

Rocket Design Guide

The goal of this guide is to show people how to go about modeling their rocket in both rocksim and solidworks. The main emphasis is focused on SolidWorks, as that is far less straightforward if you have no experience. My level 1 and level 2 rocket are shown as well to serve as a template for the RockSim in case you don't know where to start. SolidWorks models should be based off RockSim. By modeling in SolidWorks, you can 3D print and laser cut your designs which is critical for the level 1 rockets.

RockSim Basics

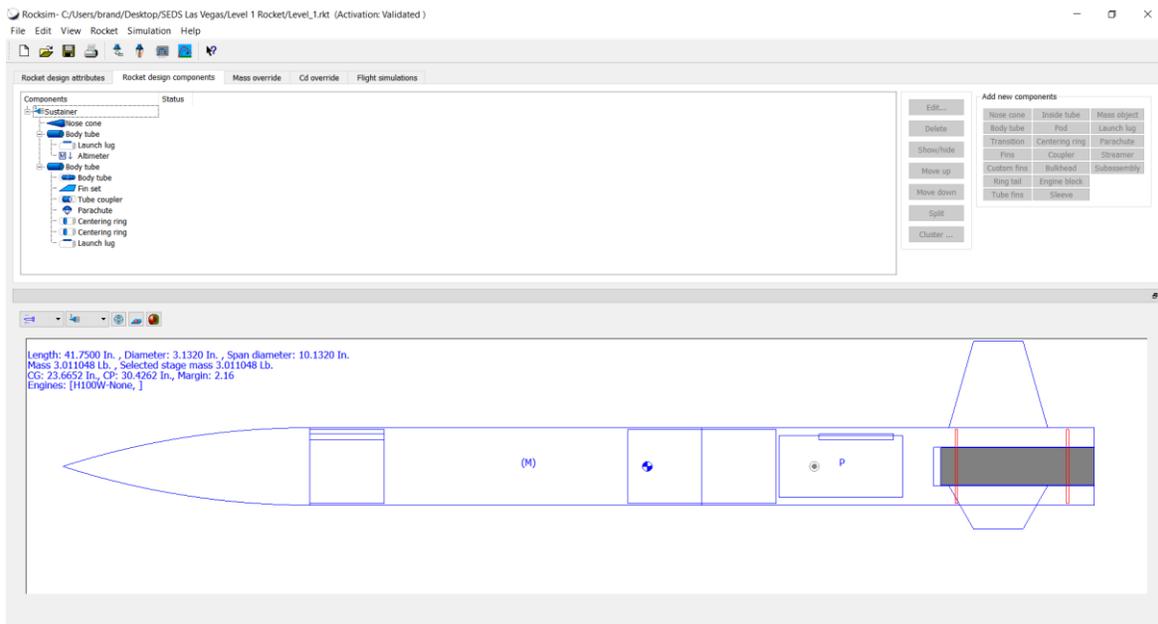


Figure 1: Level 1 RockSim Rocket

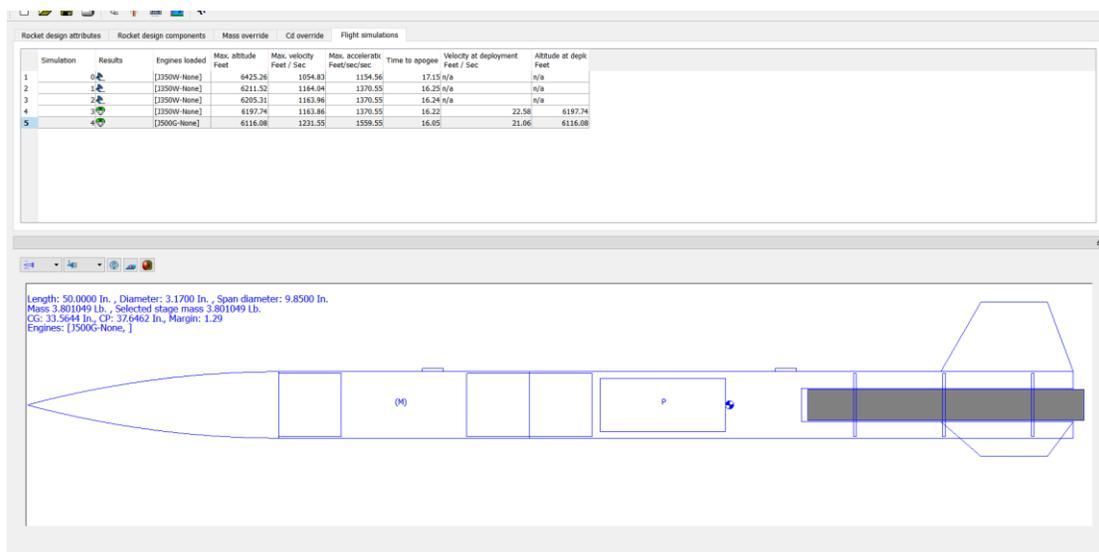


Figure 2: Level 2 RockSim Rocket

I won't be going in depth on RockSim. I would say that you should have most of the components shown here, though your design can vary as you please.

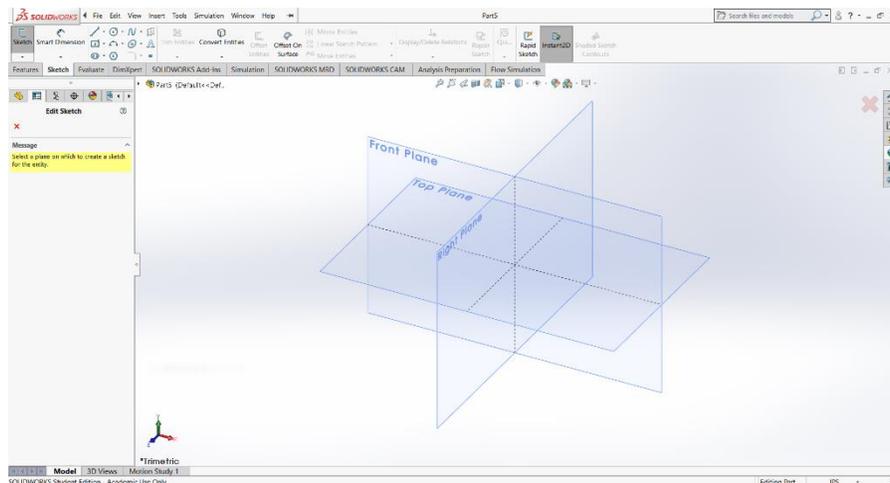
Some design considerations:

- It is important that your center of pressure (CP) stay below your center of gravity (CG)
 - This helps your stability stay within an okay range
 - In general, keep stability between 1 and 2
 - Make sure motor is loaded when checking stability
 - If overstable, your rocket will follow the wind
 - If understable, your rocket will zigzag and can be dangerous
 - Ensure the coupler (piece connecting the tubes) is double the length of your tube diameter
 - i.e. 3in diameter = 6in coupler
- on the bottom tube, add an inner tube and make sure to check it as the motor mount
 - this allows you to add the motor to it

SolidWorks Design

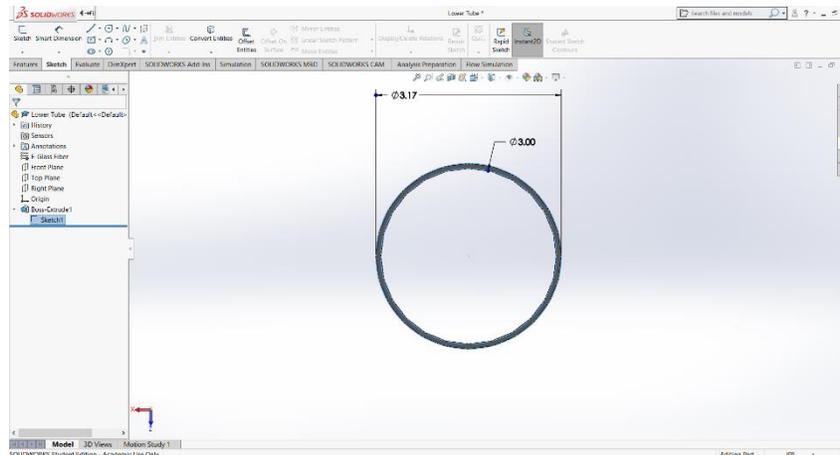
Tubes

Step 1: Start a sketch



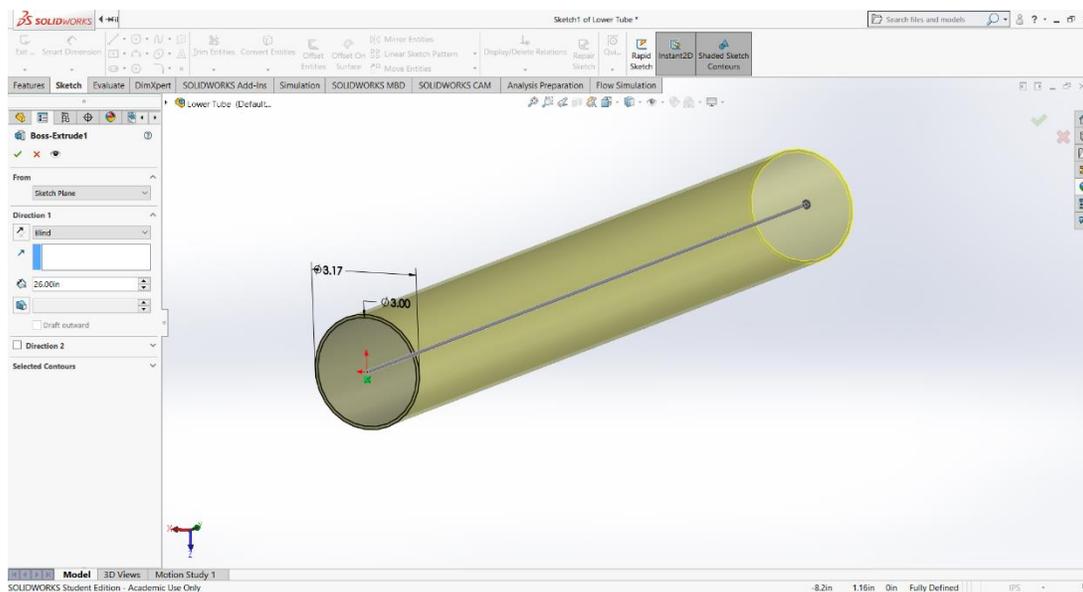
- Select sketch in the top left corner. The three drawing planes will appear
- These are the faces that you will be drawing on. Selecting a specific one is not critical, though its orientation in assemblies can be made easier if the appropriate plane is selected
- For the tubes, select the front or top plane

Step 2: Draw the inner diameter and outer diameter of the tube



- Once in the sketch, select the circle tool in the top left under the sketch tab
- Draw two circles and dimension them
 - Select smart dimension to dimension
 - Click each circle one at a time and type in the known diameter

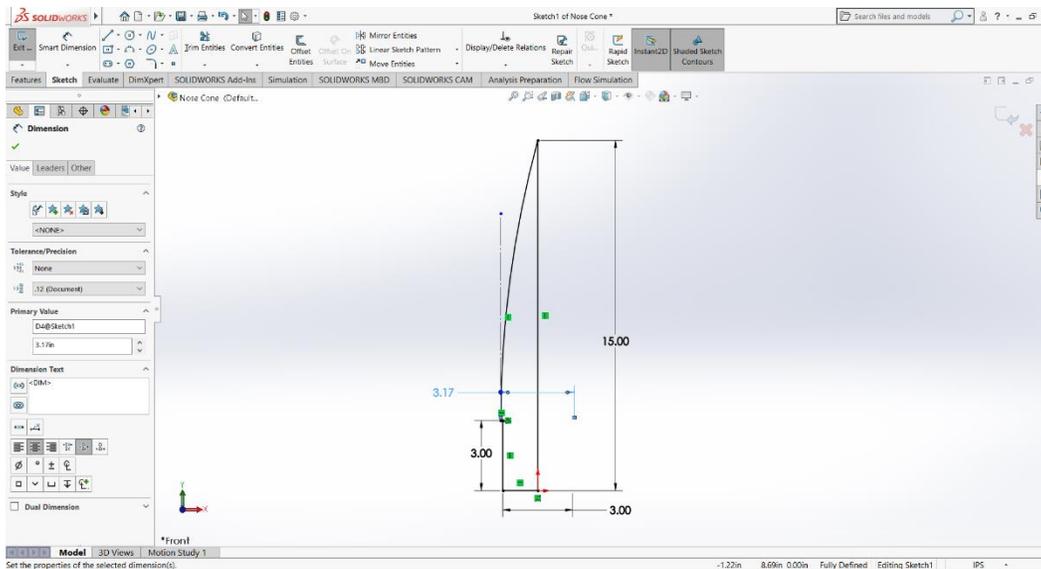
Step 3: Extrude the circle to make the tube



- Select the features tab in the top left
- Select extrude body
- It should auto select the desired contour, but if not, select the ring you drew
- In direction 1, set the desired length of the tube

Save and Repeat for all tube lengths

Nose Cone



Step 1: start a sketch and draw the contour above

- I typically start with a centerline to dimension everything from. This is because I like to revolve symmetric objects
 - Centerline can be found by clicking the down arrow on the line tool
 - Simply connect it to the dot in the center of the drawing and draw straight up
 - Dimension the centerline using smart dimension. Set it to the full length of your nose cone including the shoulder
- For the contour, start with the shoulder and dimension it appropriately
- Create the outer lip for the nose cone. This is where it sits on the rocket tube.
- Since I'm lazy, I just use the arc tool to eyeball the cone curve
 - Arc is located next to line under the sketch tab
 - Connect one end to the outer lip of the shoulder and the other to the tip of the rocket.
 - Draw it so that the arc does not extend past the lip
 - I drew a straight construction line from the shoulder lip that I tangent-ed the arc to which ensured that it did not exceed the bounds
 - To tangent them, select the arc and centerline both while holding ctrl.
 - An option for tangent will appear on the right

