

Step 3: Designing Adapter for Your Vacuum

You only need to complete this step if you are creating the Wheelchair Tire Scrubber with your own vacuum rather than the Brigii Vacuum.

In this step, you will be designing an adapter where one end inserts into the vacuum shroud/chamber at the bottom of the Wheelchair Tire Scrubber and the other end connects to the hose on your vacuum. This adapter is for a vacuum hose with a circular opening.

The attached directions are for SolidWorks, but any CAD software will work to make this part.

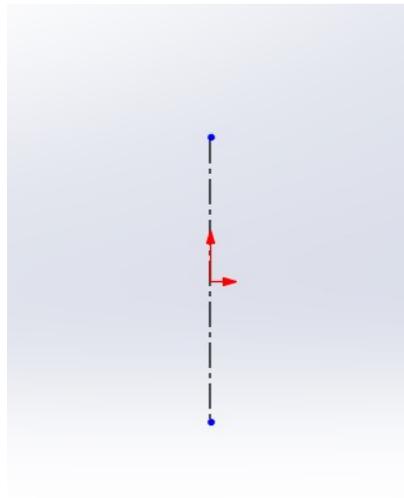
If you do not want to design this part, you can use duct tape to secure your vacuum hose to the shroud but making this adapter will allow you to easily switch between using the vacuum with the wheelchair tire scrubber and using it for other things.

Tools

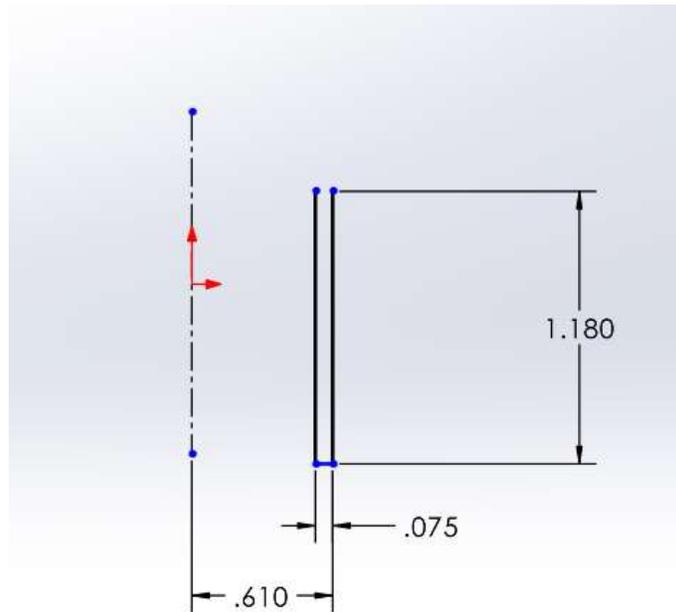
CAD Software

Calipers

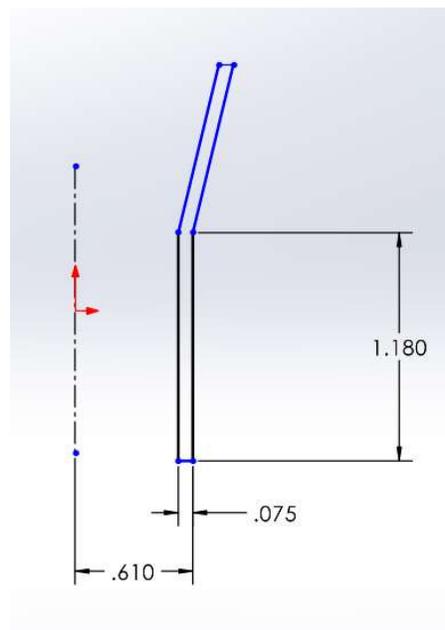
1. Set units to IPS (inches-pounds-seconds).
2. Select a plane and start a Sketch.



