

Practice Exercise – Vacuum Nozzle

Sketched Features, Essentials for Parts and Assemblies

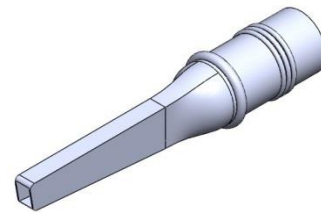


Summary:

In this exercise you'll use a set of provided sketches to create a vacuum nozzle model. To make sure the nozzle is created accurately you'll need to make sure to use the correct sketch profiles for each feature.

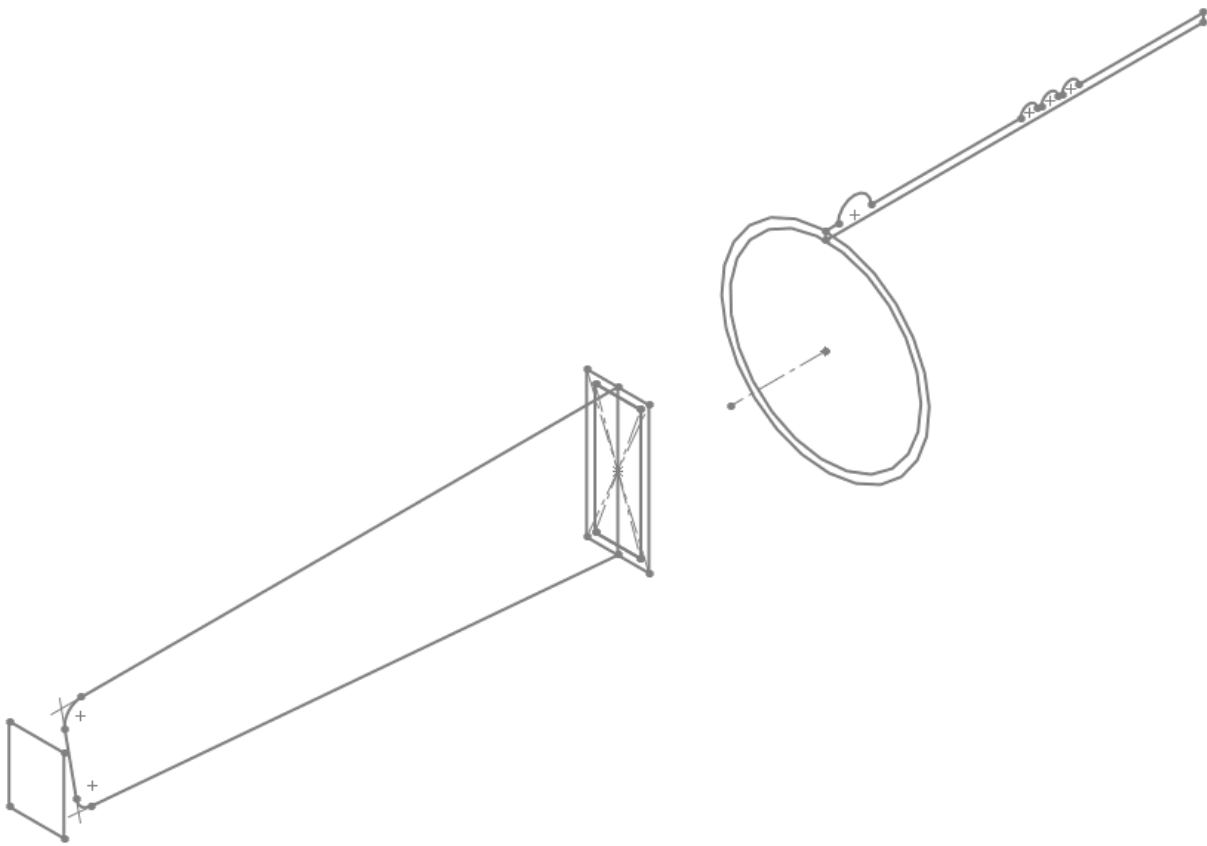
Reference Lessons:

- [Extruded Boss](#)
- [Revolved Boss](#)
- [Loft](#)



Instructions:

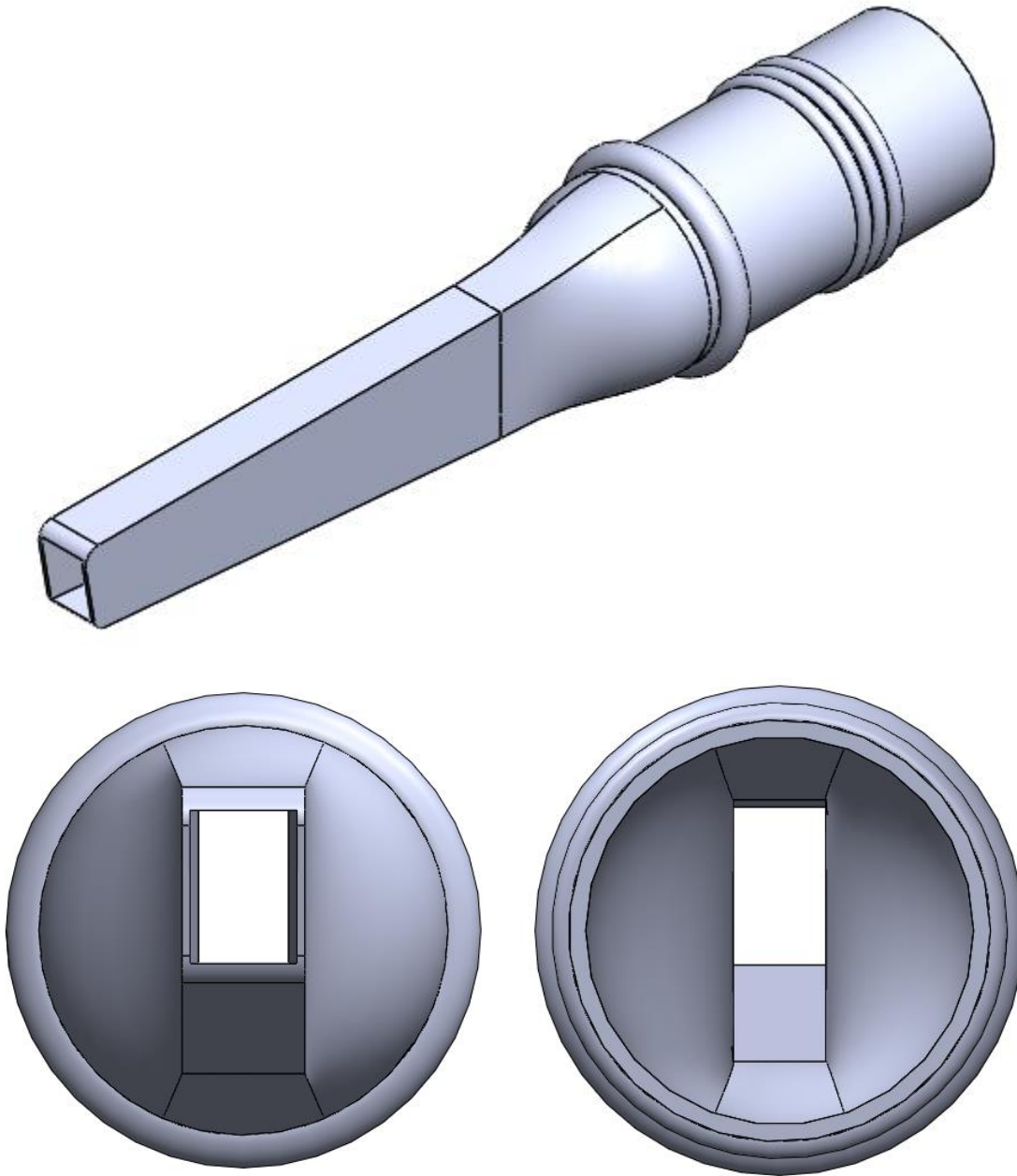
1. Download and unzip the part file [here](#).
2. Open the "Vacuum Nozzle.SLDPRT" file in SOLIDWORKS.



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3. Using the provided sketch profiles create the hollow geometry shown below.



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Final Result:

If you completed the exercise successfully, your result should look like the image shown below:

