Project DLA: SolidWorks Independent Study (Soil Sensor Manifold)

Student: Tate Putman   **Instructor: Yuezhou Wang**

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JUSTIFICATION

Background Information

It is valuable for an engineering student to learn CAD modeling, specifically SolidWorks, for multiple reasons. CAD modeling is a widely sought-after skill in the engineering industry because it allows for the user to translate an idea or sketch into a three dimensional prototype. SolidWorks specifically is valuable as it is widely considered the industry standard and by far the most widespread CAD modeling software in use. In addition to 3D modeling, it has extensive simulation capabilities including Finite Element Analysis (FEA). Although simulations are not within the scope of this technical elective, it is important to learn the basics of SolidWorks in order to be able to eventually use the simulation aspect of the software to its full capability.

Learning Outcome Justification

The Scope of Work in this specific design was to design and print a manifold that would connect the three solenoid valves as intakes, and lead to a single output going to the gas chamber. The part had to be within certain size constraints, as detailed later in this DLA.

I was the only team member who worked on the design of this part, and therefore all of the work shown in the FDR in this area represents my personal technical learning.

This activity withing the design project meets many of the "student learning outcomes" of this advanced technical competency. Every specific learning outcome in the syllabus is listed below.

A brief description of how the working knowledge of each outcome is shown in this project is then described.

- Sketch Entities
- Sketch Tools
- Sketch Relations
- Boss and Cut Features
- Fillets and Chamfers
- Patterning Tools
- Dimensioning
- Feature Conditions
- Mass Properties and Materials
- Inserting Components
- Standard Mates
- Reference Geometry
- Drawing Basics

The **sketch entities** are used in the basic sketch of the sweep profile, specifically the elliptical arcs. But all basic components of the sketch are considered entities. **Sketch tools** such as sketch snaps and "stretching" the sketch were both used in the preliminary sweep sketch as well.

There is a **sketch relation** ensuring the sweep and the sweep base are perpendicular to each other. Midpoints are selected with dynamic highlighting in the sketch as well.

All of the major components of this part are extruded as a **boss feature**, and the hollow inside is a **cut feature**. The circular base is the **reference geometry** by which the shape of the solid sweep is defined. While extruding the inlet valves, the "up to next" **feature condition** was used. **Fillets** are added around the inlet extrusions to increase the robustness of the part and reduce the stress concentration. See below for an illustration of fillets in a scenario similar to this part (Figure 1).
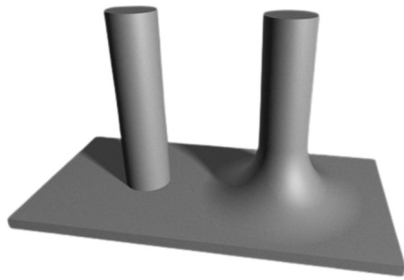
Figure 1 – Fillet vs no Fillet[2]

To ensure the part meets the dimension requirements, **smart dimensioning** was used to define the part and stay within measurement constraints while being edited.

At the end of the document, details of mass and volume are included from the **mass properties** tool on SolidWorks, and a **part drawing** is included as well at the end of the document.

As this part is not an assembly, the concepts of **standard mates** and **inserting components** are not used. A standard mate is a constraint which defines how parts of an assembly are connected. Inserting components is the process of how each part is added into the assembly. **Patterning tools** were not used in the creating of this part either, because we did not have a repetitive design. It would have been an ineffective use of time to use a patterning tool to create the two circular bases for the inlet extrusions.

MAIN BODY (Content from FDR)

(note: this content structure is formatted to look more DLA-like, but the content itself is the same as the FDR)

Design Overview and Constraints

To connect the three airlines from the probes into a single airflow into the gas chamber, a manifold had to be designed. The manifold needed three inlets which fit into the solenoid valves themselves. The outlet outer diameter needs to fit airtight within the tubing. Specific dimensional constraints are in the following section.

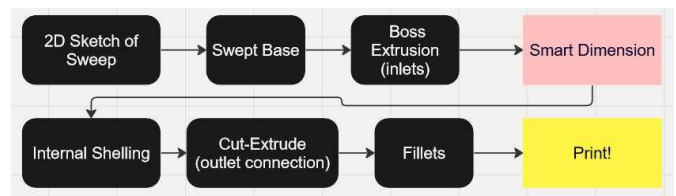A general flow chart of the process of modeling this part is shown in Figure 2.

Figure 2 - Manifold Modeling Flow Chart (miro.com)

Dimensions:

The first step in designing a part such as this is to measure the dimensions of the parts which it will connect to, in this case the solenoid valves as well as the plastic tubing which connects to the gas chamber.

The inner diameter of the plastic tubing measured to be 4.5 mm.

The inner diameter of the solenoid connections measured to be 6.4 mm.

The three solenoid valves are designed to sit side-by-side in the Pelican case. Because the manifold is designed to be a rigid part, the distance between the center of each manifold inlet valve has to equal the distance from the center of each connection point on the solenoids. This measurement was 24.0 mm.

The area in which the manifold had to fit within an area of 100.0 mm, but the smaller the length of the manifold the better in order to allow space for other components to fit.

Sketch:

To create this part, a new sketch is created on SolidWorks. First, to make sure all units of measurement are the same, the units are set to metric "MMGS" on the bottom right corner of the screen. The sketch is based off of a reference plane called "Plane 1", which is parallel to the "Front Plane".

The first step in the actual sketching of the part is to create a sweep. A profile of the sweep is sketched on the plane, which will represent the outer diameter of the manifold tubing. This is a circle with a diameter of 6.4 mm.

Next, the sweep was created, starting with a perpendicular relation to the profile. The 90° curves with a radius of 10.0 mm are to create a smoothed-out bend to reduce pressure loss in the tube compared to a right-angle. The total length of 60 mm is the minimum length for all of the intakes to be evenly spaced while still being connected to the horizontal part of the sweep. See the sweep in Figure 3 below.
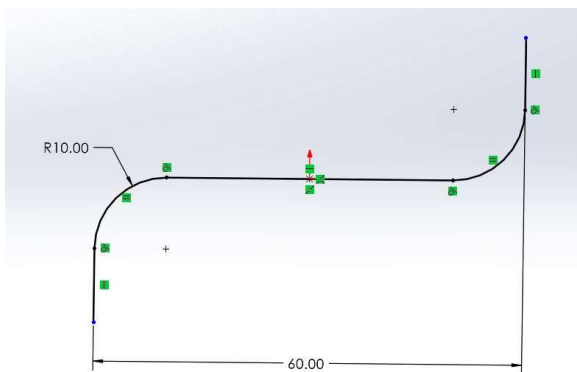

Figure 3 – Sweep

The base is then swept around the "sweep" sketch using the "swept boss/base" tool. This results in the 3-D "skeleton" of the model, shown in Figure 4.
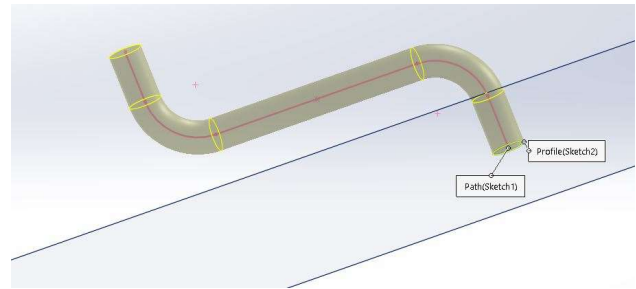

Figure 4 - Swept Base

After this initial skeleton is created, the next step is to extrude what will become the other two inlets. These bases are sketched 24.0 mm apart on Plane 1 and extruded to the main manifold using the "up to next" feature condition in the Boss-Extrude Menu. See Figures 5 and 6.
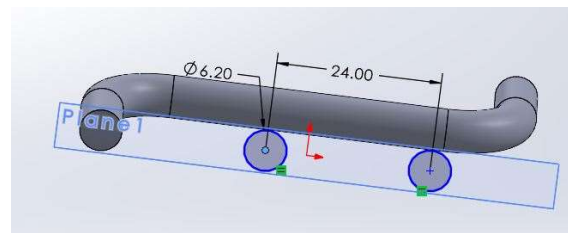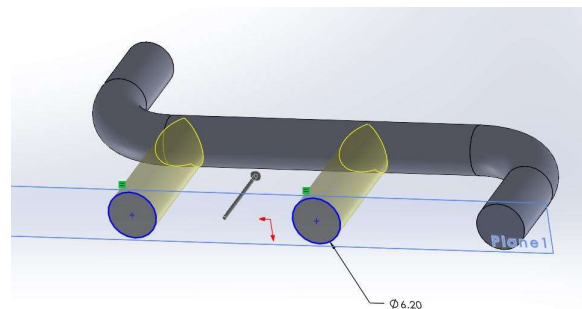

Figure 5 – Dimensions

Figure 6 – Boss Extrusion

Next, in order to hollow out the manifold structure, the "shell" tool is used. Each circular base is selected except for the top, as the top needs to have a smaller diameter than the others for the tubing to attach. The parameter for "thickness" is set to 1.2 mm. This dimension is chosen due to it being on the "high-end" of the minimum suggested wall thickness for a PLA part[1]. See Figure 7 for a visualization of how the bases were selected and then shelled out.
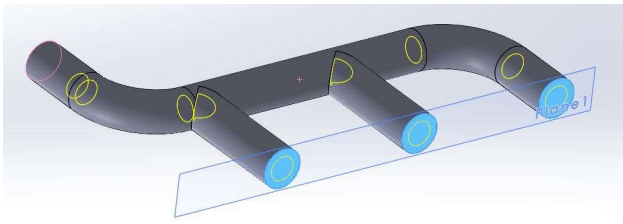
Figure 7 - Shell Feature

