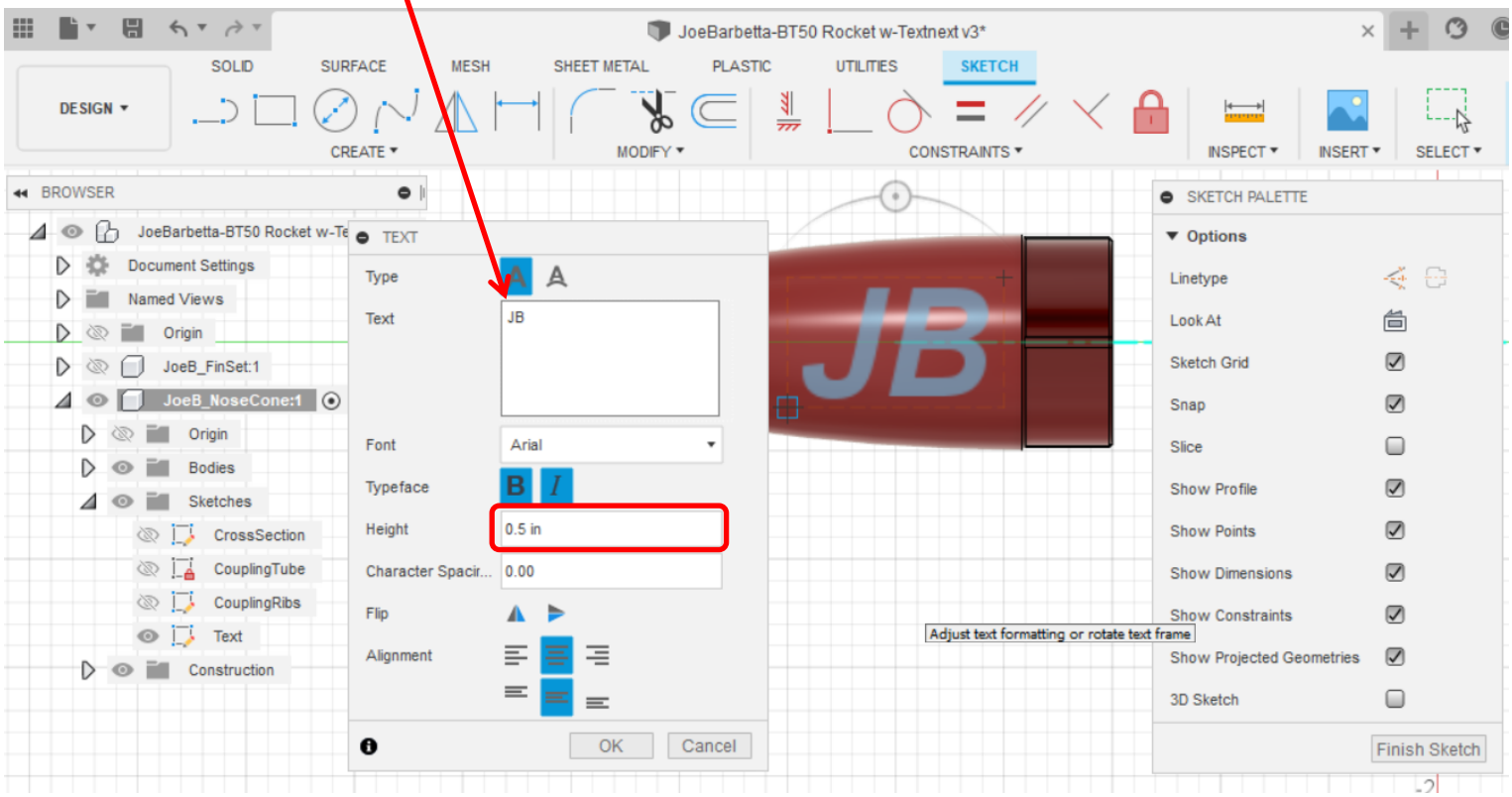
If text orientation along the axis is desired, click on a blue curved arrow that appears when hovering over the View Cube.



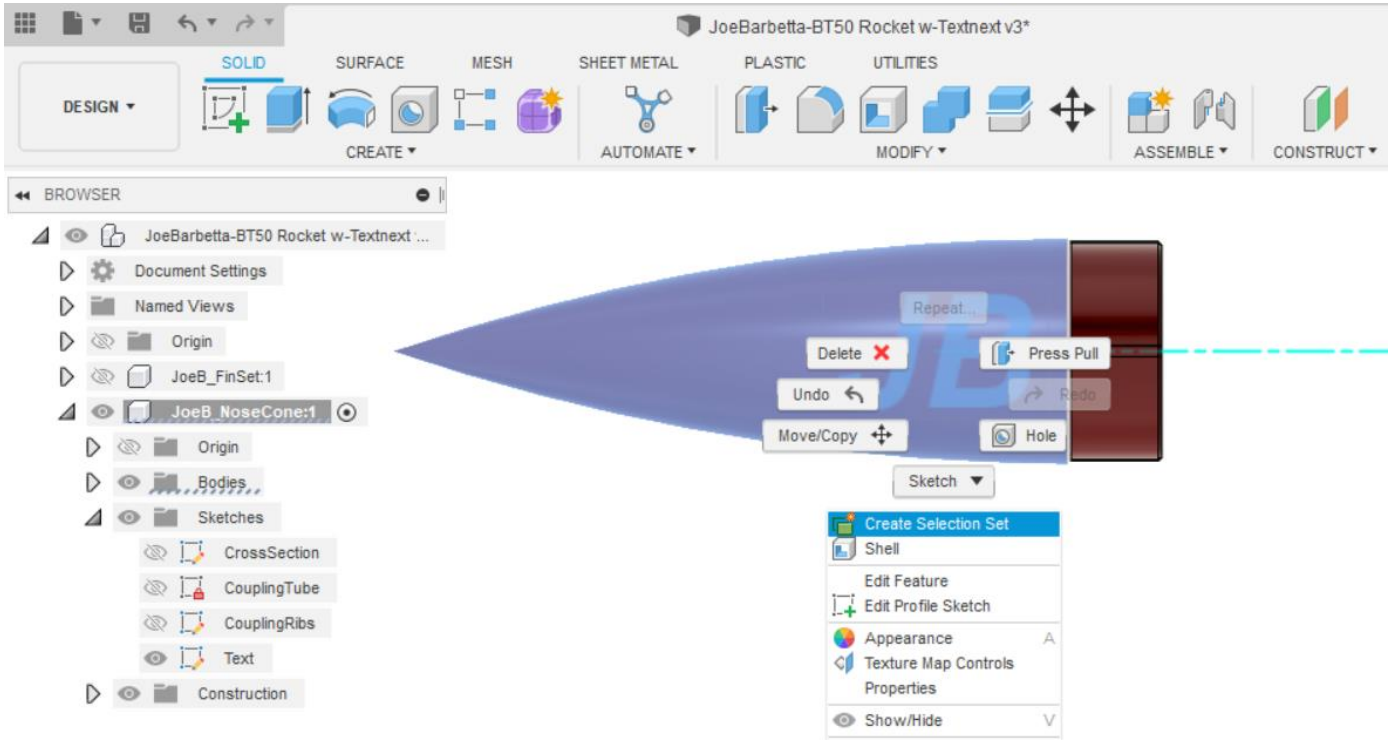
- select **Text** in the **CREATE** menu and specify a rectangular region for the text by **clicking on the locations for two opposite corners**.



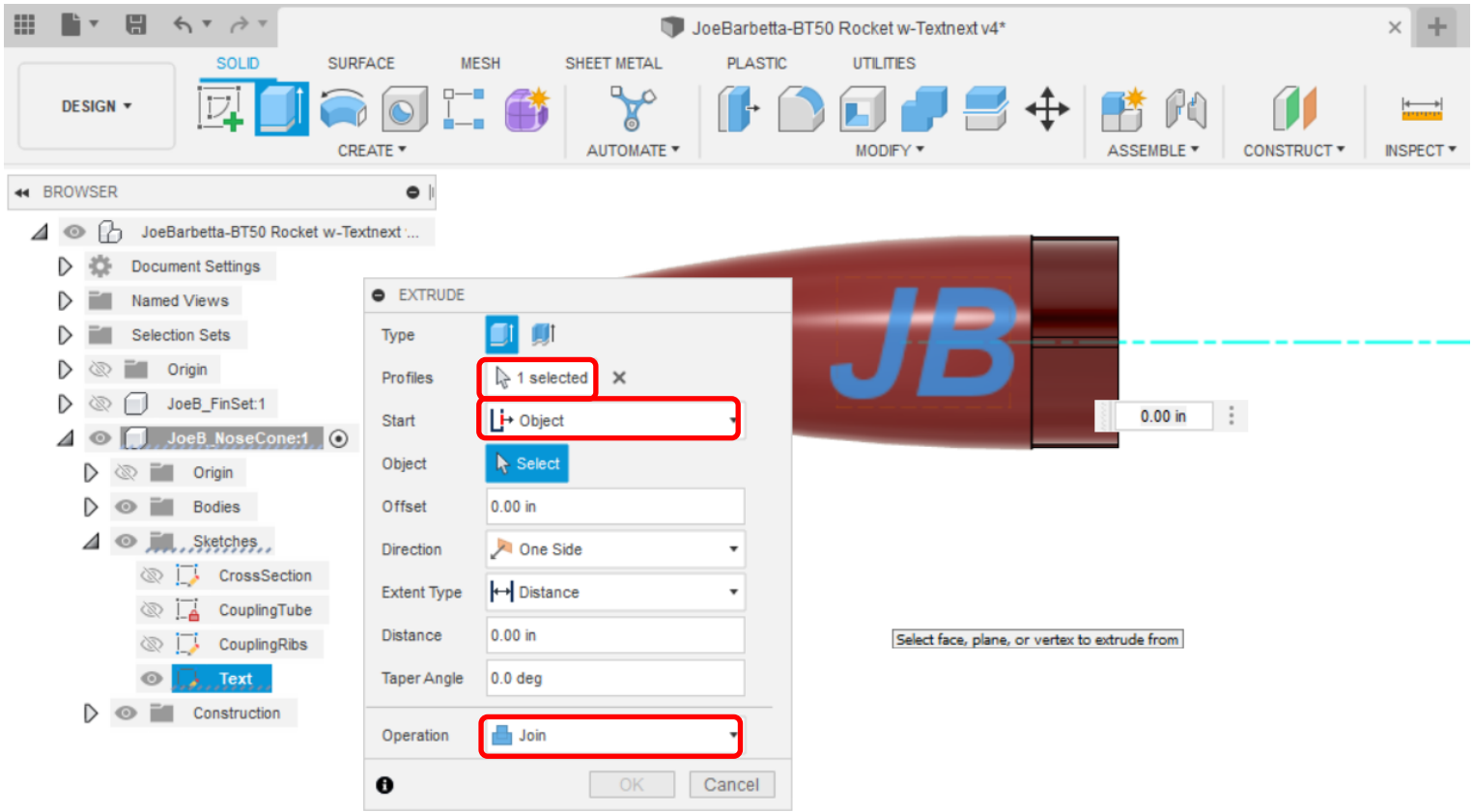
- enter the desired text in **Text**. If the text is too large or too small change the **Height** until the desired size is attained.
- one can also change the Font and make the **Typeface bold** or **italic**
- one can also click on **Alignment** options to center the text horizontally and vertically
- the text can also be mirrored by using a **Flip** option
- click on **OK** and then **Finish Sketch**



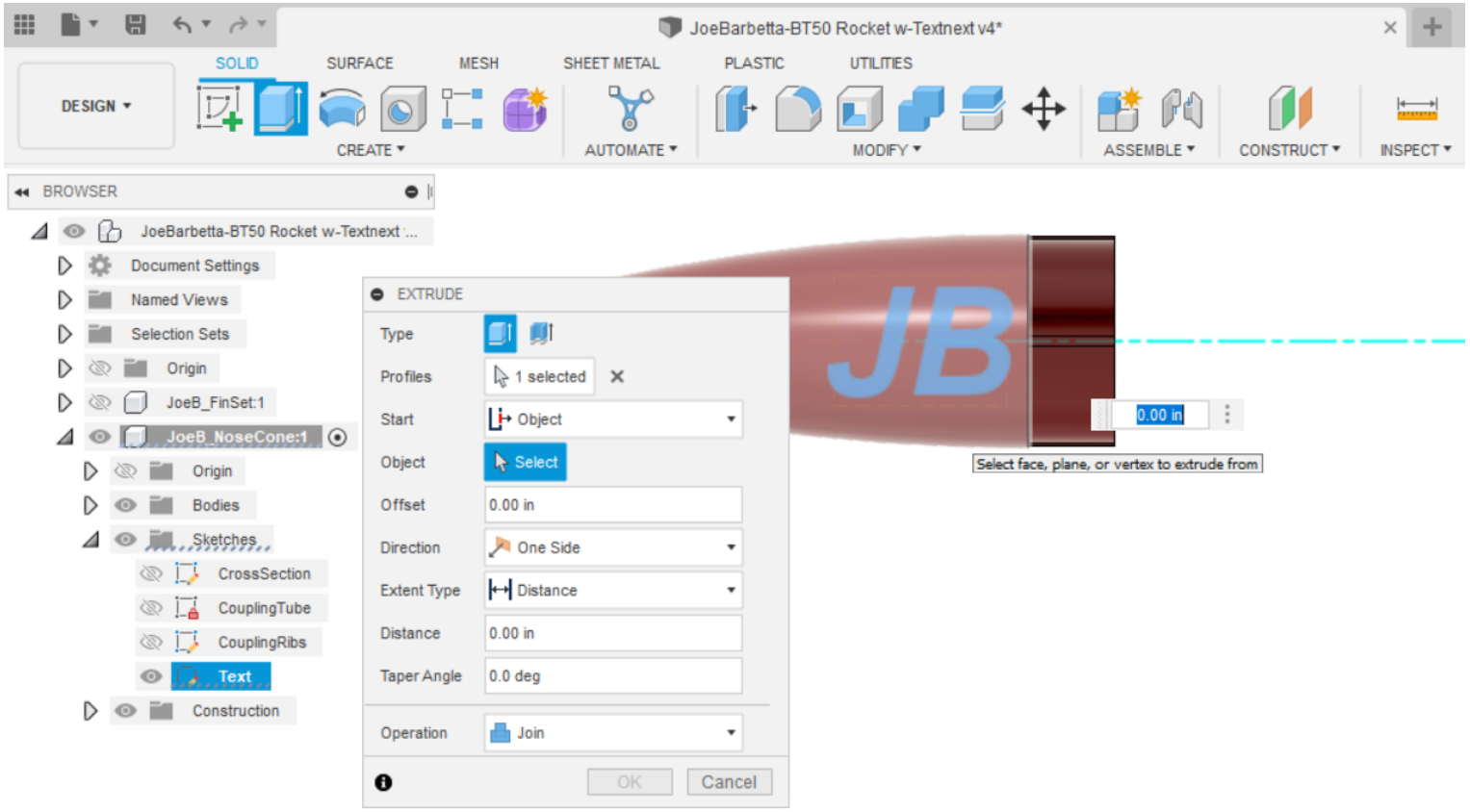
- right-click on the text and select **Create Selection Set**.



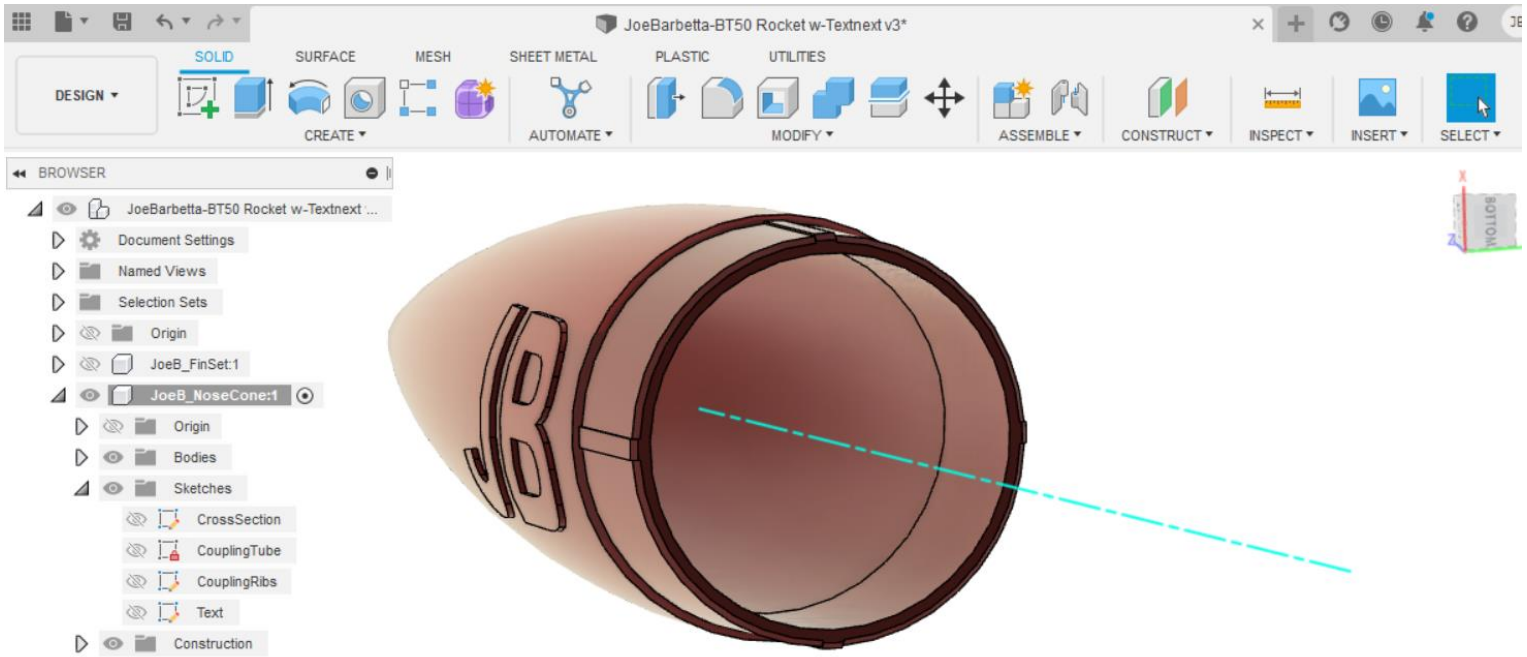
- select the **Extrude** tool and **click on the text**.
- next to Start select **Object**. Note that next to **Profiles** it should show **1 selected**.
- ensure that **Join** is selected next to **Operation**.



- **click on the Nose Cone**, enter **0.015**, and click OK. The text should now stand out.

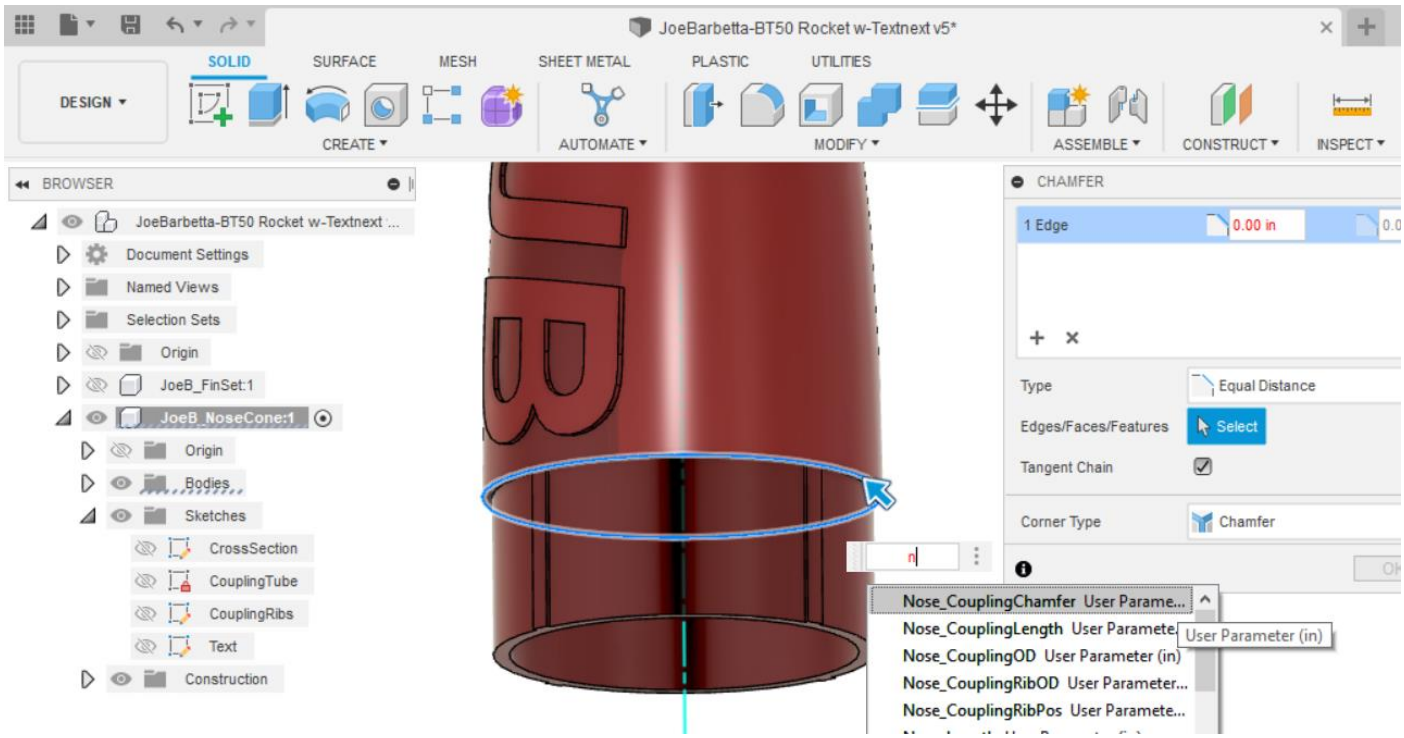


- rotate the view to look inside the nose cone. Text should only appear on the outside surface. There should be nothing inside the nose cone. If there is material for the text inside, use Undo and revisit the previous Extrude operation settings.



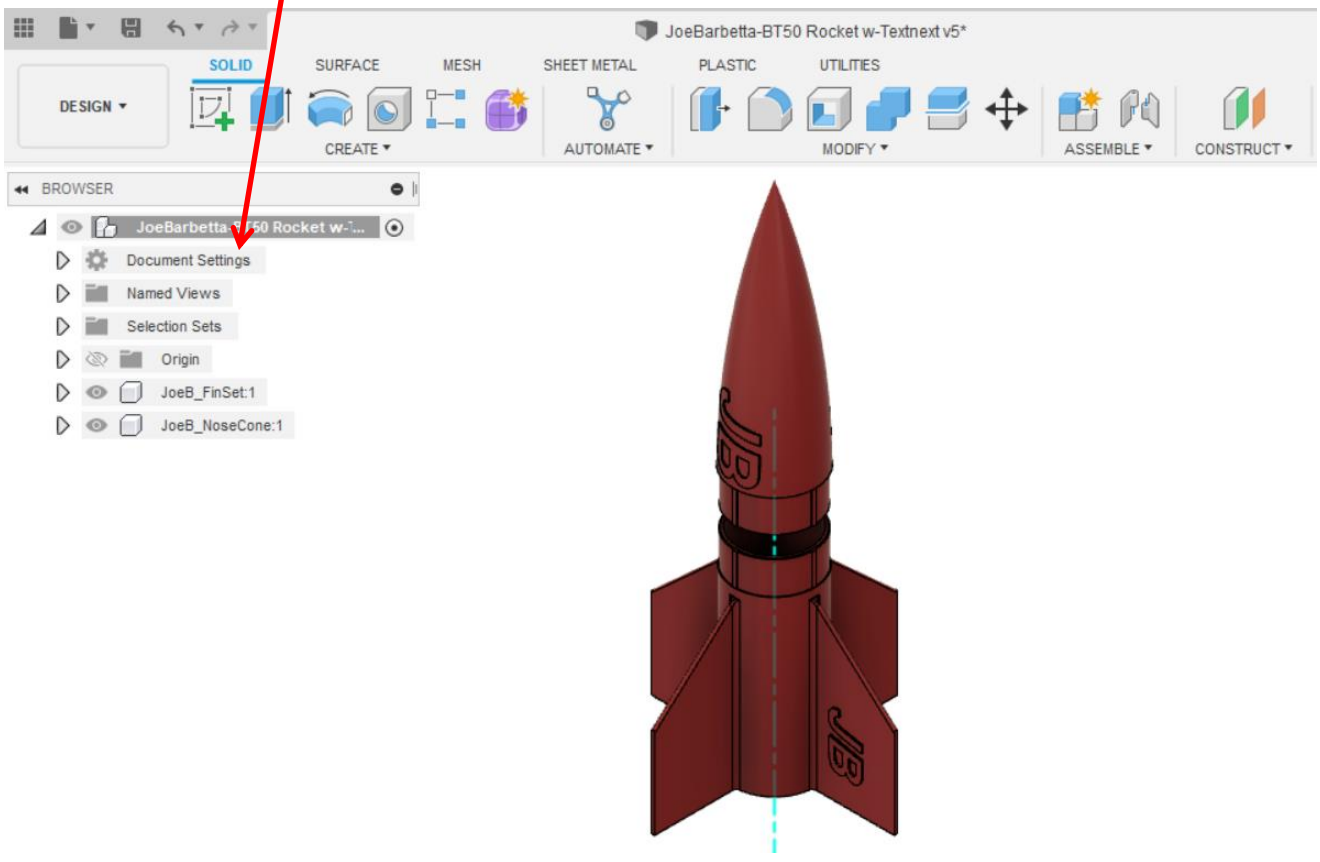
- adjust the view similar to that below
- within the **MODIFY** menu select **Chamfer** and ensure that **Type** is set to **Equal Distance**.
- click on the **outer edge** of the Nose Cone, type **n**, and select **Nose_CouplingChamfer** and click **OK**

This is needed because the Nose Cone will print from the bottom and the chamfer eliminates the overhang.



- click on the **Home** icon at the top-left of the **View Cube**. This will reset the view.
- right-click on the **Project Name** and select **Activate** to see both the FinSet and NoseCone Components

Our rocket looks a little short. However, this is all that's needed to 3D print these components.