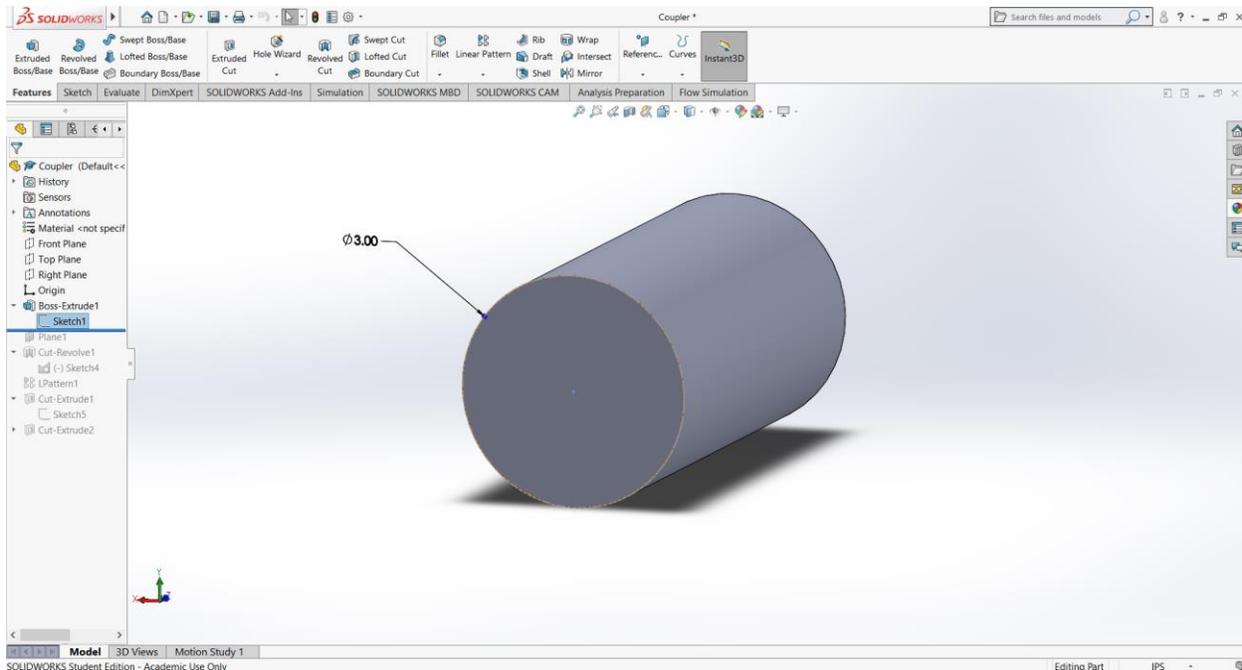
Step 2: Revolve

- Under features, select the revolve tool
 - It may prompt something about it being a thin feature. Select no if you want it solid, select yes if you want the cone empty

- If yes is selected, there is a slightly different process for revolving not covered here
- Under revolve, select the axis of revolution, being the centerline created at the beginning

Coupler

Step 1: Cylinder Extrusion

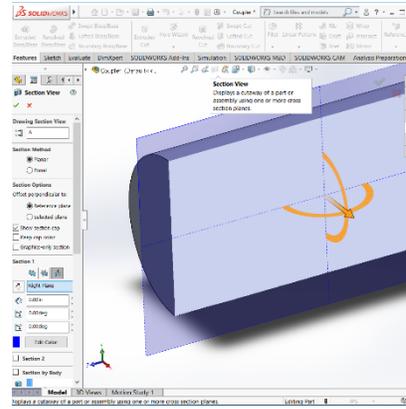
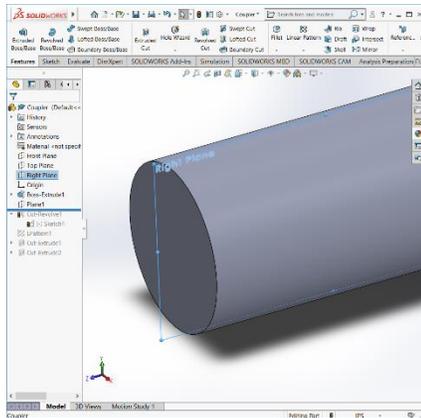


- Start with a sketch a draw a circle. Dimension the diameter. Follow by using the extrude tool to extend it into a cylinder

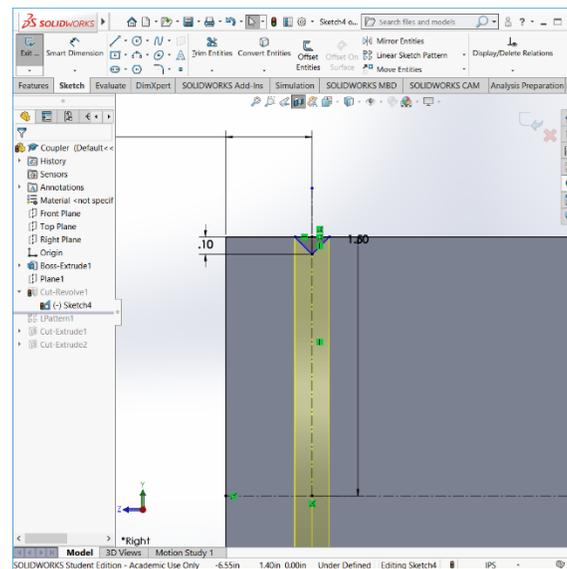
Step 2: Epoxy Slits

There are several ways to do this step. I will show the way I chose to do it. It may be rather complicated if you are unfamiliar with solidworks.

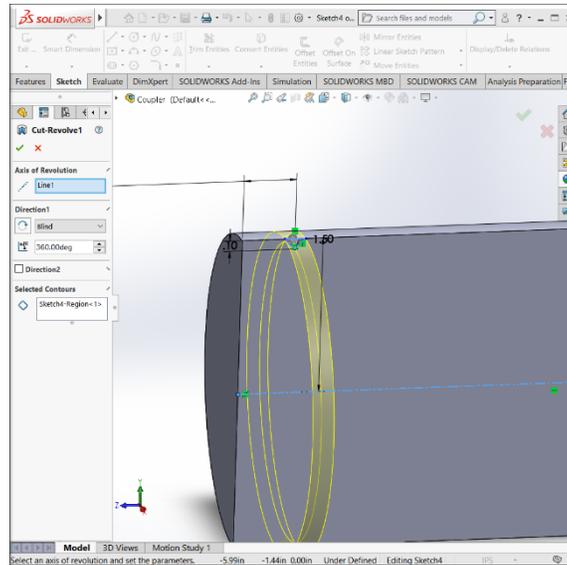
- Start by selecting a plane (in my case the right plane) that goes through the center. We will need to draw on it to cut the slit it
- While the plane is selected, click the section view button to cut the object in half. This is useful for drawing items within the object



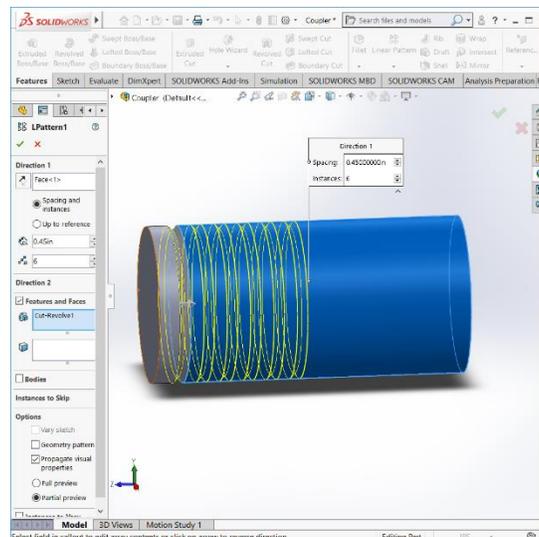
- Select okay. Reselect the right plane and start sketching on it.
- Make a triangle on the surface and dimension as desired
- I made a center line and offset it to get the edges of my triangle
 - Offset is a tool in the sketch widget used to clone an item a set distance and direction. There is a bidirectional option if you want it to go both ways
- From there I dimensioned where I saw fit.
 - I trimmed away most of the work for this before pictures (sorry!)



- Before exiting the sketch, I made sure to draw a center line through the center of the object. This will be the reference line for a cut-revolve.
- Select cut revolve and choose that centerline as the axis and click ok

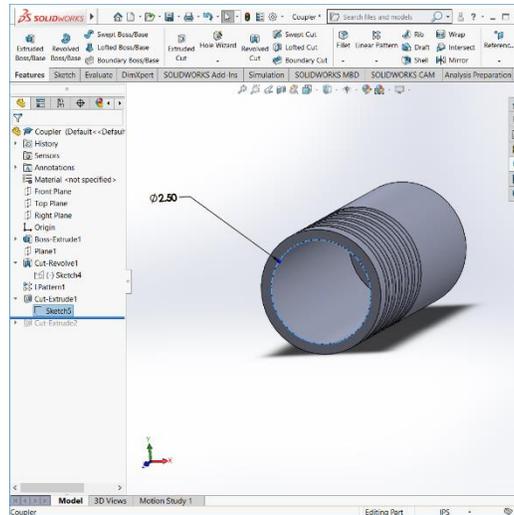


Step 3: make many slits!



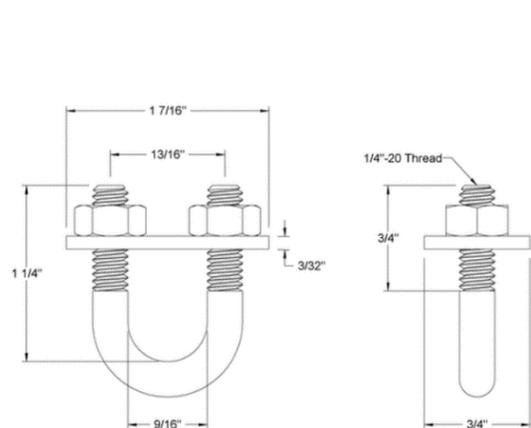
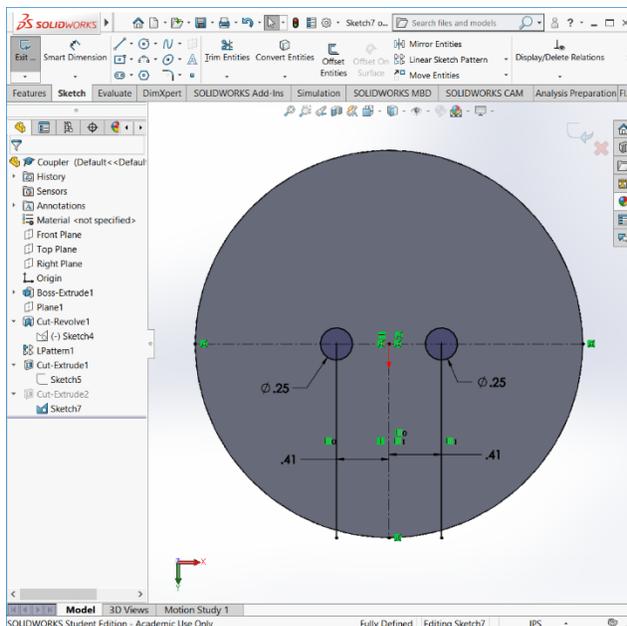
- Under features select linear pattern
 - In the features and faces are, simply select the cut-revolve by choosing it from the build tree or directly clicking it in the model
 - I selected the face for direction
 - Set the distance
 - Choose the amount

Step 4: Hollow it out



- Start a sketch on the face by the slits
- Draw a circle and dimension
- Select extrude cut and type the desired distance

Step 5: Optional U-bolt mounting



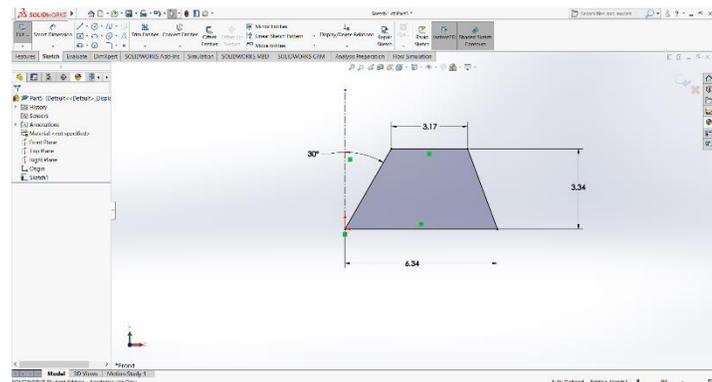
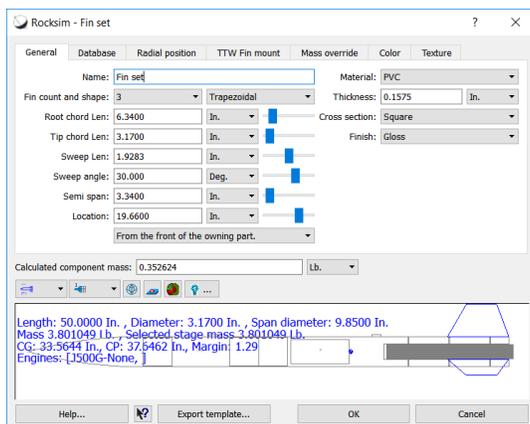
- This is optional because your design may not need the U-bolt
 - You may also choose to drill your print by hand instead
- I found a U-bolt that fit what I wanted from mcmaster carr
- Proceeded to design the holes to the specs of the U-bolt

- I used two centerlines
 - One at the middle across
 - One straight down from the middle
- Use the offset tool on the centerline to get the centers of the circles.
 - From the schematic, we can see that bolt heads are 13/16 apart. Therefore they are 13/32 from the center, which is the offset distance used with bidirectional
- Finish with cut extrude
 - You may have to select both circle if solidworks doesn't auto recognize them as the contour to be cut.

Fins

Step 1: Draw it out

- Start by drawing the general shape
- Once the shape is created, dimension it to around your rocksim fins
 - Reference my dimensions if you don't know what the names of different lines refer to

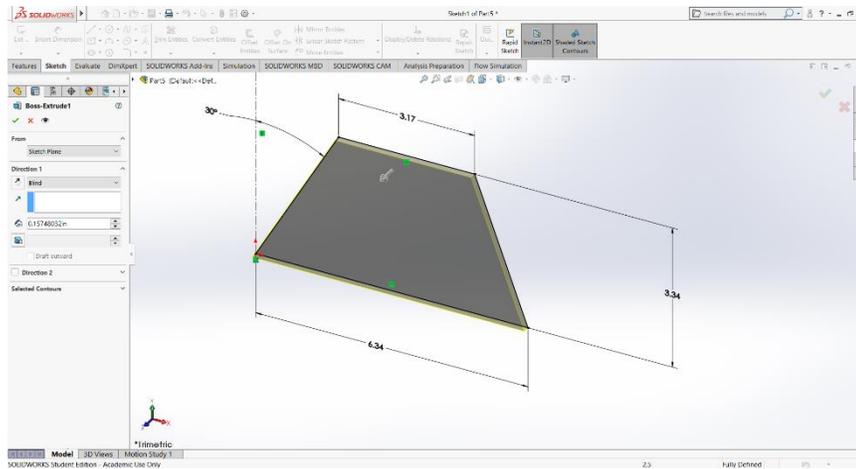


Step 2: Decisions

There are two options for fins. If laser cutting, you will end up with a square edge that can be modified by sanding. This is easy to make from here. However, if you choose to make an airfoil, the design is more complex. I will show both methods for solidworks

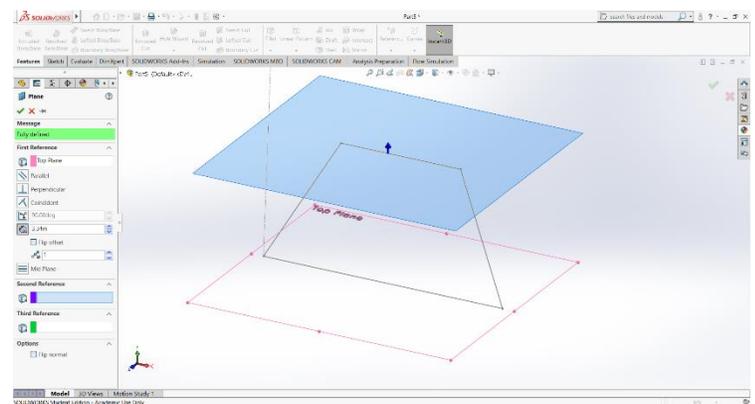
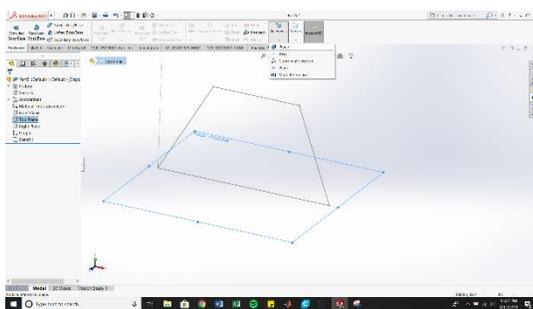
Step 3.1 Square Cross Section

- Once your fin is dimensioned, simply use the extrude tool to extend it the desired distance

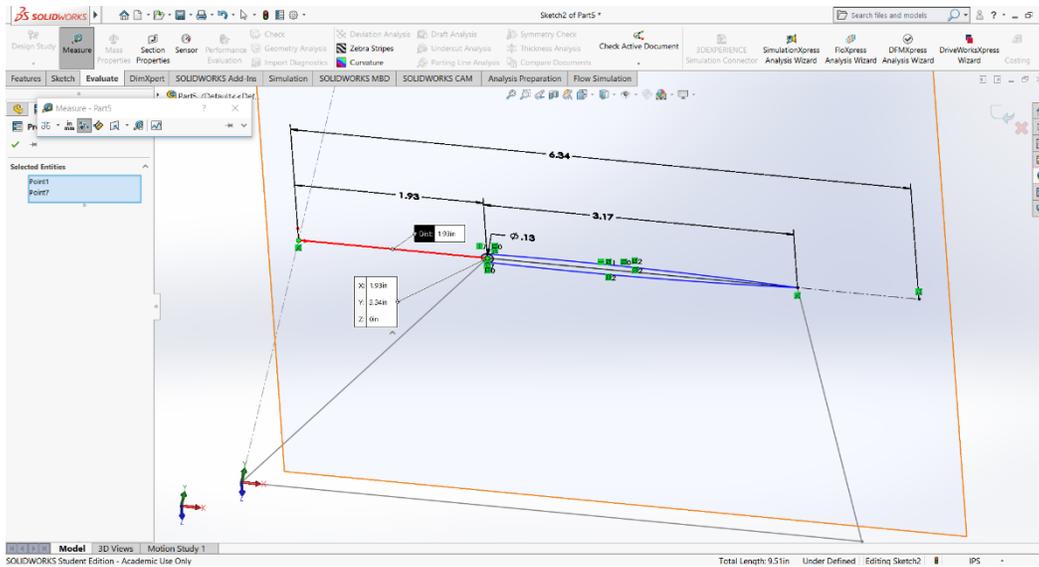


Step 3.2 Airfoil / Tear Drop Cross Section

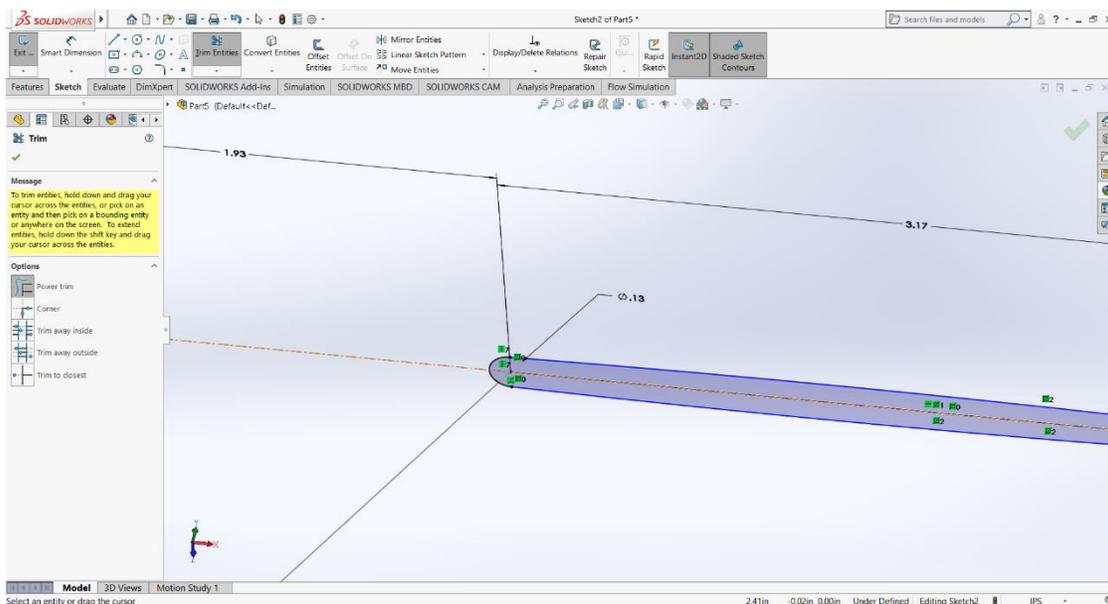
- To do this, I will create a loft by sketching the airfoil at the top and bottom
 - This requires making separate planes to sketch the design



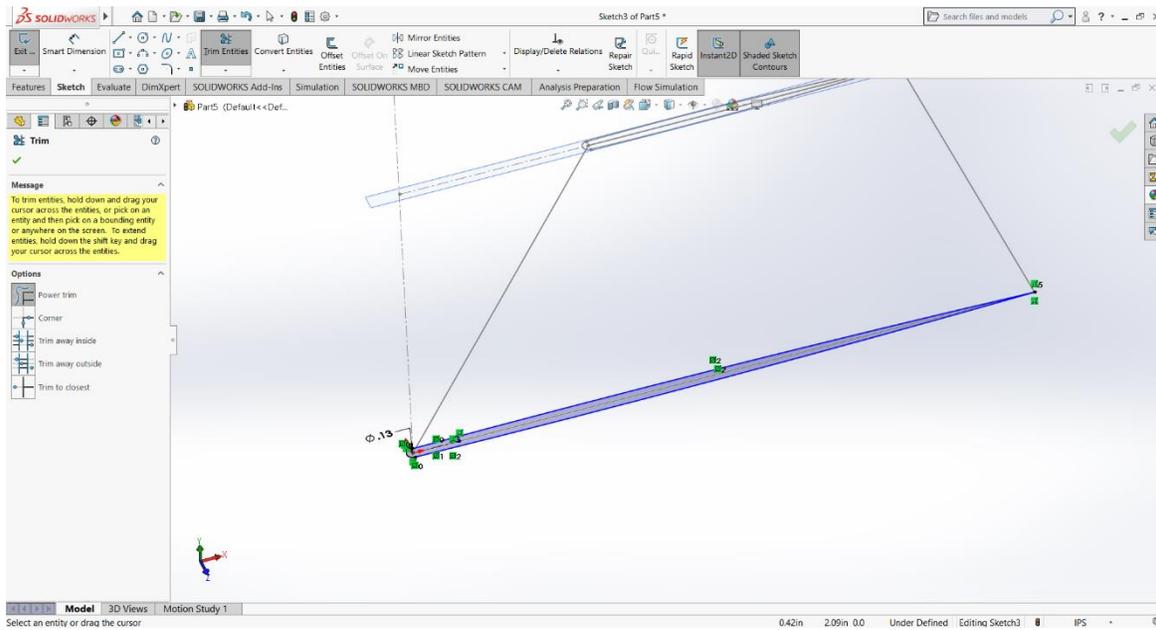
- Select the bottom plane, then choose the drop down arrow under references to select the plane option
- There is a distance option where you can tell the plane where to do. Choose the distance required to land on the top of your fin



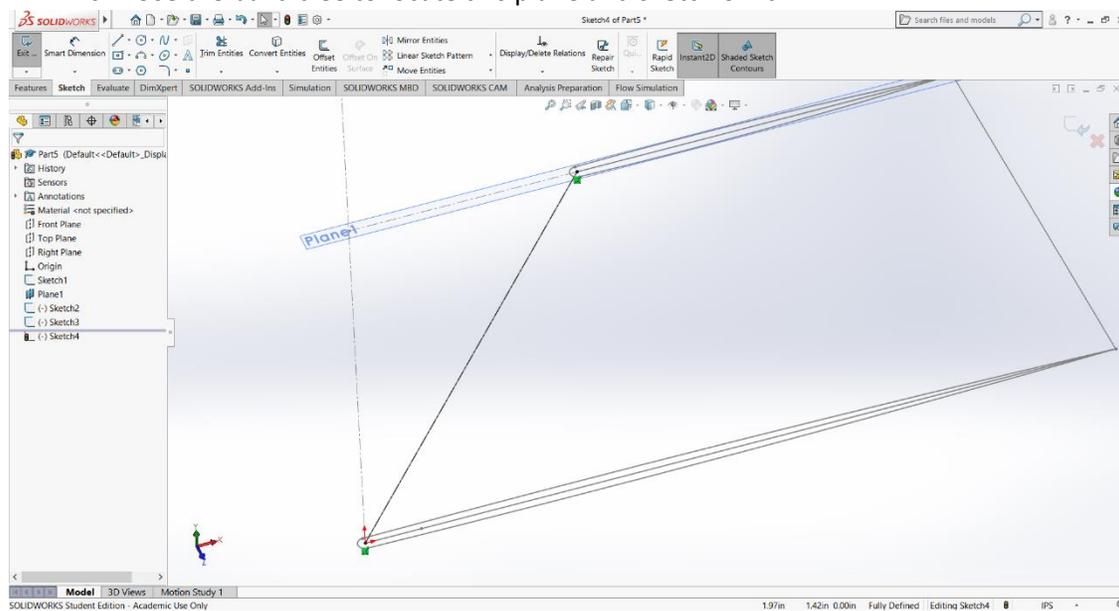
- Start a new sketch on this plane
 - We need to start drawing on the flat part at the top of the fin
 - I found the leading edge distance by using the measure tool located under the evaluation tab
 - Choose the center line and the point where the circle will go
 - I had to do this because the point wouldn't snap to a line. By finding its horizontal distance with measuring, I dimensioned it over the correct distance
 - Draw a circle at that point and dimension it the max thickness. I chose 1/8".
 - Use the arc tool to connect the edge of the circle to the back tip of the fin. Make sure the arc does not exceed the max diameter
 - Select the mirror tool and choose the arc as the feature and the centerline as the axis
 - This ensures we are producing the same arc on the other side



- Select the trim entities tool to cut out the inner circle. This creates a closed sketch
 - Simply click and drag to trim lines

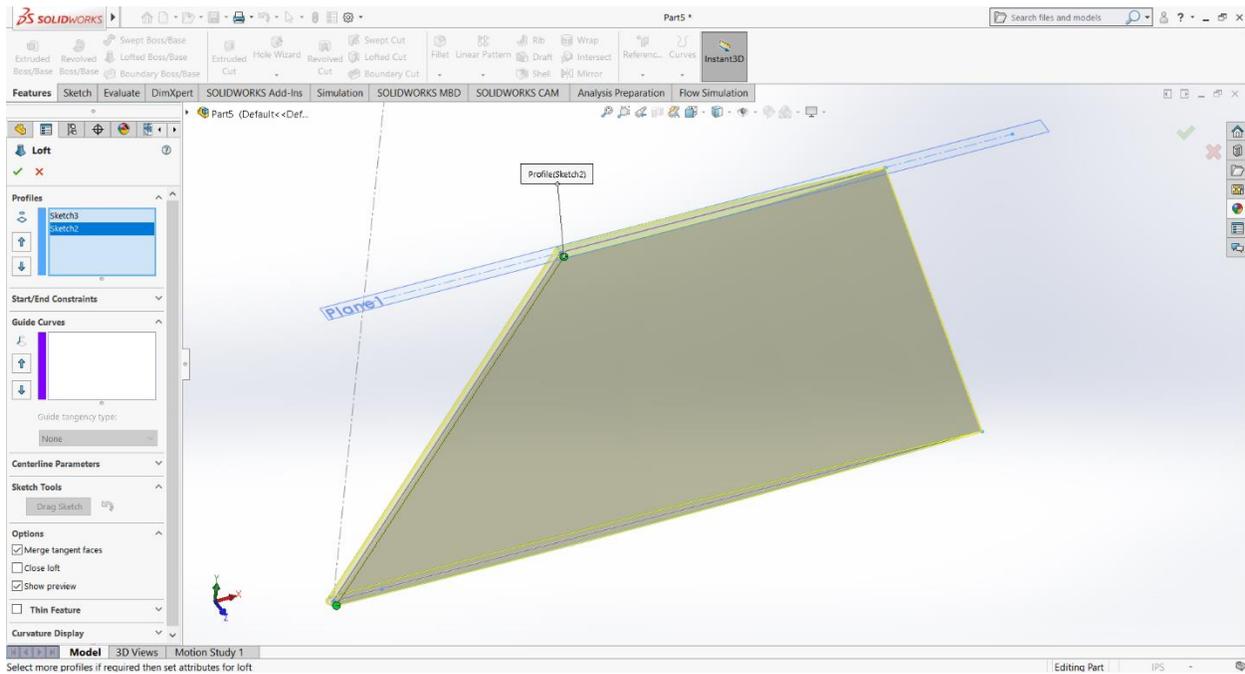


- Do the same thing on the bottom plane by exiting the current sketch and starting a new one on the bottom plane
 - Use the build tree to locate this plane and sketch on it