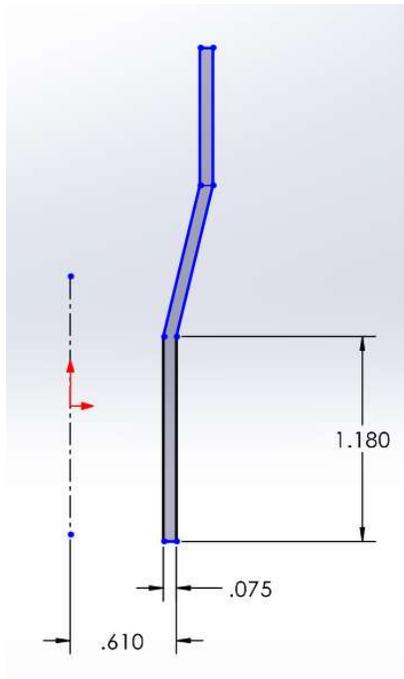
3. Draw a construction line vertically through the origin. We are creating a revolved part, so this will be the axis of rotation.



4. Create the geometry shown above. This is the part of the adapter that will slide into the opening on the vacuum shroud.



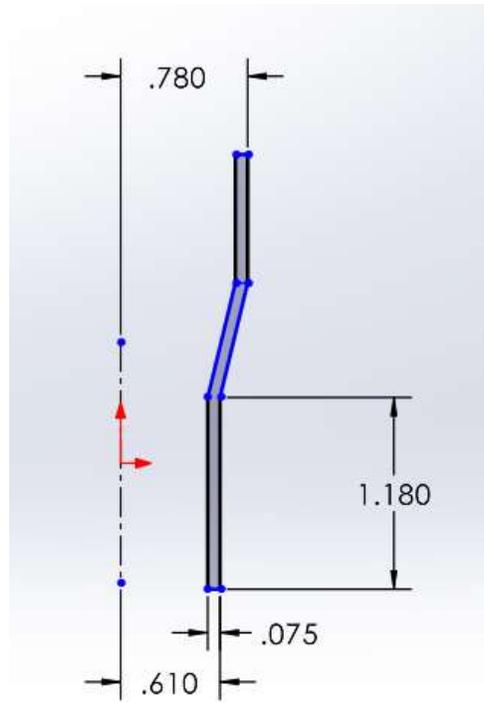
5. Draw two parallel angled lines above the shape. Connect them with a horizontal construction line at the top. Set the length of that construction line to be equal to the length of the bottom edge.



6. Add the two vertical lines and connecting horizontal line shown above. This is the part of the adapter that attaches to a vacuum of your choice with a round hose.



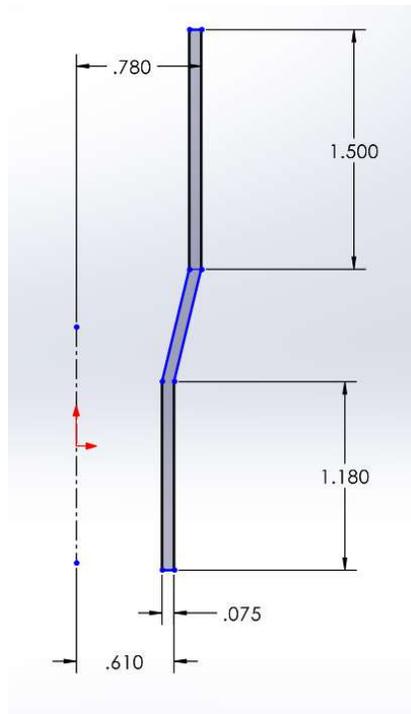
7. Measure the inside diameter (in inches) of your vacuum hose using calipers. Subtract this value by 0.02 inches and divide it by two.



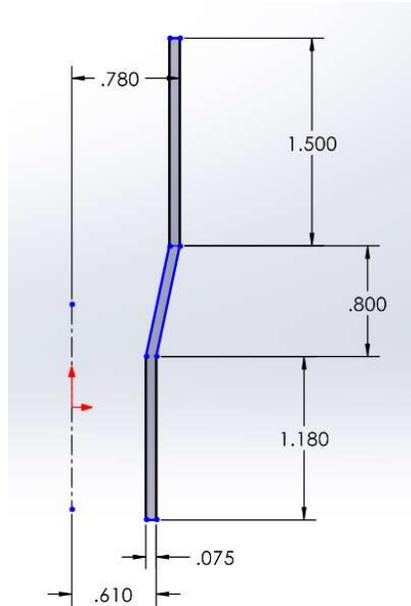
8. Make the dimension spanning from the axis of rotation to the outer edge of the top portion equal to the resulting value from step 7.