Using a Joint

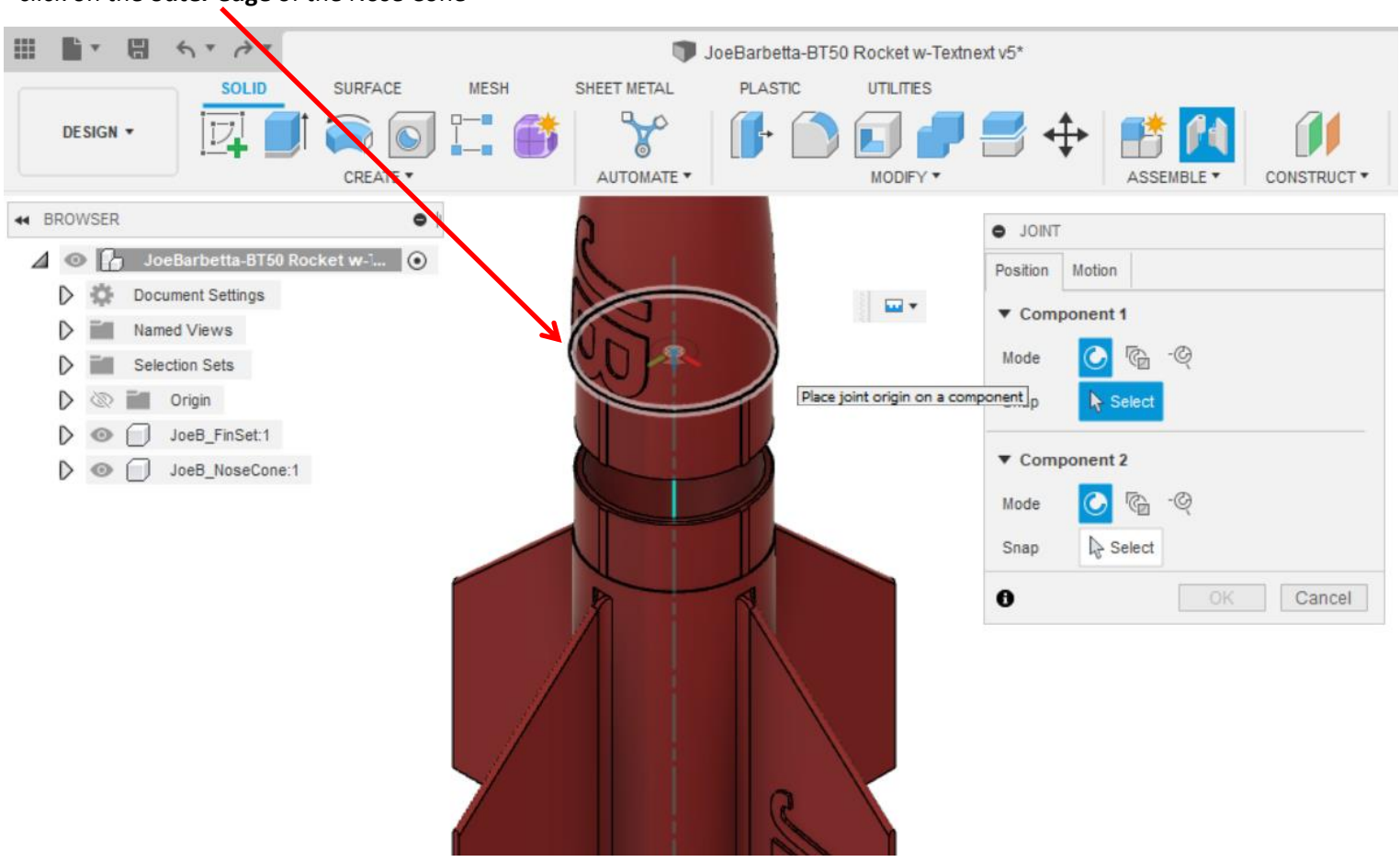
Fusion 360 allows one to create ***Joints*** to control the positions of objects. These positions can be controlled by *Parameters* or controlled dynamically for animated movements.

Because a Joint will be used to position the Nose Cone at a distance from the Fin Set to accommodate the Body Tube, create the following Parameters for the Body Tube.

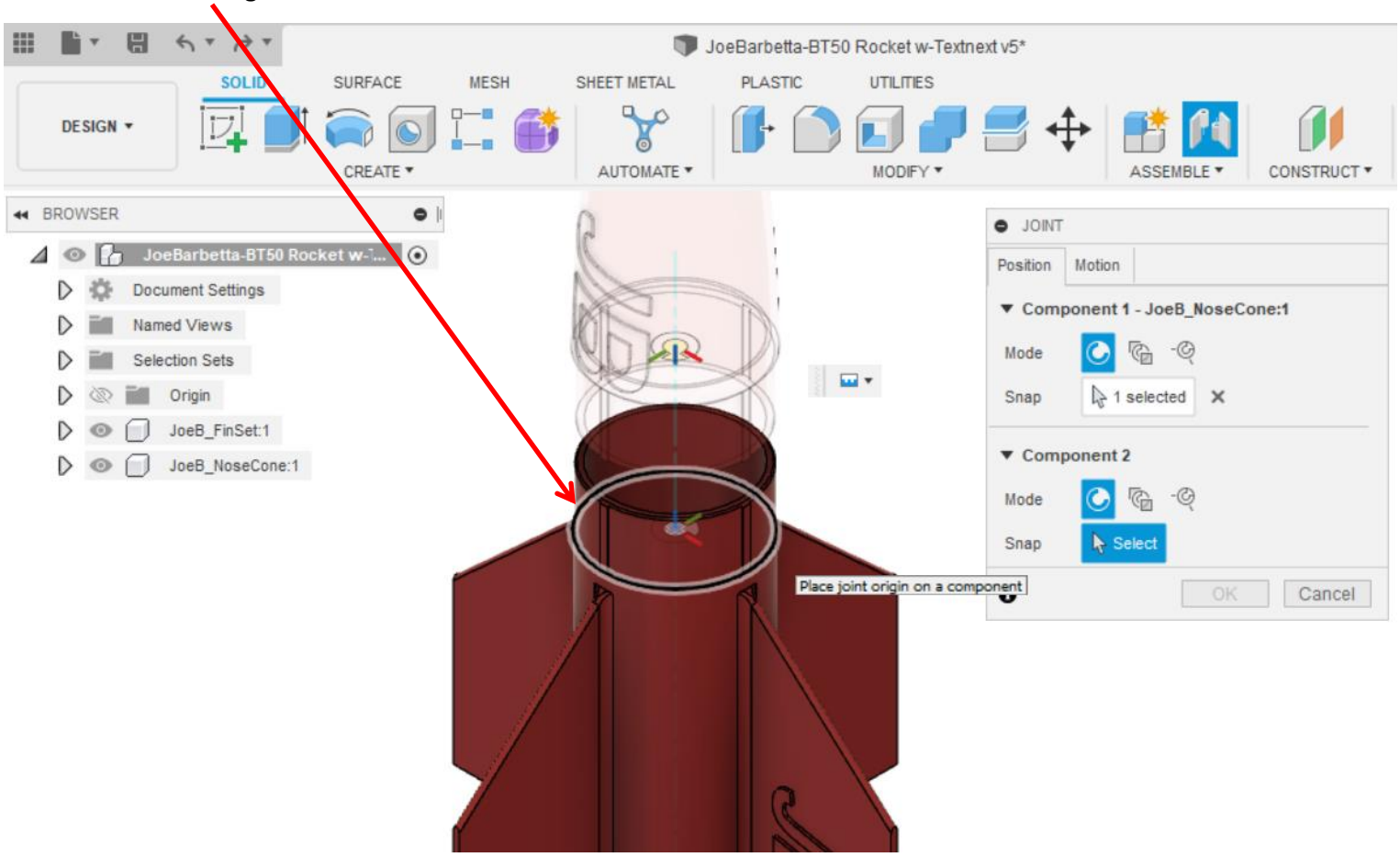
As done previously, open the *Parameters window* and add additional parameters.
- from the top **MODIFY** menu select **Change Parameters** near the bottom of the list
- next to **User Parameters** click on **+** and enter the **Name** and **Expression** for each item in the below list. Note that you will need to click **+** for each item. You may be able to copy and paste Expressions, but **if it shows as red you may have to type the expression.**
- **Don't forget to click OK when done!**