The final major modification to this part is the outlet, which needs to have a smaller diameter (4.6 mm) to fit inside the plastic tubing. First, 8 mm of material was extruded from the top of the outlet face downward, with a diameter of 2.6 mm. This will act as the inner tube of the outlet through which gas will flow. After this, 7.0 mm of material with an inner diameter of 4.6 mm is also extruded from the top of the outlet face. This results in a outlet connection point with an outer diameter of 4.6 mm and 1.0 mm wall thickness. (The length of this connection point was changed from 5 mm to 7 mm over three iterations in order to better connect to the tubing.) See Figures 8 and 9 below for the above descried steps.
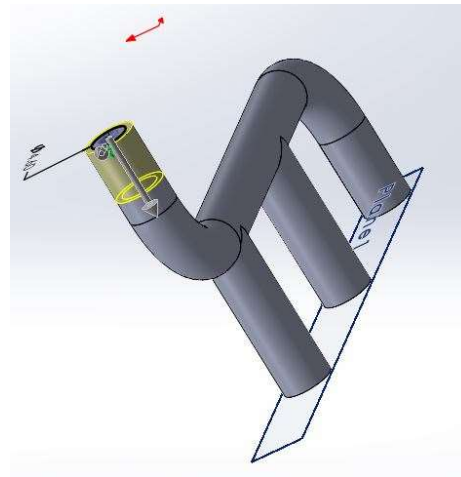

Figure 8 - Cut-Extrude 1

Figure 9 - Cut-Extrude 2

The final modeing step is to add fillets, which were added after the first print iterations in order to increase durability of the part. (This was not due to part failure, but to follow best design practices.) Fillets distribute the stress over a broader area than the would-be sharp "corners" of where the extrusions meet. The dimensions of the fillets are 2 mm. To add these to the model, click "fillet", and then click the boundaries to which you would like them to be added. Then, set the radii and click save. The locations of the fillets on the manifold are highlighted in blue on Figure 10 below.
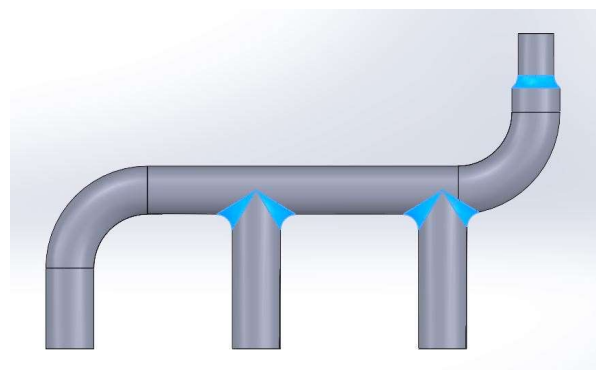

Figure 10 - Fillets

This concludes the modeling portion of the creation of this part. The technical drawing of this part, annotated with smart dimensions, is on the last page of this document. Please see Figure 14.

Printing Instructions

The .STL file of this manifold is available in the project GitHub. Download this .STL file and upload it to the 3D printer slicing software that corresponds to your printer. The Soil Team uses a Prusa 3D printer, so the file was sliced using PrusaSlicer.

Once the .STL file is uploaded to the slicing software, the first step is to change the print orientation to "upright", where the three inlet holes are resting on the printer bed (Figure 11).

The print settings should be set to a "Quality" setting (0.15 mm or 0.10 mm are both acceptable). The filament used is a "Generic PLA." Set the supports to "Support on build plate only" and infill to "80%". Once the settings are set, it should look similar to Figure 12, depending on your slicing software.
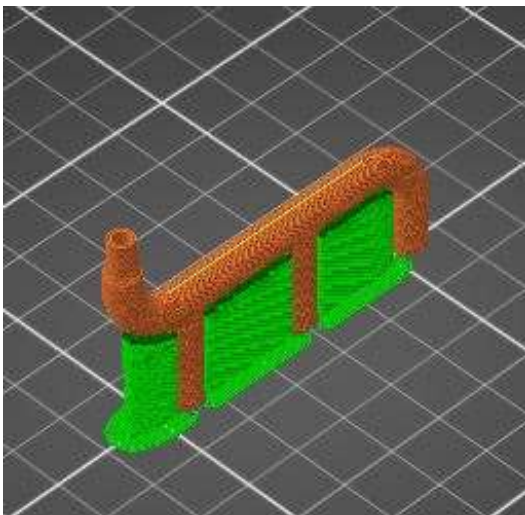

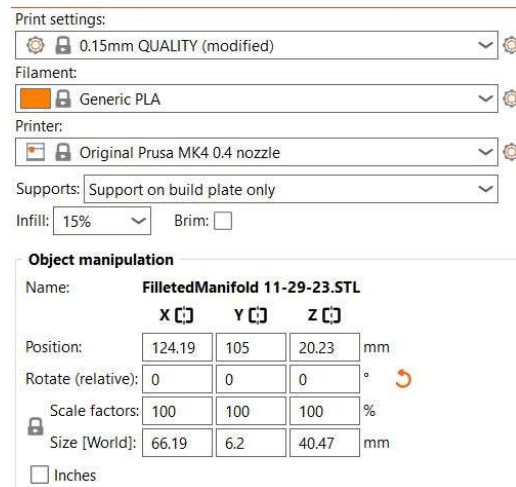**Figure 11 - Printer Bed Orientation (PrusaSlice)**

Figure 12 - Print Slicer Settings (PrusaSlice)

If the print settings are set correctly, press "slice now" to export the g-code file that can be uploaded to the printer. Move this file to a storage device that can be inserted into the printer.

Follow the instructions on the printer display to continue to print the part. Default settings were used when the Soil Team printed this part.

Mass Properties & BOM

Although not crucial for the purposes of this project, details relating to the exact mass and volume of material that this part takes up can be found by clicking on the "evaluate" tab, and then "mass properties" in SolidWorks. The manifold that has been created weighs 2.35 grams and uses 2,348 mm^3 of material.

Another interesting tool to use in SolidWorks is the Bill of Materials creator. If this project were using a parts assembly, it would be more relevant, but in case it is of interest here is the BOM generated for the single-part manifold (Figure 13).

ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Manifold (Soil Sensor Project)		1

Figure 13 - Manifold BOM

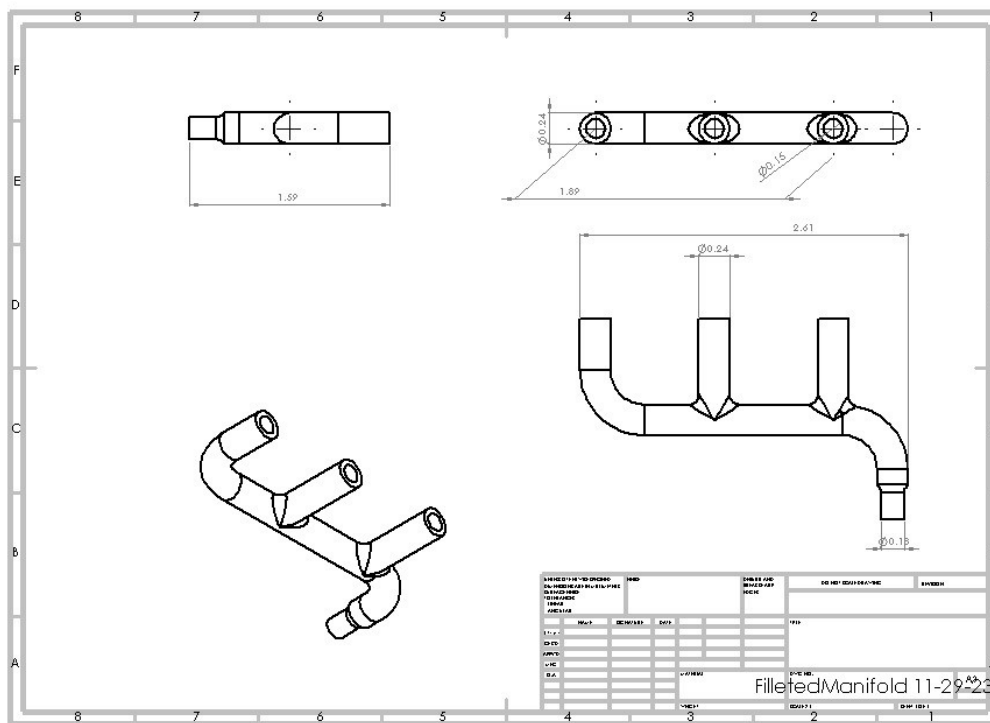# SolidWorks Part Drawing

Figure 14 - SolidWorks Drawing--Manifold

Relevant Sources

[1] *Wall thickness (3D printing): How to make it perfect*. All3DP. (2023, October 13). https://all3dp.com/2/3d-printing-wall-thickness-tutorial/

[2] *Fillet (mechanics)*. (n.d.). Wikipedia--Fillet (mechanics). Retrieved December 1, 2023, from https://en.wikipedia.org/wiki/Fillet_%28mechanics%29.