Project #1:

Solid Modeling a Poland Spring Water Bottle

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I. Overview

For this project, I chose to model a 16.9oz Poland Spring water bottle. It is one of the most popular water bottles that consumers purchase. It contains many curves that might be difficult to model at first. The major parts of the bottle include the mouth, thread, neck ring, neck, shoulder, the body, and the base. The label panel is also important to note as that is the region where the label of the bottle is placed. These parts can be seen in the free-hand sketch found in the back of this report (Fig. 15). In order to simplify the modeling in this project, I decided to ignore the label and lid to instead focus on the main bottle and its curves.

In this report, I will explain in detail how the measurement process was performed and how I approached the graphic modeling in SolidWorks. The final results of the graphic model with its engineering drawing will also be shown. Since it's impossible for the measurements and the model to be exactly the same as the original object, I hope that the model created will be close enough to the object as possible.

II. Procedure

a) *Measurement*

In order to get the overall shape of the water bottle, I decided to measure the circumference of the bottle at different intervals along its height. I first started by measuring the diameter of the base of the bottle by using a ruler passing through the center. For the curves around the bottle, I used a tape measure by wrapping it around the curve to calculate the circumference. After getting the circumference for each part of the bottle, I calculated the diameters for each by dividing each value by pi. The height in between each increment was determined by approximation with a ruler.

Fig. 1 – Measuring the circumference of a curve using a tape measure.

For the more detailed measurements such as the neck ring, I used a caliper to get the dimensions of the side view. A protractor was used for the base of the bottle and its indents where the angle between each point of the star-shaped figure was found to be 60 degrees. The unit for this model is in millimeters for better accuracy when it comes to getting measurement data. The hand-drawn dimensioned sketches including all the dimensions found, are included at the back of this report as figures 16 and 17.

An alternative method that I tried using for the indents, was a contour gauge. However, this failed to work because the plastic of the bottle was very flexible. When trying to use the gauge on the bottle, the plastic would deform under the pressure applied on the tool.

b) ***Modeling***

Initial Skeleton

The first step in the modeling process was to make a skeleton of the bottle using the calculated diameters of each circular cross section of the bottle, and the height of each from the base. I started by sketching the bottom-most circle of the base first on the top plane. I then added parallel planes from the top plane upwards for each circle, making sure the distances from the top plane were the same as those I measured.

On the front plane, I sketched a spline to use as a guiding curve for the shoulder of the bottle. Figure 2 below shows the initial sketches. The loft feature was used three times: the first to construct the body, the second to construct the shoulder, and finally the neck and mouth. Once that was completed, the shell feature was used to make the bottle hollow with a 1mm thickness (Fig. 3).

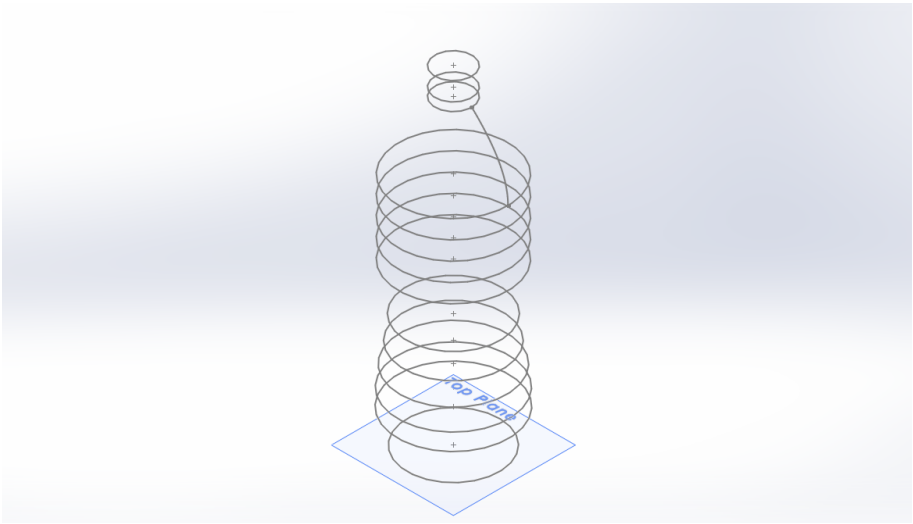

Fig. 2 – Initial sketches for the skeleton of the bottle model.

Fig. 3 – Model after applied lofts and shell.

Threading of the Top and Neck Ring

The next part in the modeling process was the threading. For this, I drew a trapezoidal sketch on the front plane making sure the point lines up with the top-most edge of the mouth, and then dimensioned it (Fig. 4). To get the pattern of the threading, I used the helix/spiral tool to sketch the path from the mouth with a pitch of 4mm and 2 revolutions (Fig. 5). By using the sweep feature on the sketches, the threading was obtained. Finally, for the neck ring I sketched a rectangle on the front plane lining up with the side of the neck. After dimensioning it, the revolve tool was used to create the completed ring around the neck (Fig. 6).

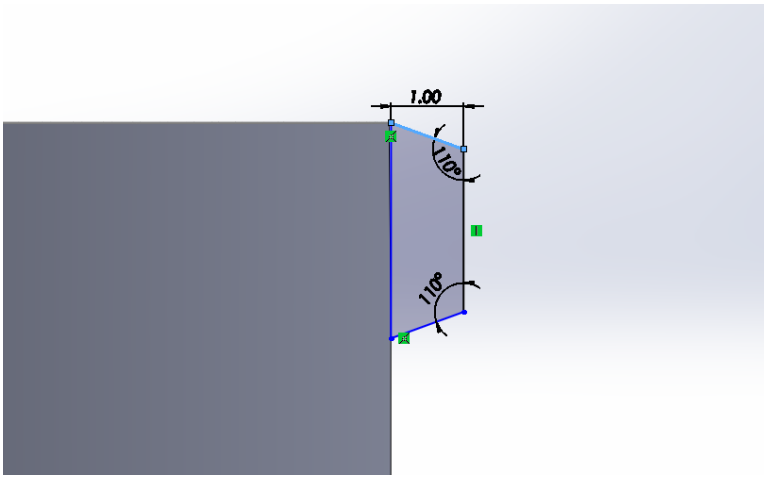

Fig. 4 – Trapezoidal sketch along the proposed thread area and mouth.

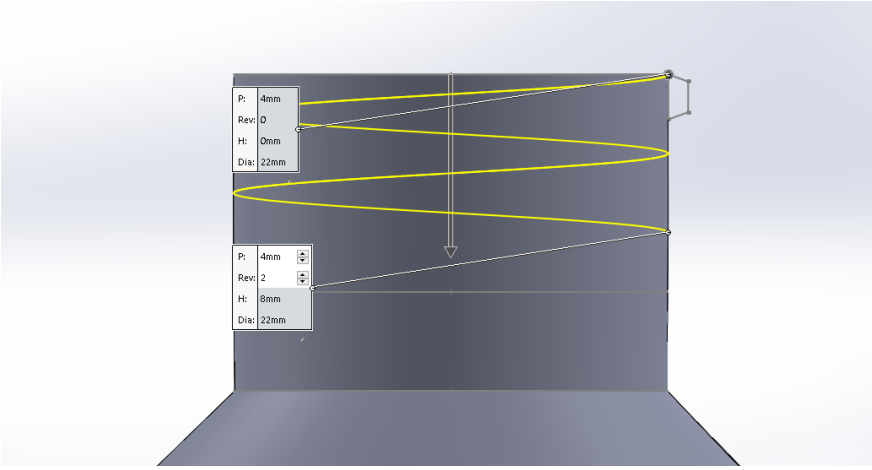

Fig. 5 – Sketch of helix to be used for threading path.