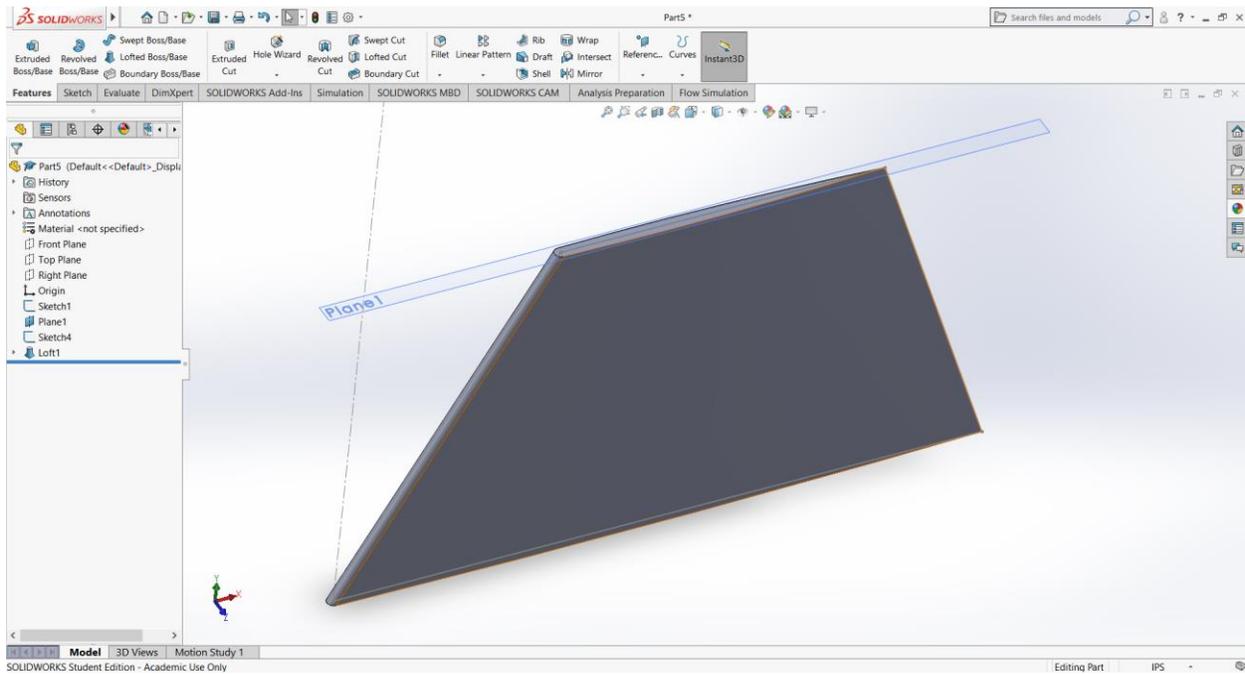
- once finished, exit that sketch and start a new one on the front plane
- Draw a line connecting the center of the top and bottom circle

- This will serve as a guide for the loft



- Exit the sketch
- Under features, select the lofted base option
 - It will give some nonsense warning, disregard it
- When the selection manager opens, simply click in the box on the left (blue) and proceed to click the circles you created from the previous sketches
 - Solidworks should connect the two as such

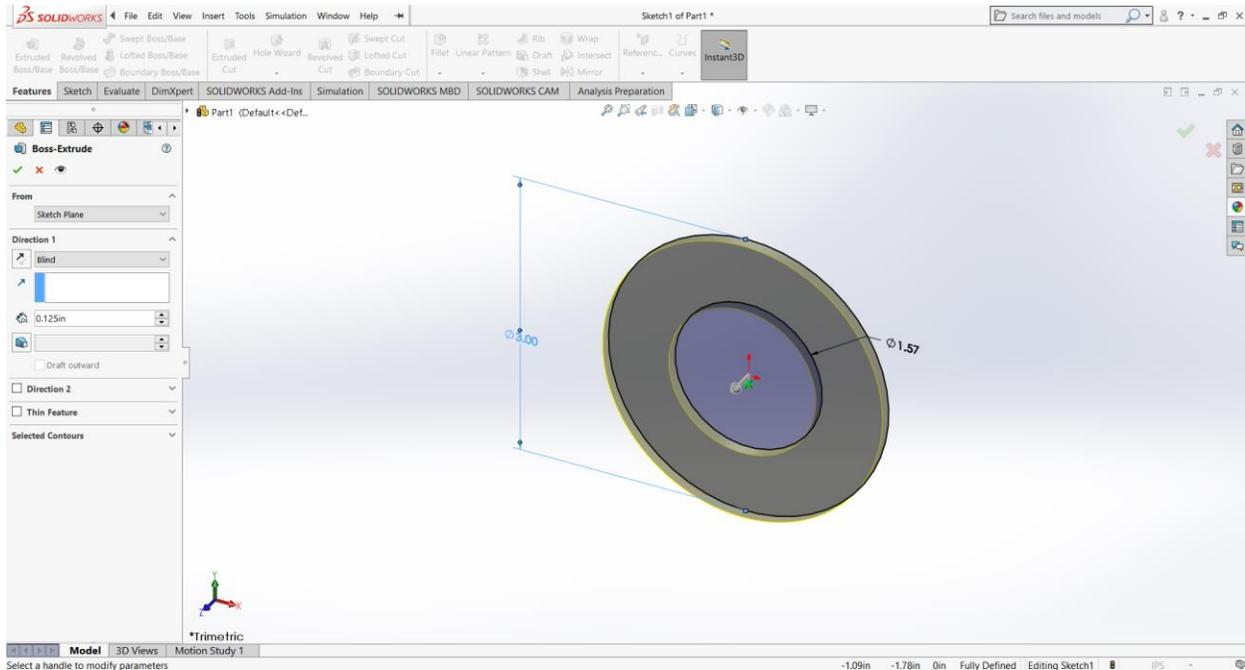
Result



You can hide the plane by right clicking it in the build tree and selecting the hide option (eye)

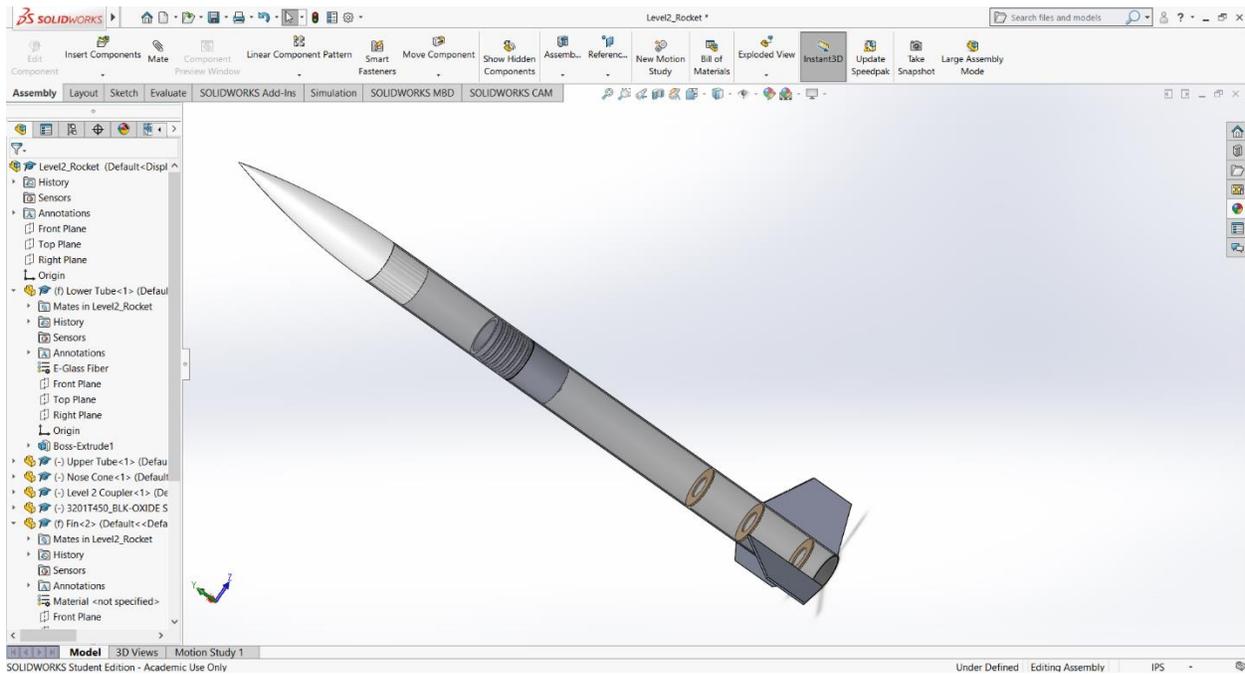
Centering Rings

Step 1: Draw and Donut and Extrude it



- Select sketch and choose the front plane
- Draw two circles and dimension the inner and outer diameter
- Under features tab, select extrude and type in the thickness of the centering ring

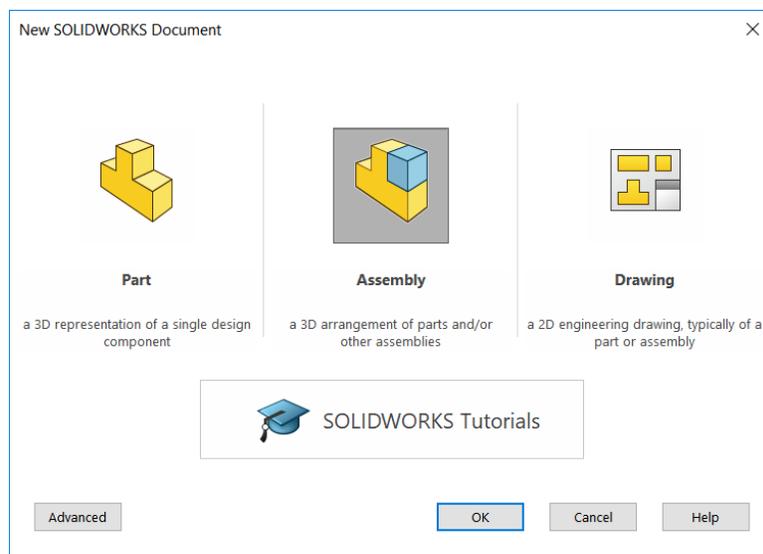
Assembly



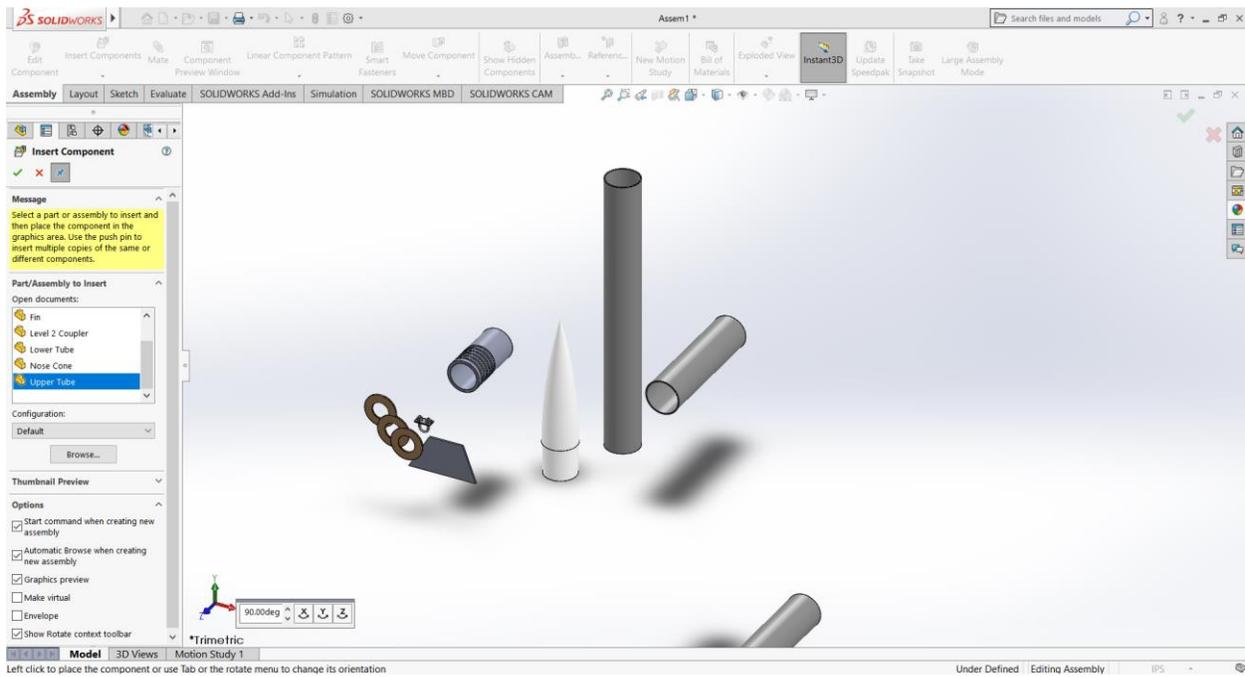
Now that all the main pieces are constructed, you can go ahead and assemble them to see how each item fits.