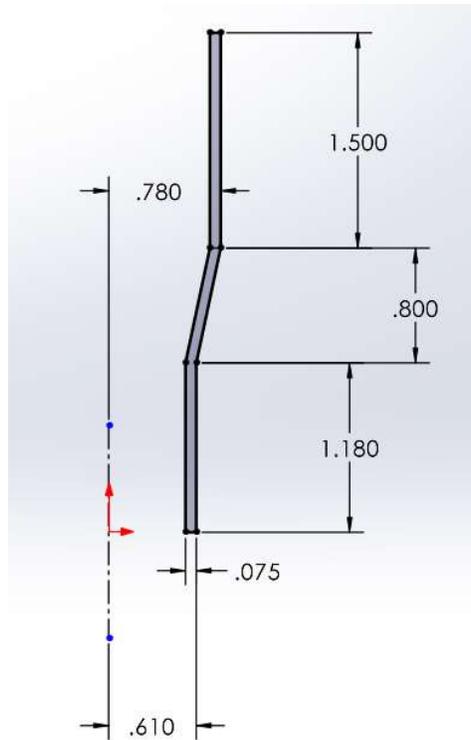
9. Measure the length of your vacuum hose.



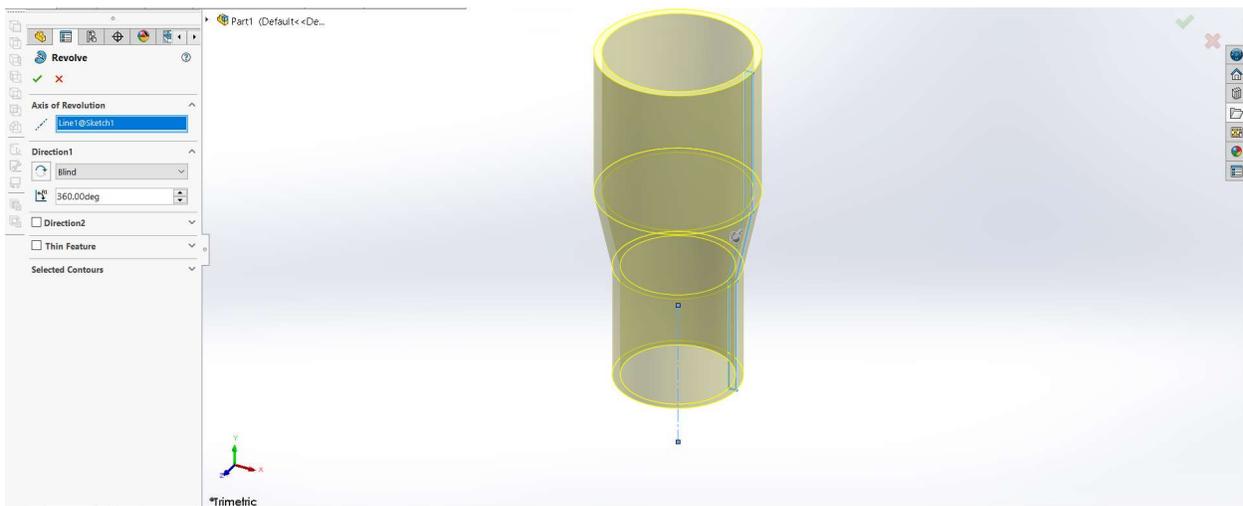
10. Make the height of the top portion of the sketch equal to this value.



11. Set the height of the middle section of the sketch equal to 0.6 inches.



12. Make the bottom edge and the origin coincident. Ensure your sketch is fully defined.



13. Exit the Sketch and go to Features > Revolved Boss/Base. Select the vertical construction line as the axis of rotation and ensure the revolve is 360 degrees. Click OK.



14. Your part should now look like this. Save the file as an .stl file and 3D print it in the next step of the Instructables along with the other parts.