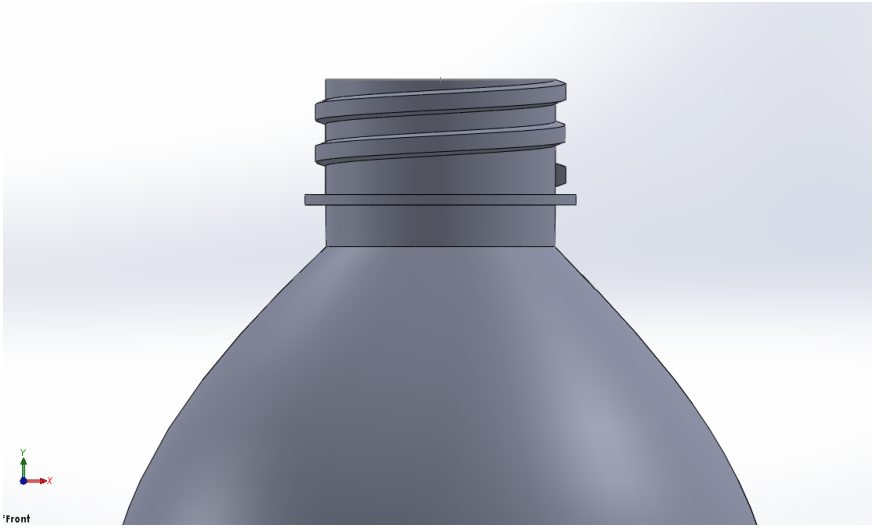

Fig. 6 – Front view of the threading and completed neck ring.

Indents on the Bottle

The next part in the modeling process was making all the indentations found on the shoulder and body of the bottle. For this, I used a series of 3D sketches and splines on the front view that closely resembled those on the bottle (Fig. 7). After getting the right curves on the bottle, the cut sweep feature was used to cut through the profile, giving the same indents as those found on the bottle. The resulting extrusions were fileted, then mirrored over the right plane and then copied by the use of the circular pattern feature around the axis of rotation which was drawn as a vertical line through the origin on the right plane (Fig. 8). For the V-shaped indents, the back view proved to be more effective in using these features as the symmetry can clearly be seen from this view.

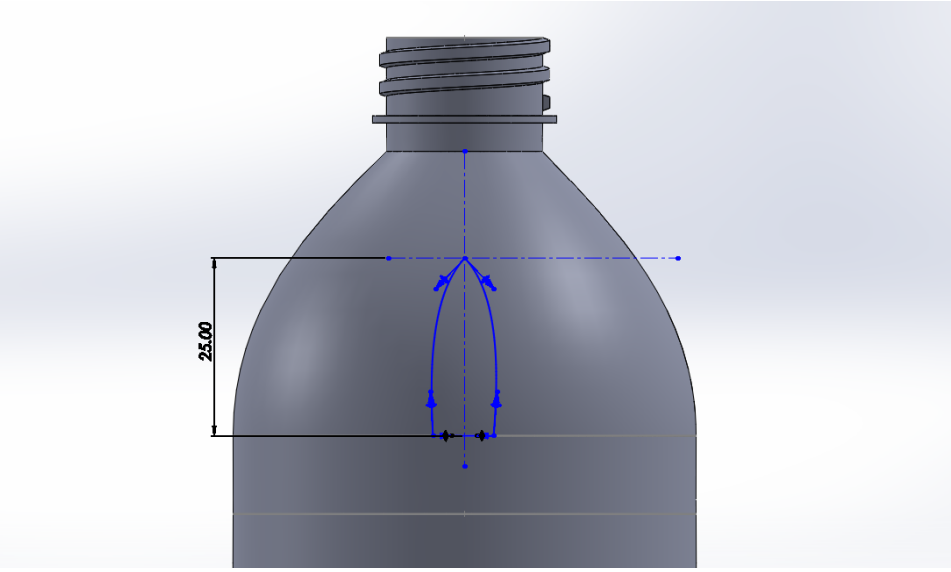

Fig. 7 – Sketch of spline for shoulder indents of the bottle.

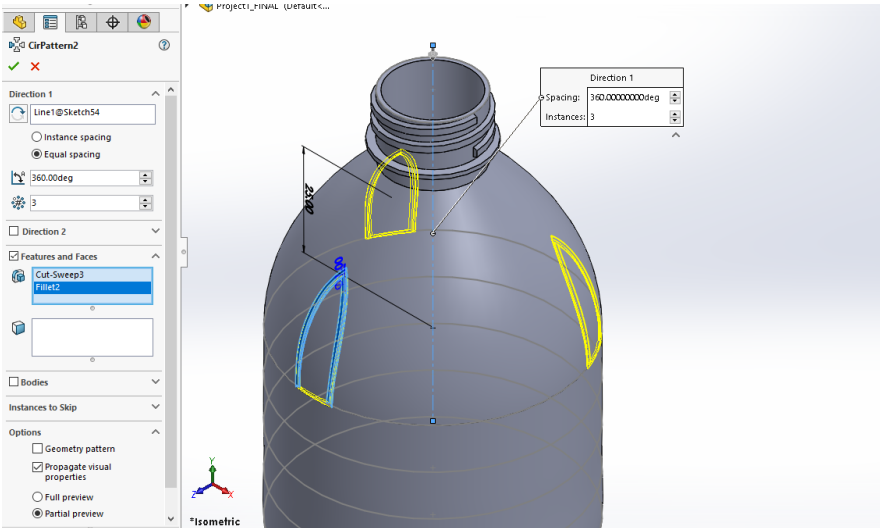

Fig. 8 – Circular pattern used for replicating indents around bottle.