Step 1: Start an Assembly

- Start by hitting ctrl+n to bring up a menu of new documents. Select assembly



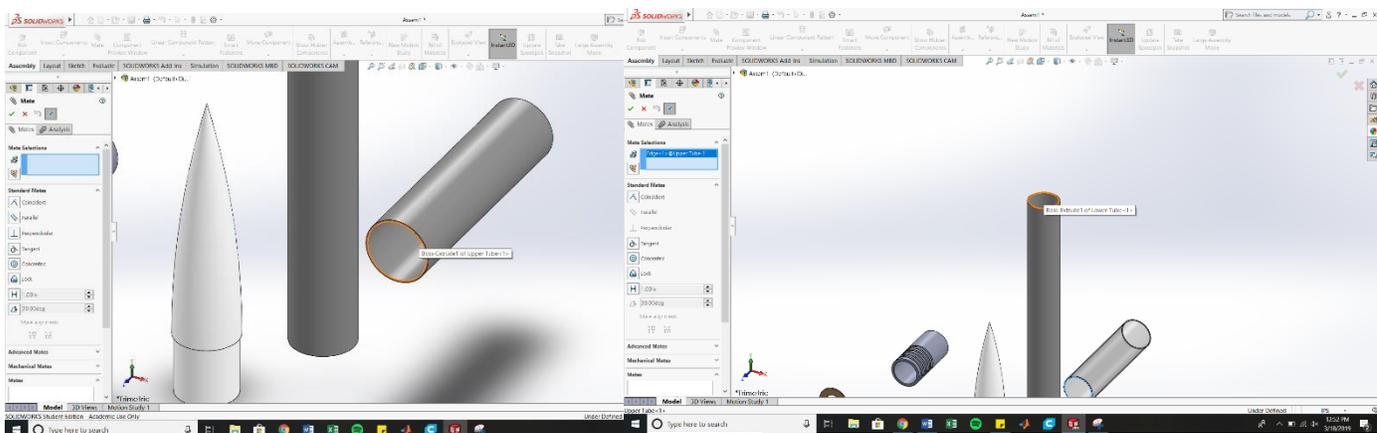
Step 2: Place your parts

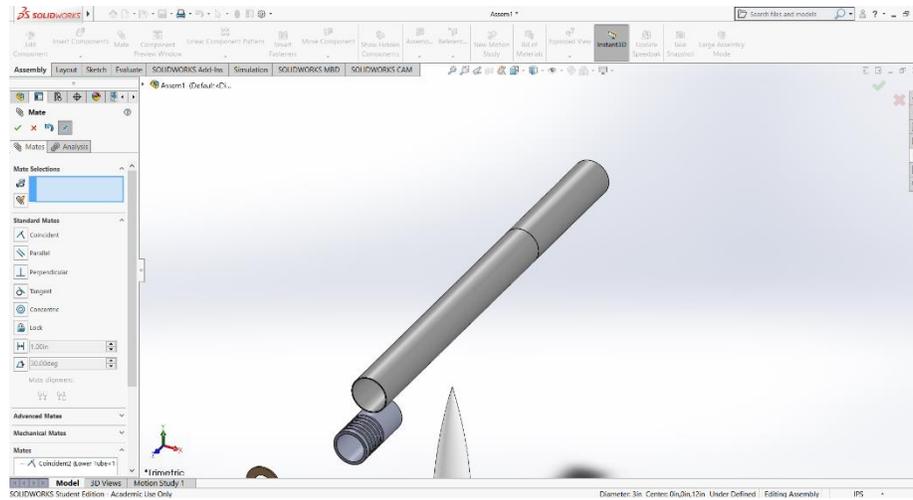


- Once in the assembly, the following screen should appear with this tree along the left
 - Select each item and place it in the assembly
- If the items are not in the tree, simply select browse and find the part from it's saved location

Step 3: start with the tubes

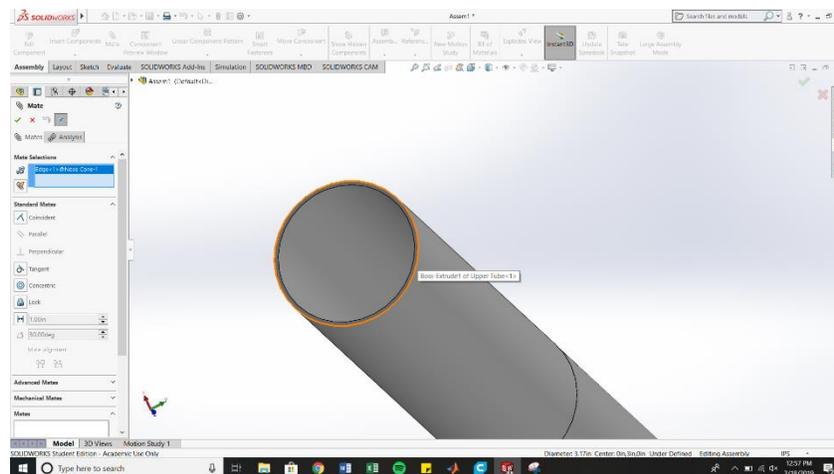
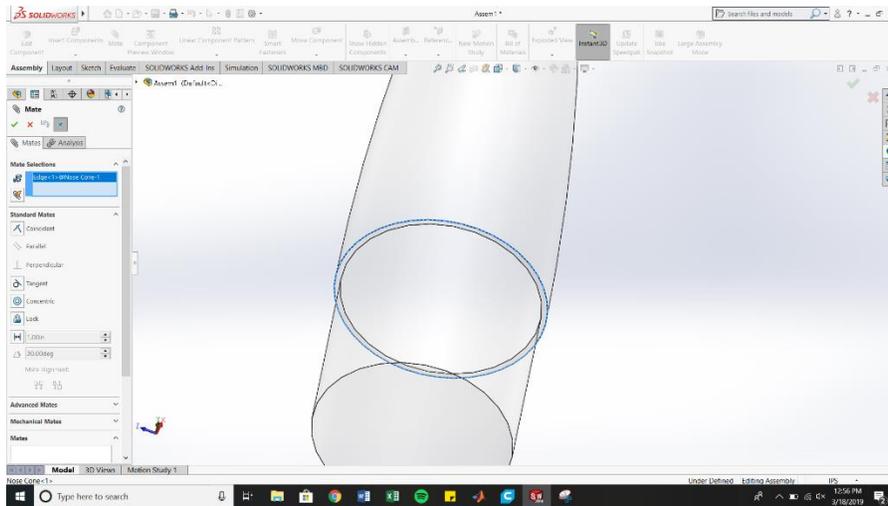
- Select mate, which is located under assembly tab
- Select the locations marked on the images to mate the tubes together

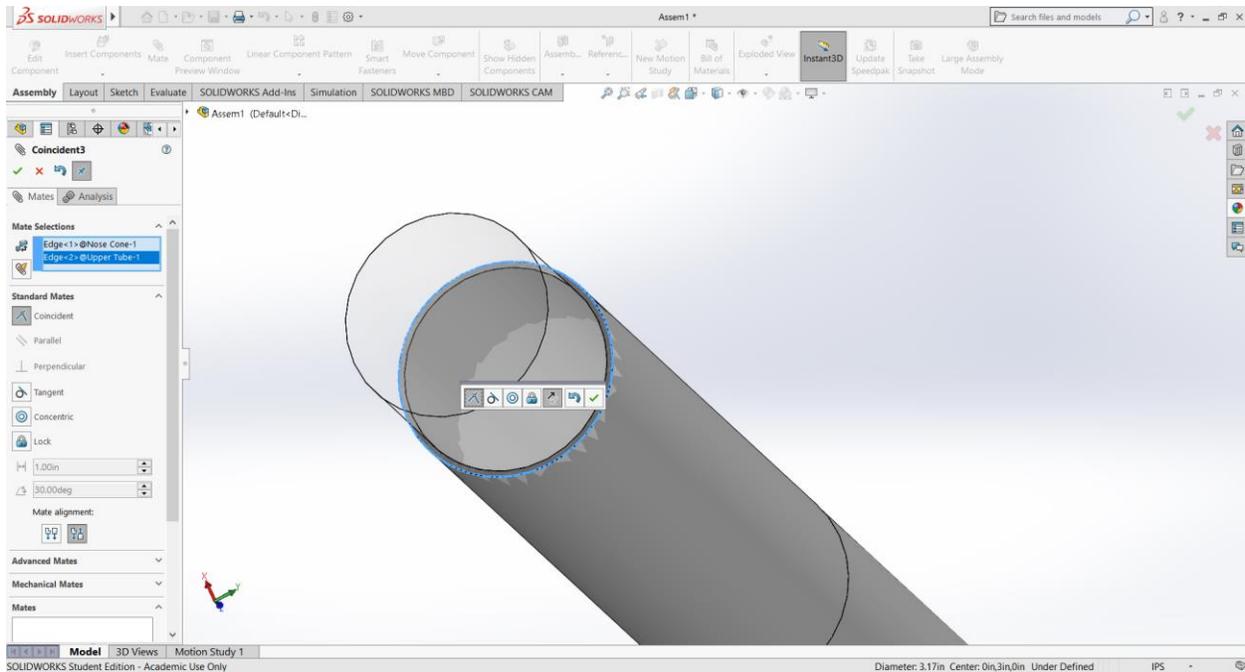




Step 4: Nose Cone

- select mates as shown

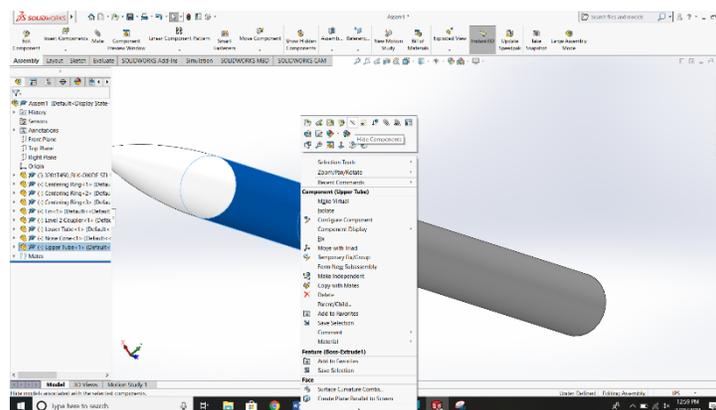


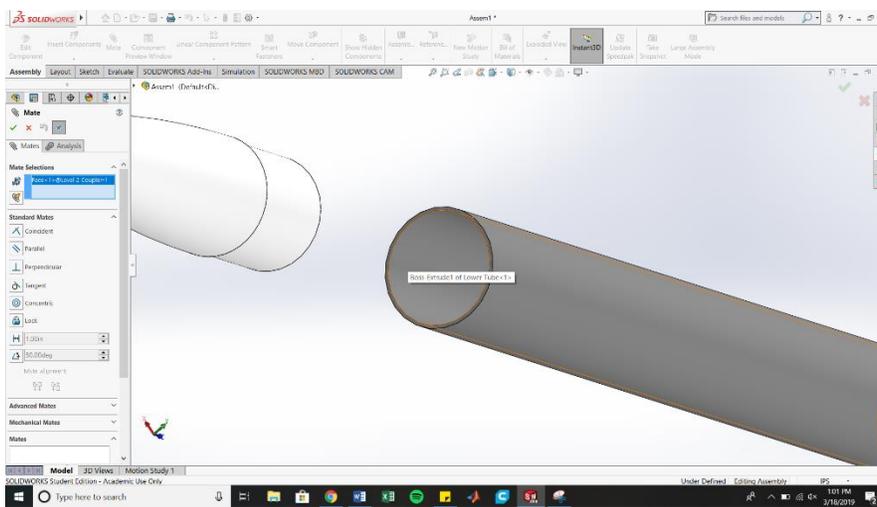
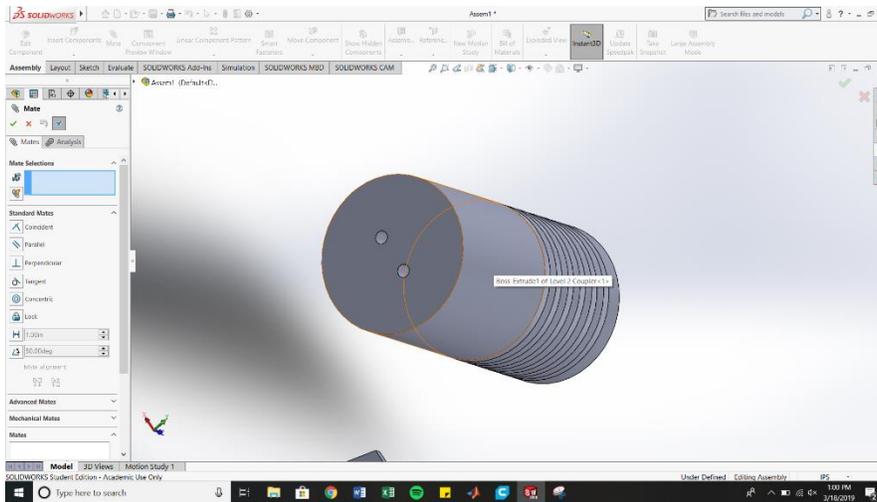


- if the nose cone is upside down (like shown), click the double arrows to flip it