Name	**Expression**
BodyTube_OD	0.997
BodyTube_Thickness	0.013
BodyTube_Length	9.000

- within the **ASSEMBLE** menu select **Joint**
- click on the **outer edge** of the Nose Cone

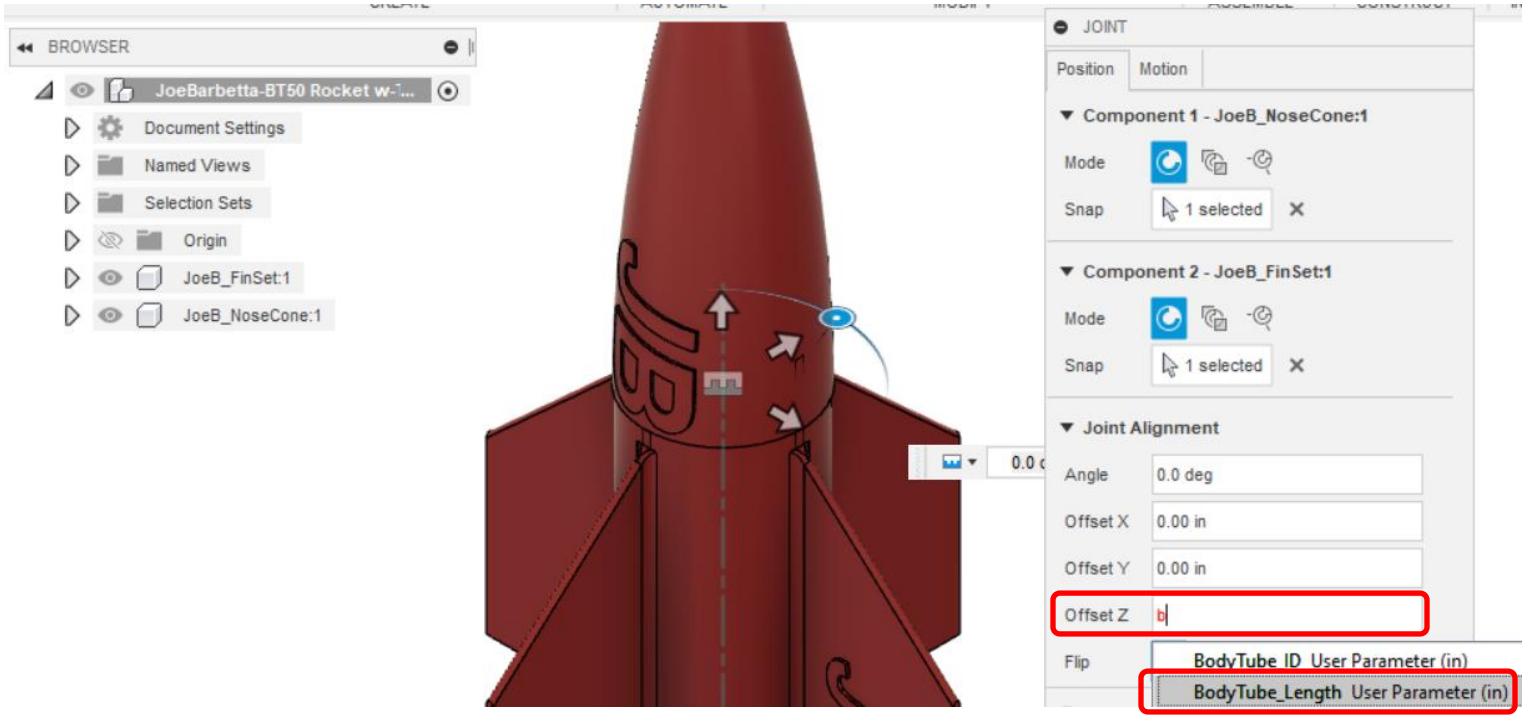


- click on the **outer edge** of the Fin Set



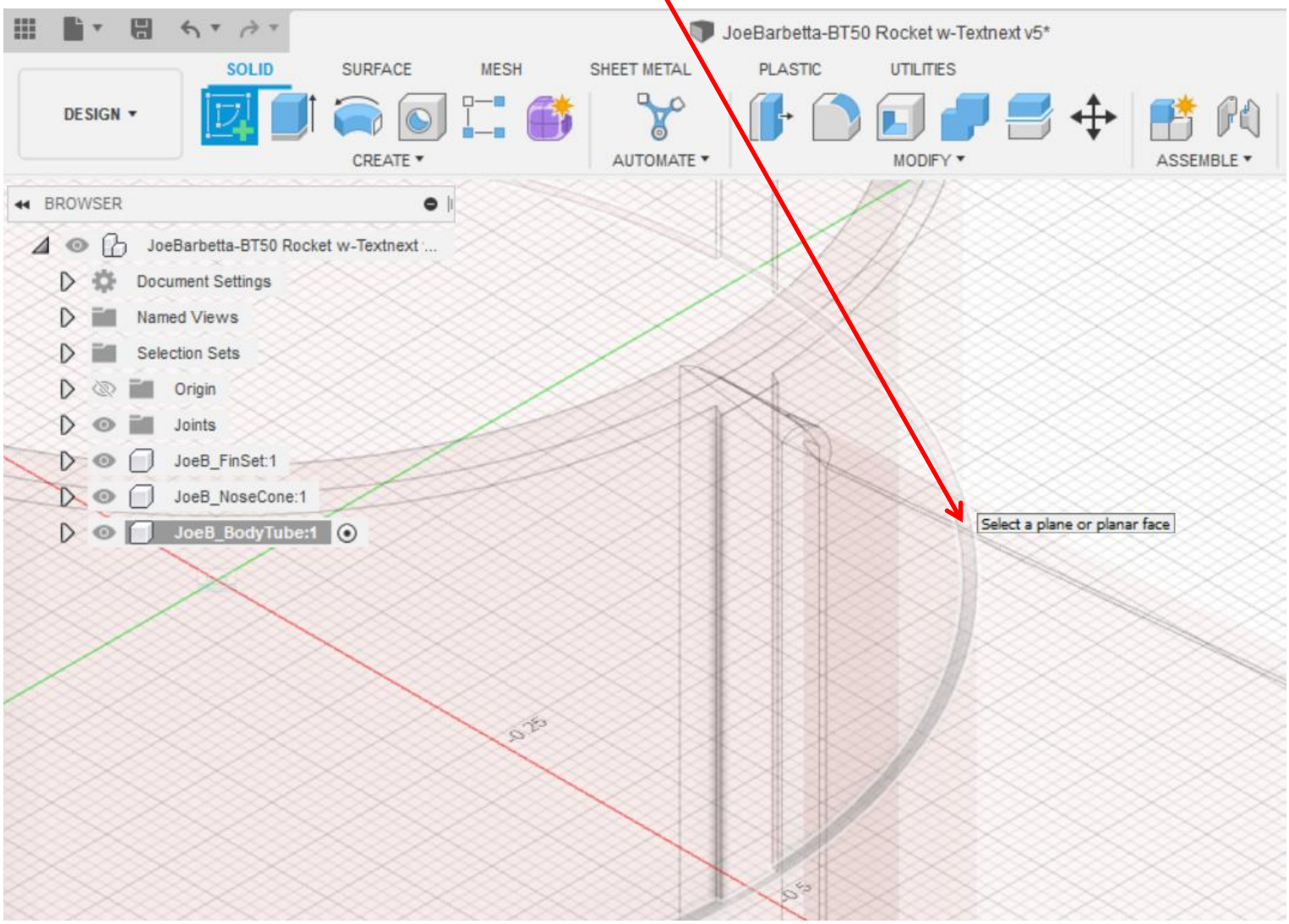
- type **b** in the **Offset Z** box and select **BodyTube_Length**

- use the **Home Icon** at the **View Cube** to reset the view and the Nose Cone should show at a distance from the FinSet

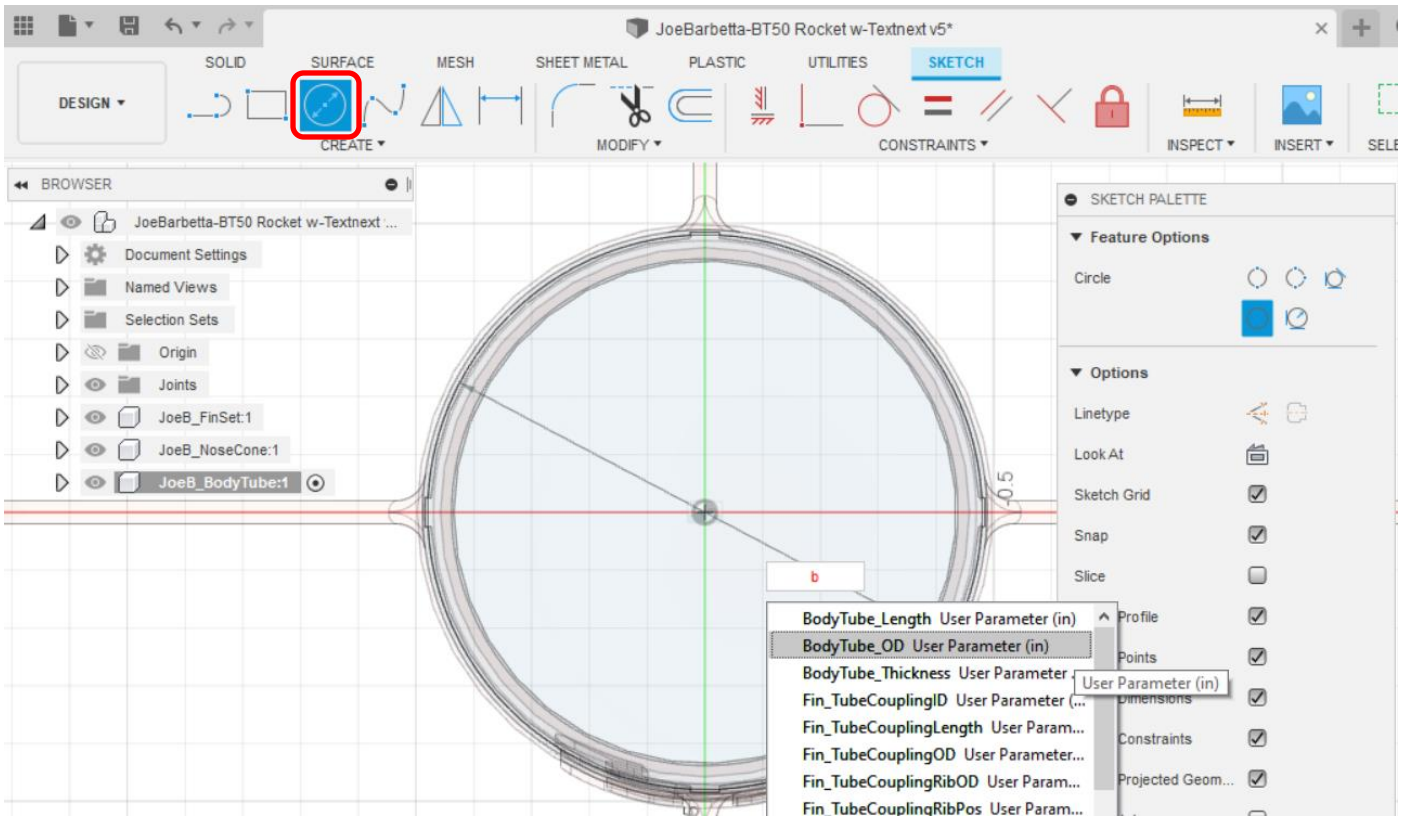


Create a New Component for the Body Tube

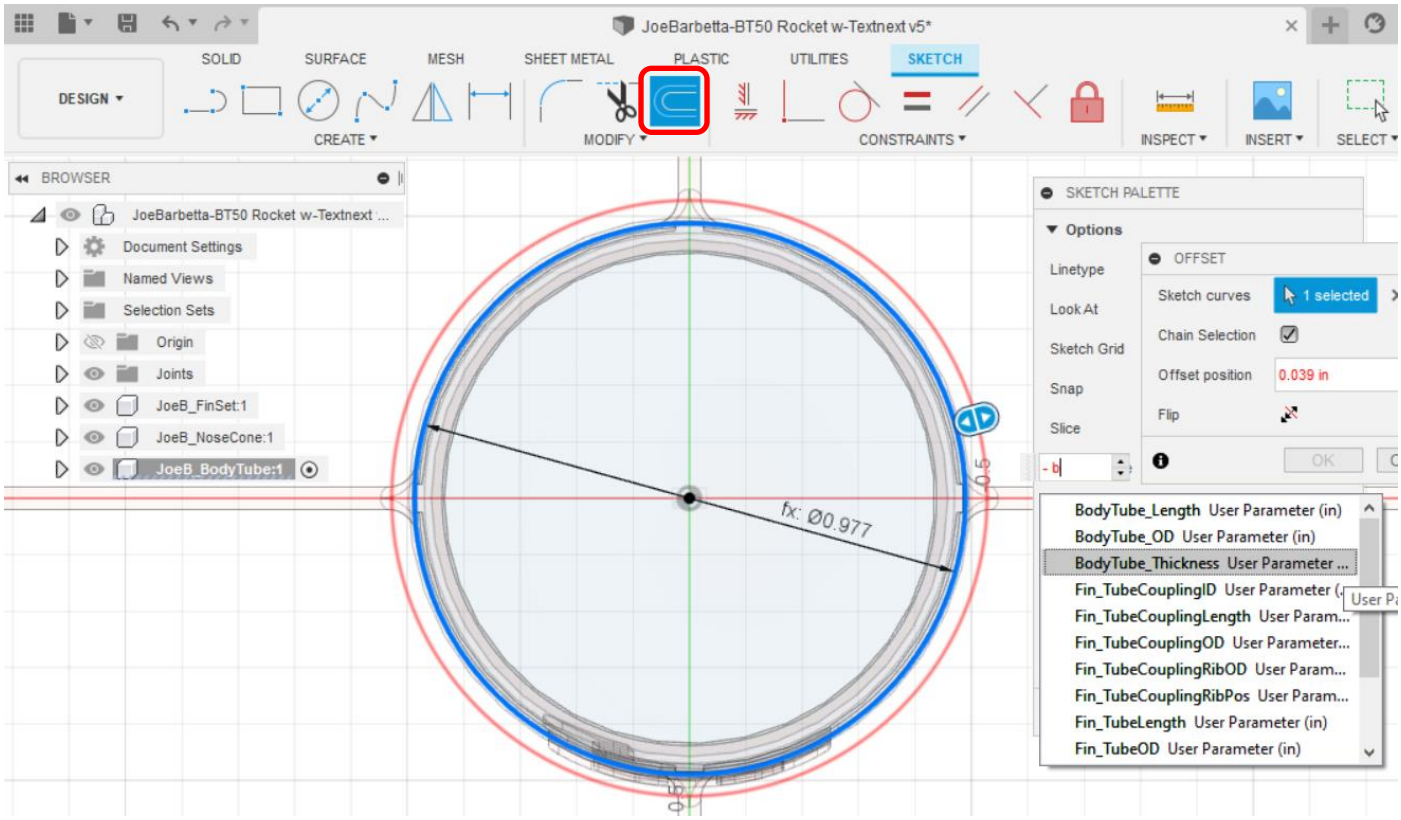
- right-click on the **Project Name** (top line) in the **Browser** and select **New Component**. This will insure it is a "top level" component and not as part of another *Component*.
- enter the component name: **<your first name and last name initial> BodyTube** e.g. **JoeB_BodyTube**
- select the **Create Sketch** tool
- zoom in to the **top of the Fin Set** and carefully click on the top face just below the Coupling Tube section. It is a very thin annular face and may be tough to click on.