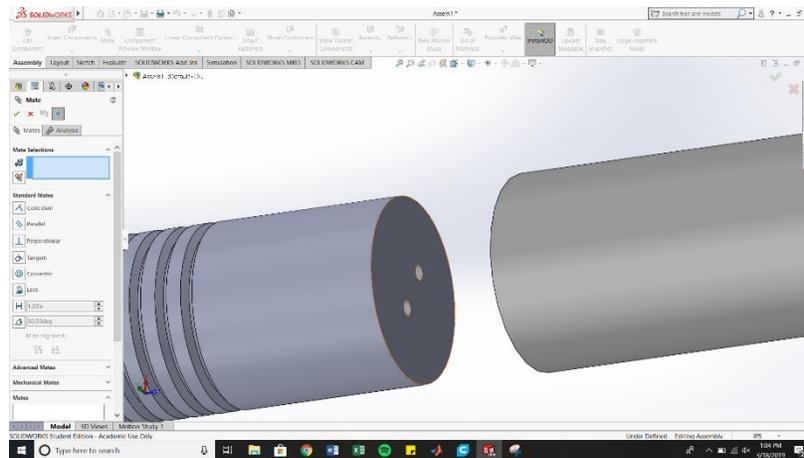
Step 5: Coupler

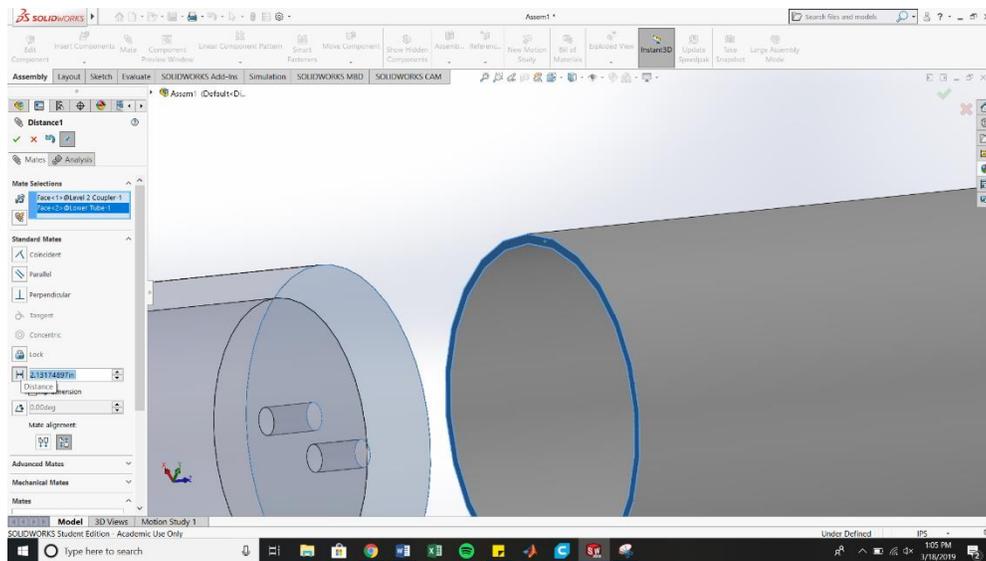
- start by hiding the upper tube
 - right click on it and select the eyeball
- Select outer face of coupler and inner face of bottom tube to make them touch
 - if the coupler is backwards, select the double arrows in the center to flip it
 - if the coupler is further down the tube and won't pull out without moving the entire model, right click the tube and select fix. The coupler should move out now if you drag it



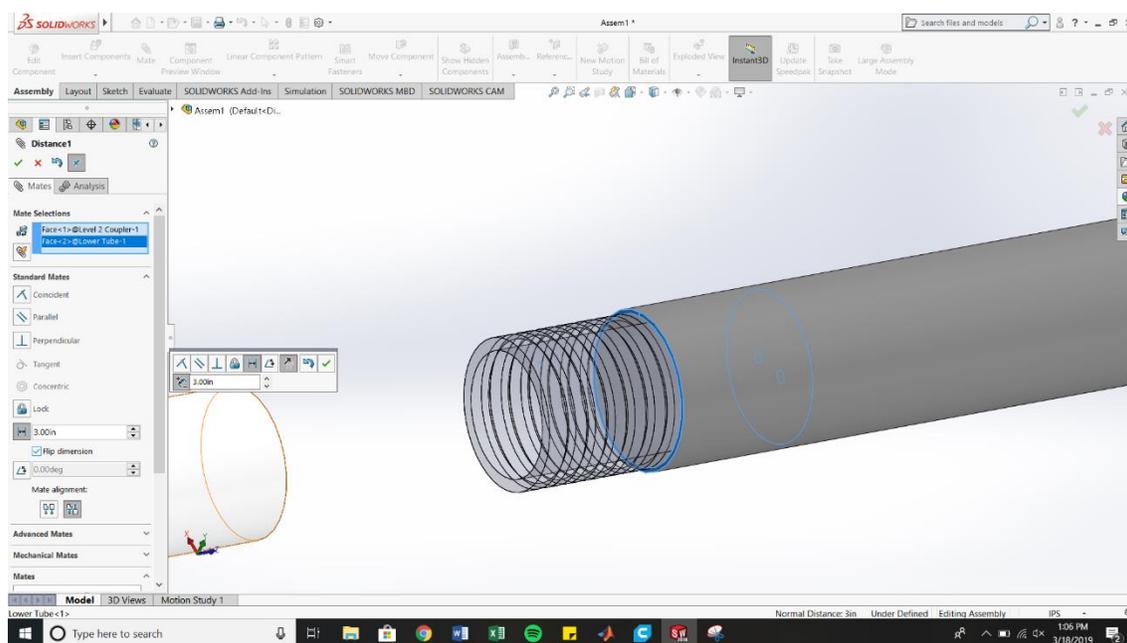


- Next, select the face of the coupler and face of the tube

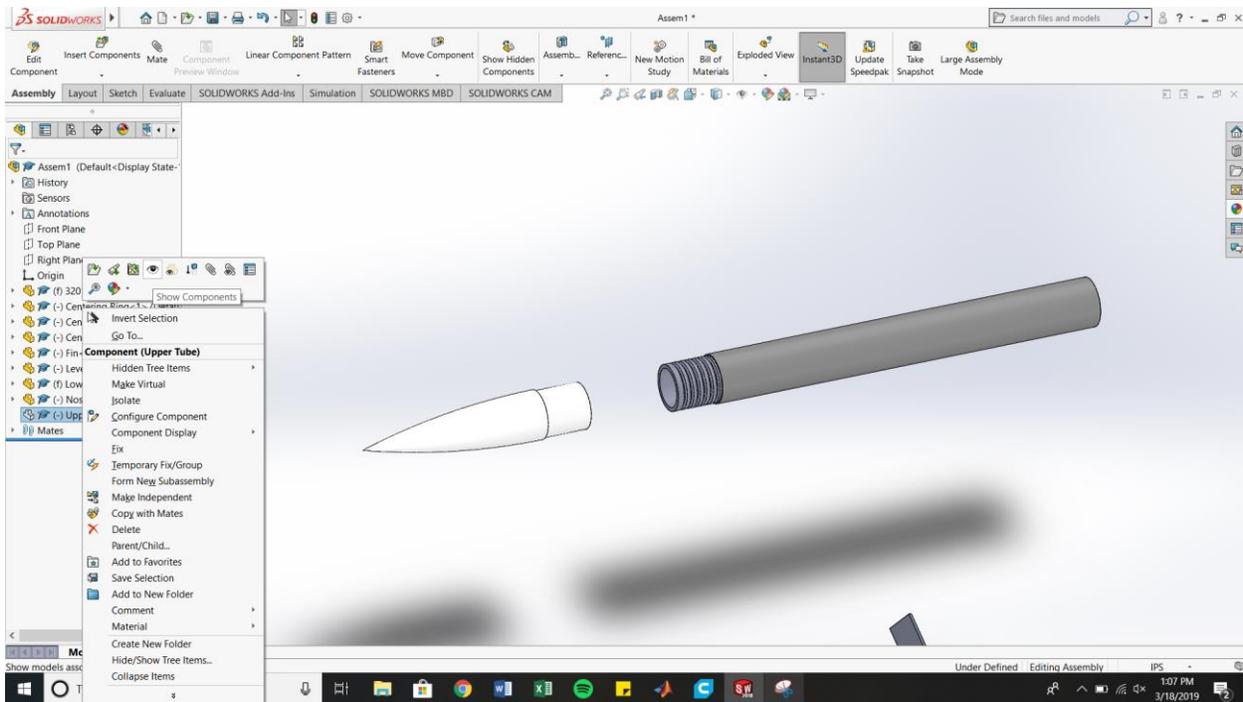




- On the left, click the arrows that look like dimensions to select a distance constraint. The couple should be halfway in each tube.



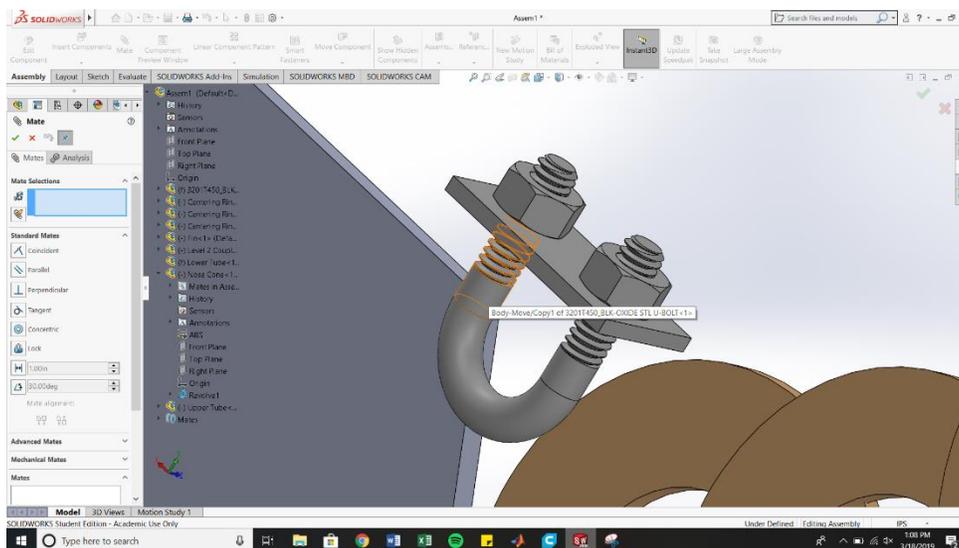
- Flip the dimension if the coupler moves in the wrong direction by checking the box

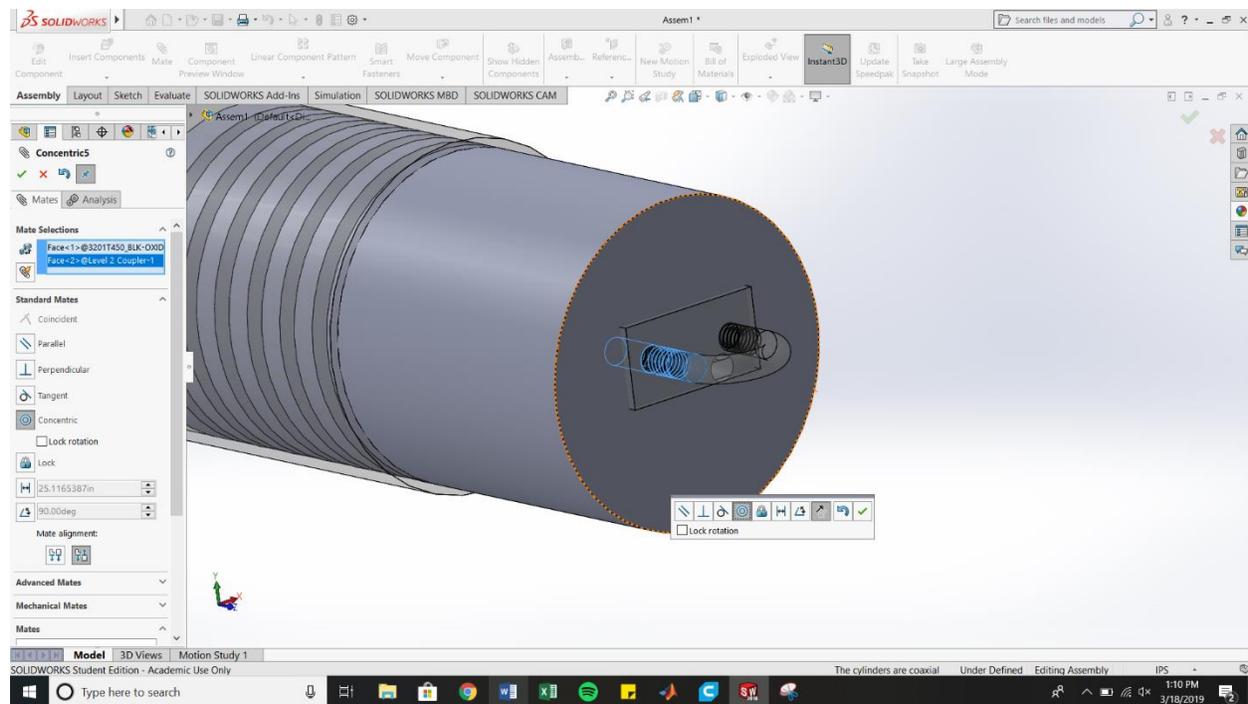
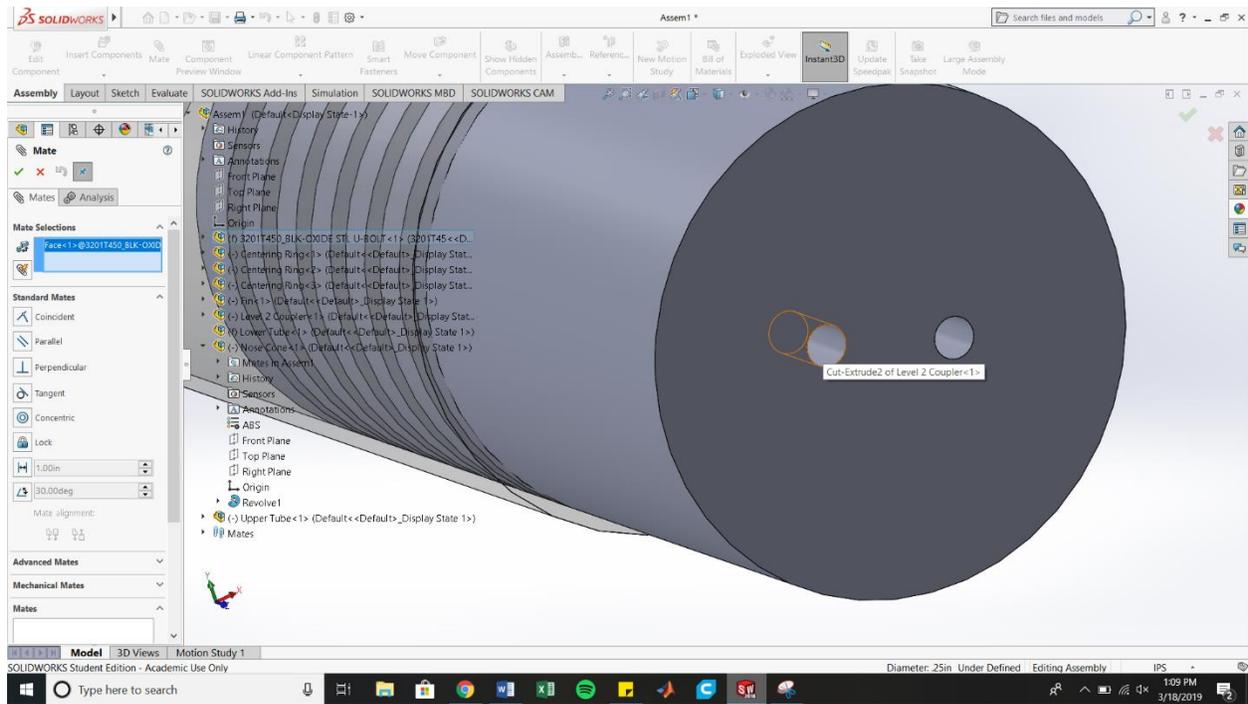


- You can view your tube again by finding it in the build tree, right clicking, and selecting the hide option again to unhide it

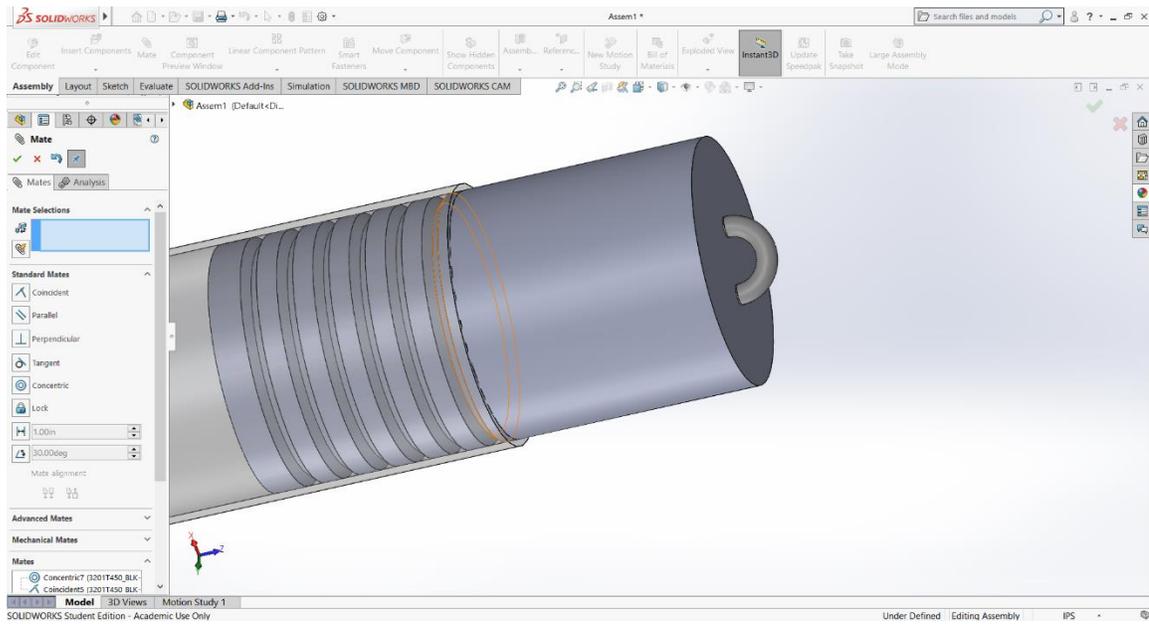
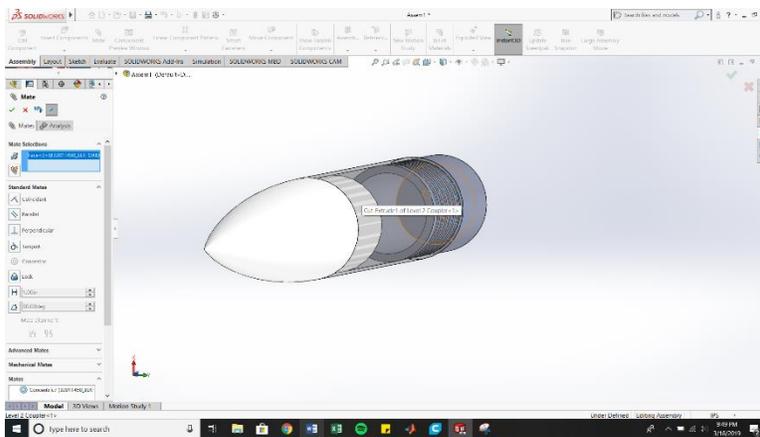
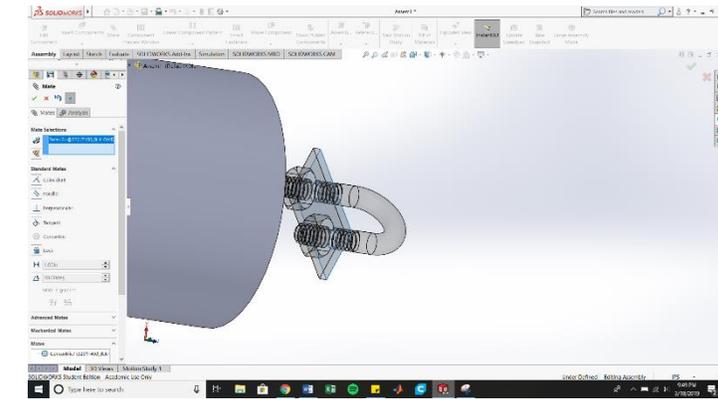
Optional Step 6: U-bolt

- I found this U-bolt on mcmaster carr. It fit the specs I needed
- Start by mating both cylindrical pieces on the bolt to the holes in the coupler



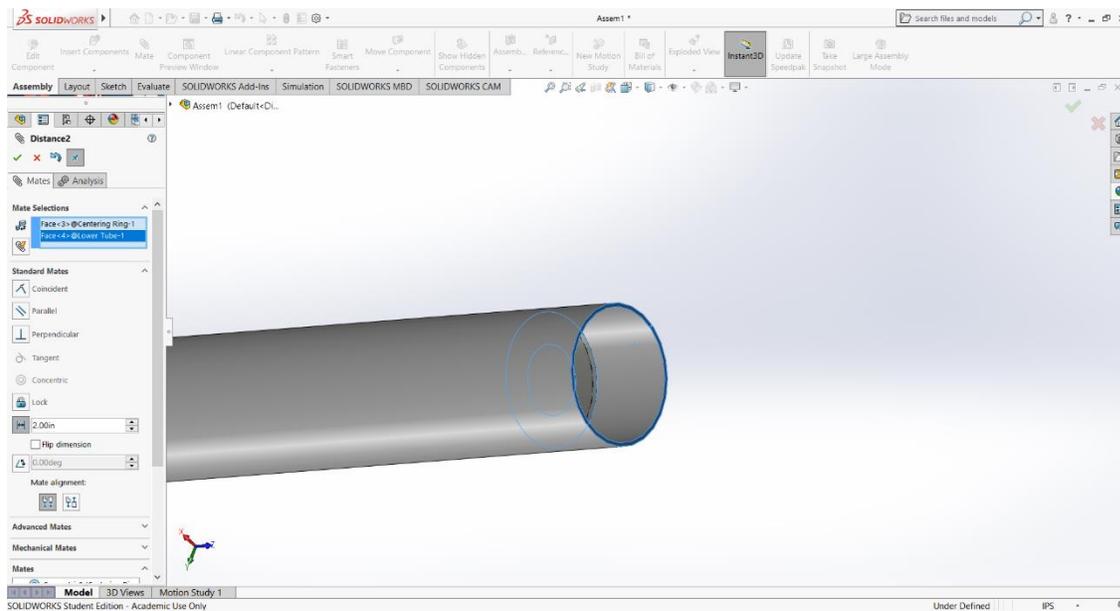
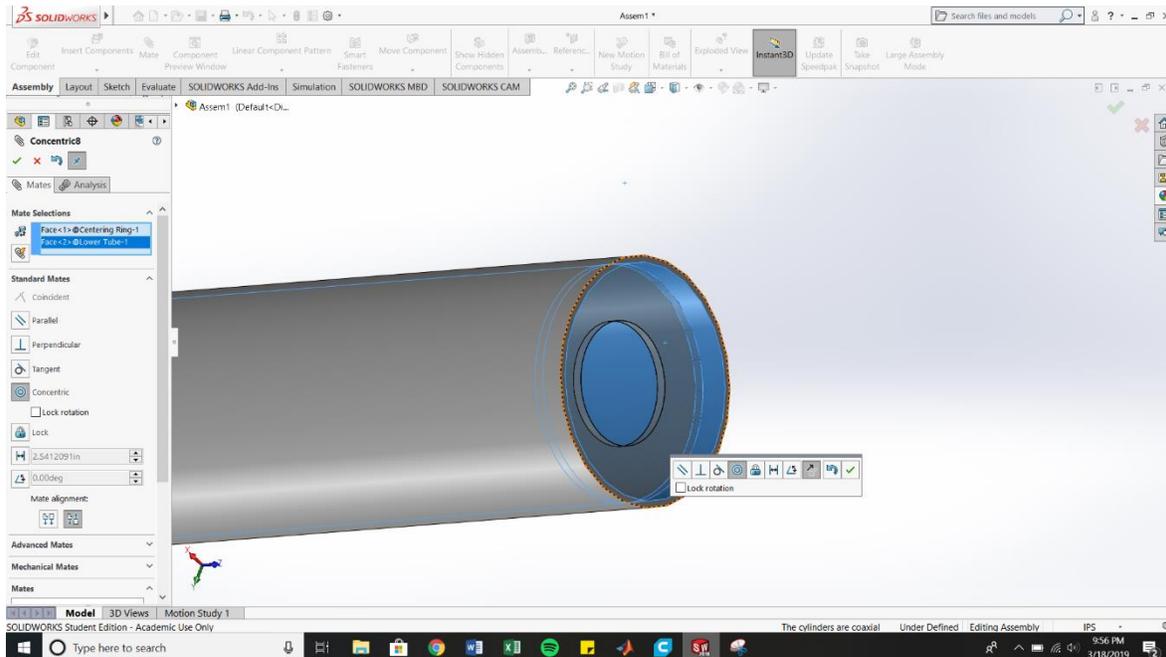


- Next, mate the plate to the inside of the coupler to prevent it from sliding



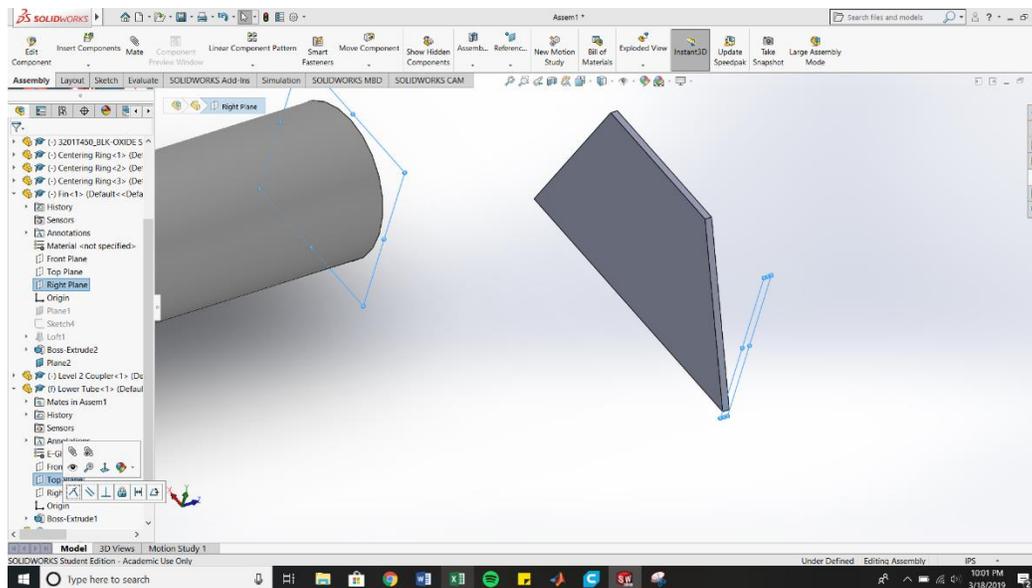
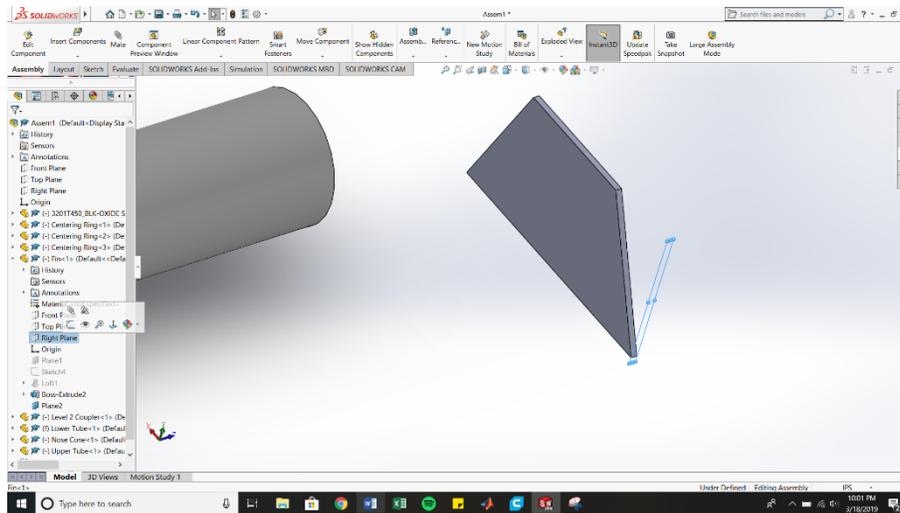
Step 6: Centering Rings