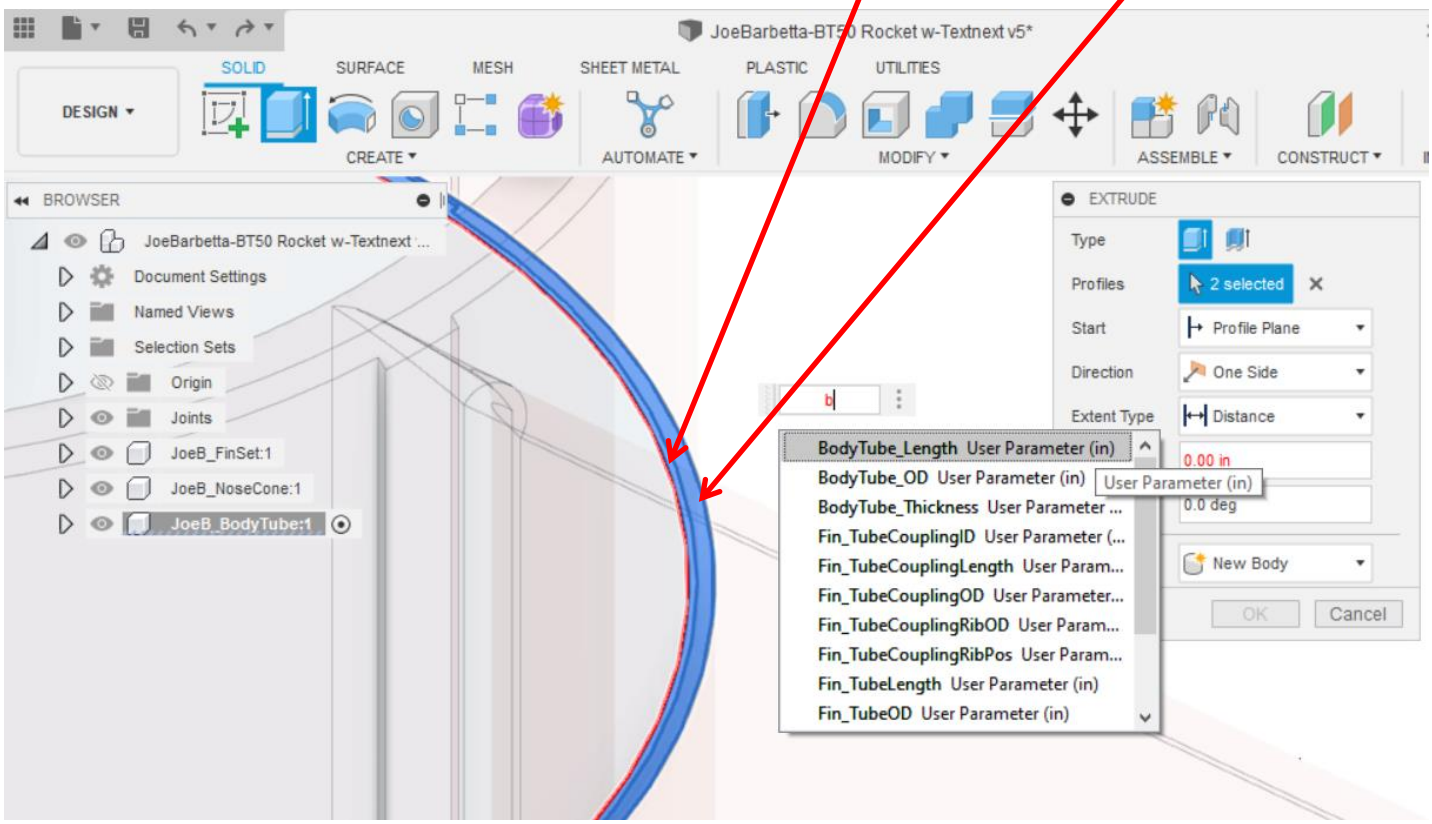
- select the **Circle** tool and create a circle from the center, type **b**, and select **BodyTube_OD**



- select the Offset tool and click on the newly created circle
- type **-b** (note the minus before the b) and select **BodyTube_Thickness**. Click **OK** and then **Finish Sketch**.



- zoom in to the **top of the Fin Set** and select the Extrude tool.
- carefully click on the **two annular regions**. There will be a **very thin inner region** and a **thicker outer region**.
- type **b** and select **BodyTube_Length**. Then click **OK**.



- right-click on the **BodyTube** Component and select **Physical Material**
- open the **Misc** folder and drag the **Paper icon** onto the Body Tube