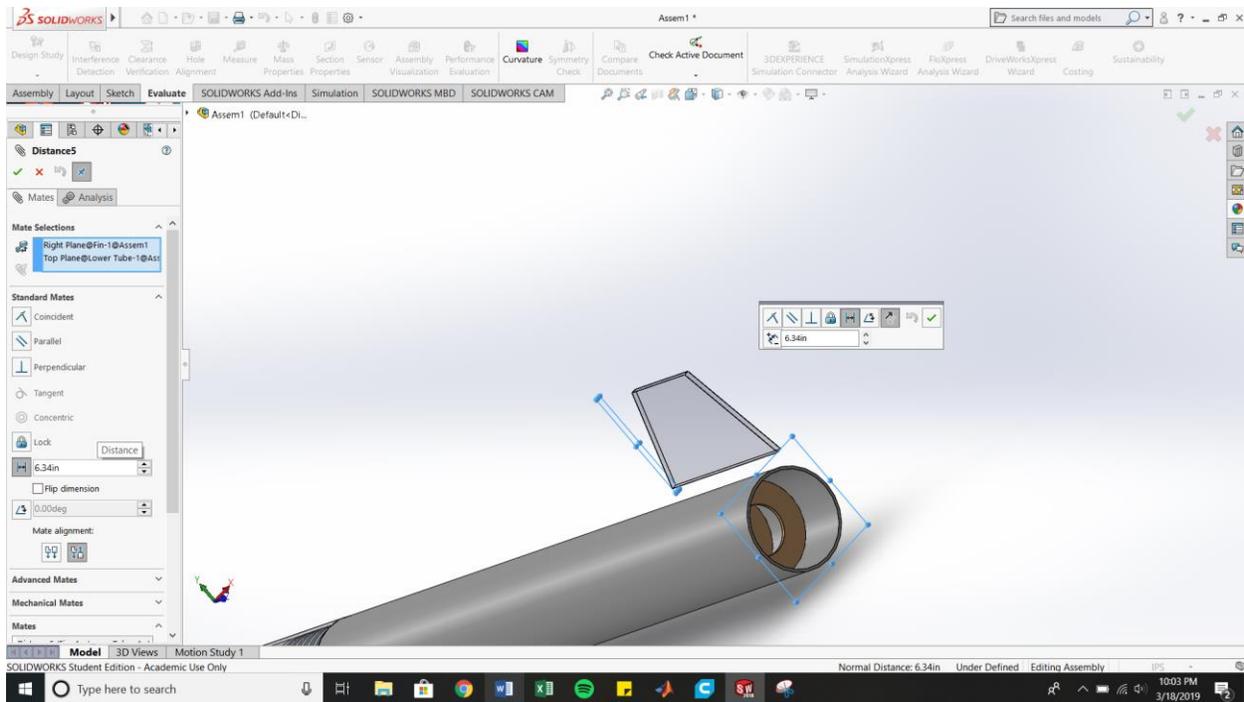
- Mate the outside of the rings to the inner walls of the tube
- Mate the face of the centering ring to the bottom of the tube, then select distance to set the distance from the edge
- Repeat for each centering ring



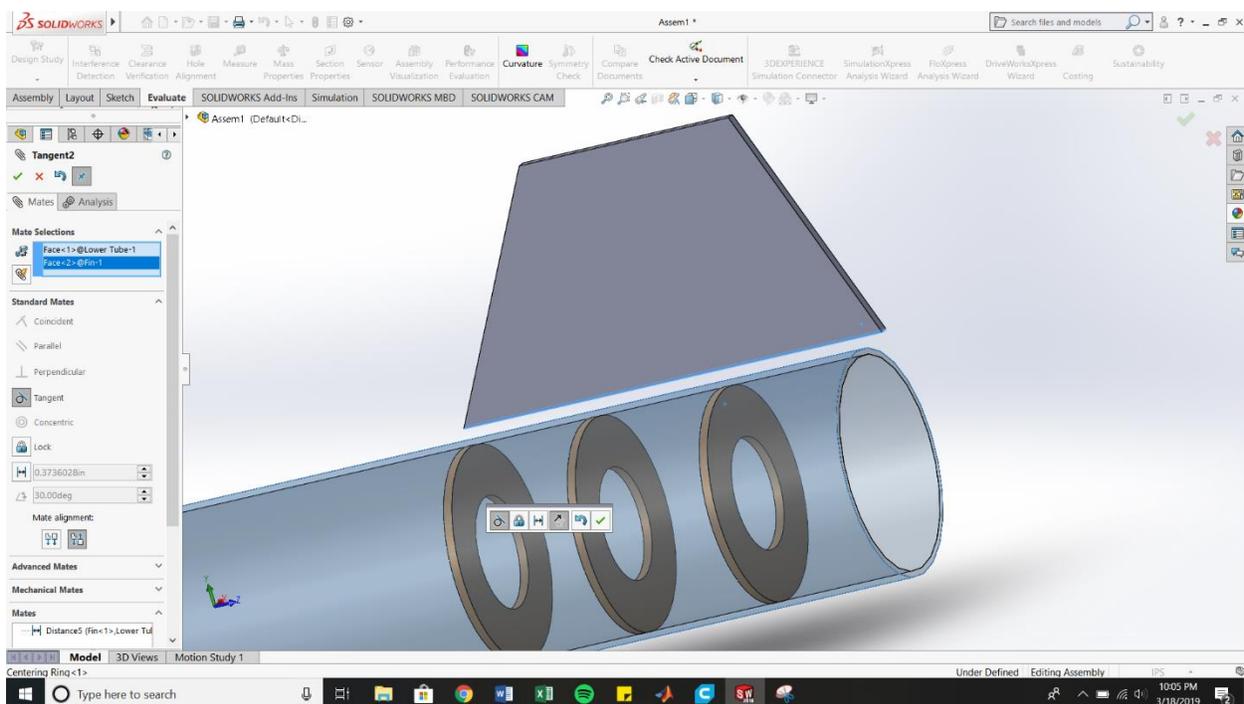
Step 7: Fins

- Fins can be tricky and sometimes not work
- When things do stick properly, I tend to use cheat codes to accomplish my goals
- Start by finding the back plane of the fin and mating it with the back plane of the tube
 - Make the mate a distance mate
 - Set the distance to the length of your fin

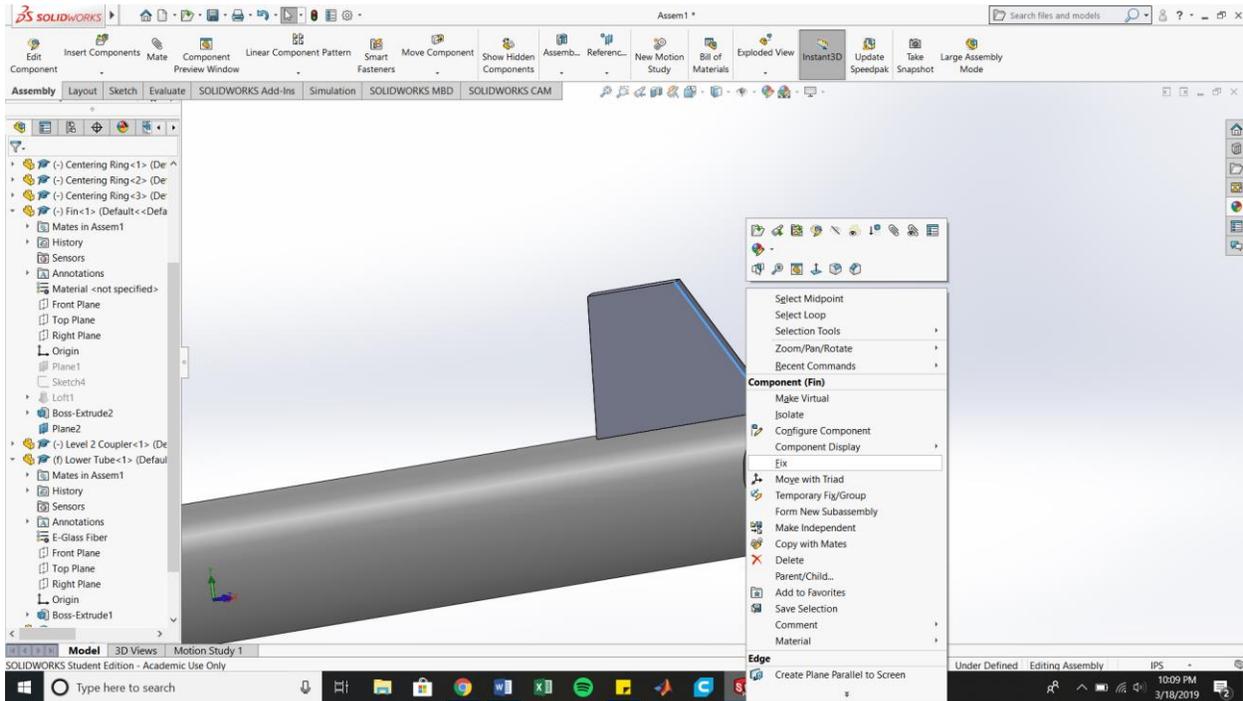




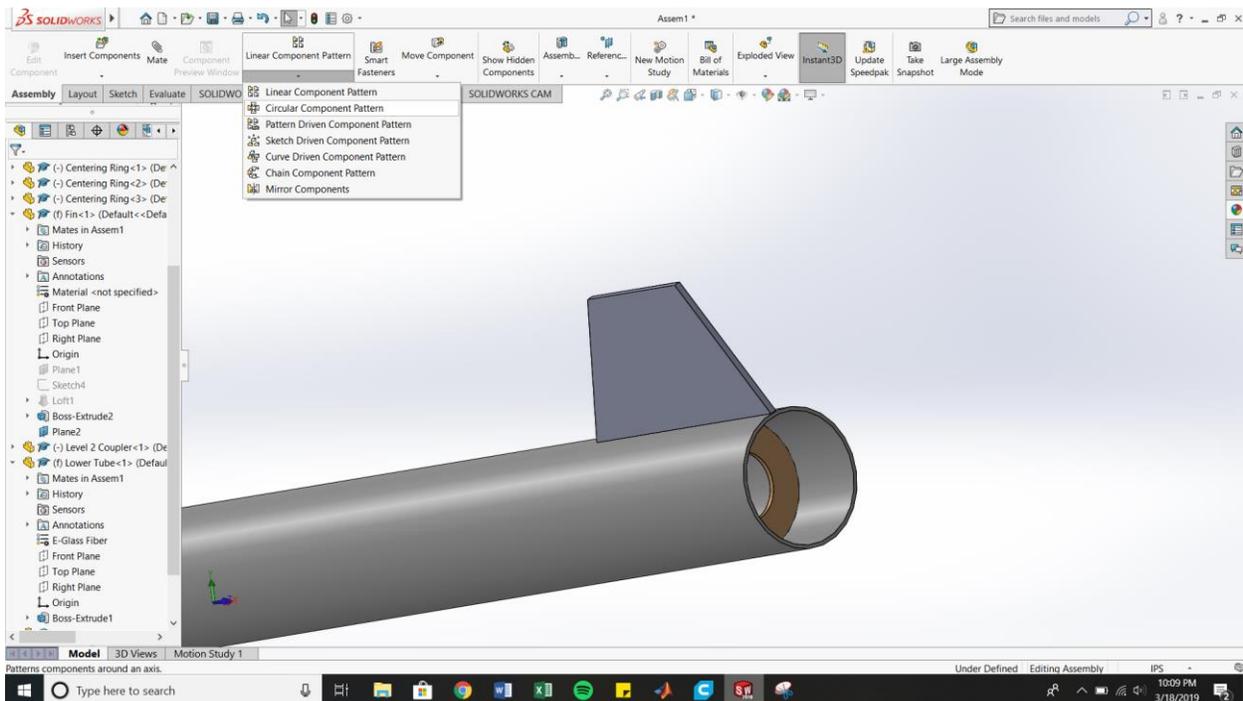
- next, mate the bottom of the fin to the body of the tube to make sure they stay in contact



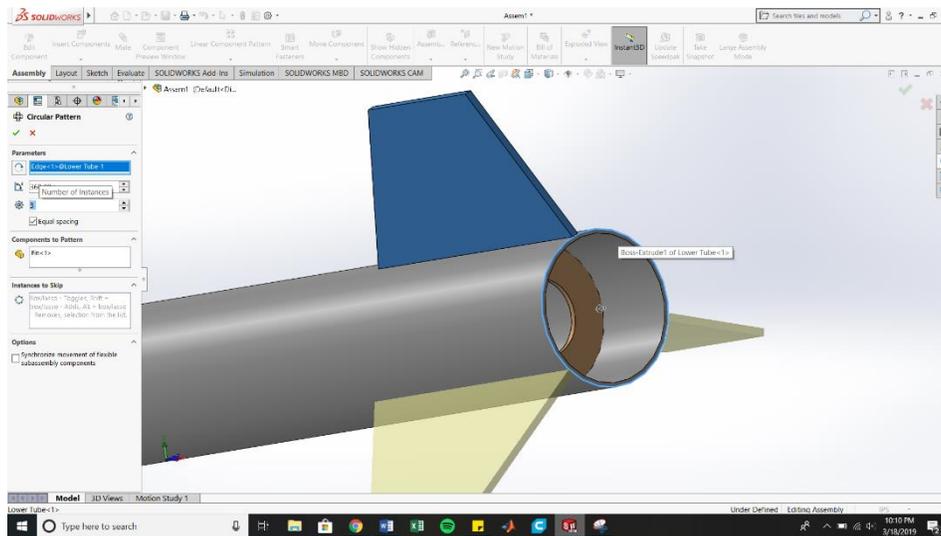
- Next, drag your fin till it lands on the rocket body.
- Once the fin is on, right click it and select “fix” to prevent the fin from moving ever again



- Lastly, under assembly, go the linear pattern button and select the drop down arrow
- Choose circular pattern



- Do the settings as specified
 - Select a circle to rotate about
 - In my case, I chose the circle at the end of the tube
 - Select equal spacing
 - Type in the amount of fins you desire



Step 8: Glory!

- The main rocket body is now complete
- All that's left to add is parachutes, avionics, and motor tube, but I'll leave that to you to figure out!
- I hope this helps!