In order to make the indentations along the circumference of the bottle, the cut revolve feature was used with a sketch of a circle on the right plane to cut through the body at each increment. The resulting model is shown in figure 9 below where the V-shaped indents are being copied.

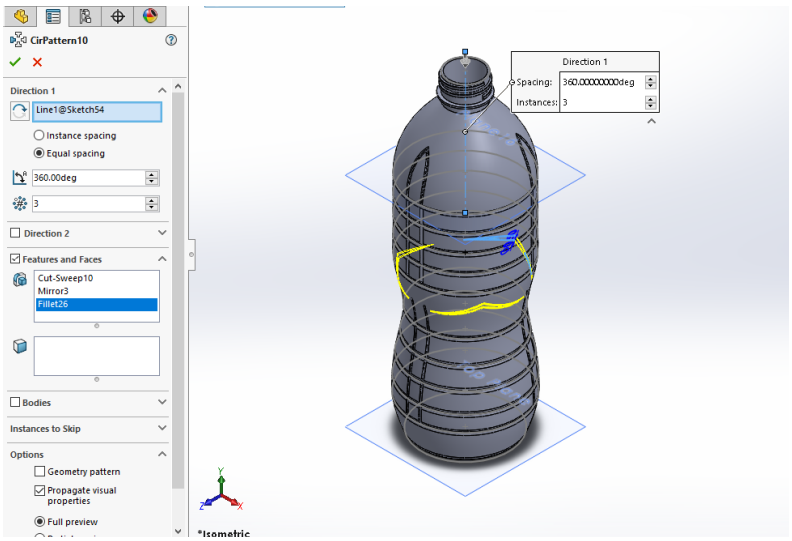

Fig. 9 – View of resulting model with the V-shape indents being copied.

Base and Bottom Indents

For the base details of the bottle, a sketch was made via the top plane. I first sketched a 55mm diameter circle for construction. I then sketched a smaller circle with a dimeter of 15mm along with one point of the star using a centerline. The point was then copied by the use of the circular pattern feature. Since there are 6 points and the circular pattern is being copied around 360 degrees, we don't need the extra dimensioning of the 60 degrees between each point on the sketch. In order to finalize the sketch at the base, the unnecessary lines are trimmed, and the sketch is then cut extruded 0.5mm towards the inside of the bottle. The resulting base is shown in figure 8.

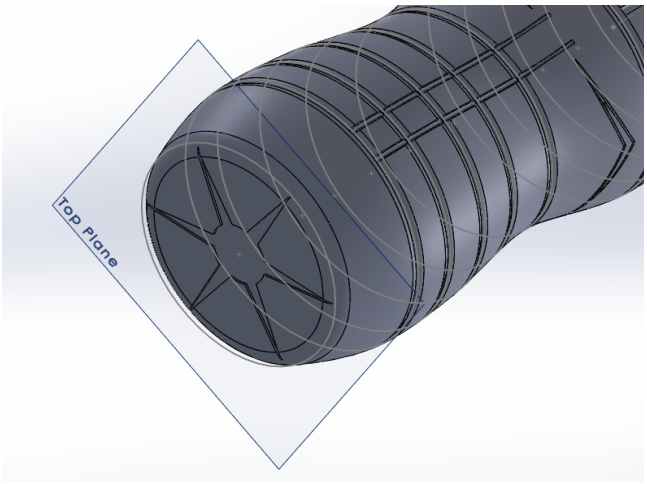

Fig. 10 – Completed base after sketching and cut extrusion.

The next step of the modeling process is the indentations found on the bottom edge of the bottle that curve around the base. Like the method used for the indents before, splines in 3D sketch were drawn and then used for the cut sweep feature which were then copied with a circular pattern. Two different splines were used at 15mm and 10mm distance from the bottom.

The final step was to choose an appropriate appearance for the model by selecting a material or color.

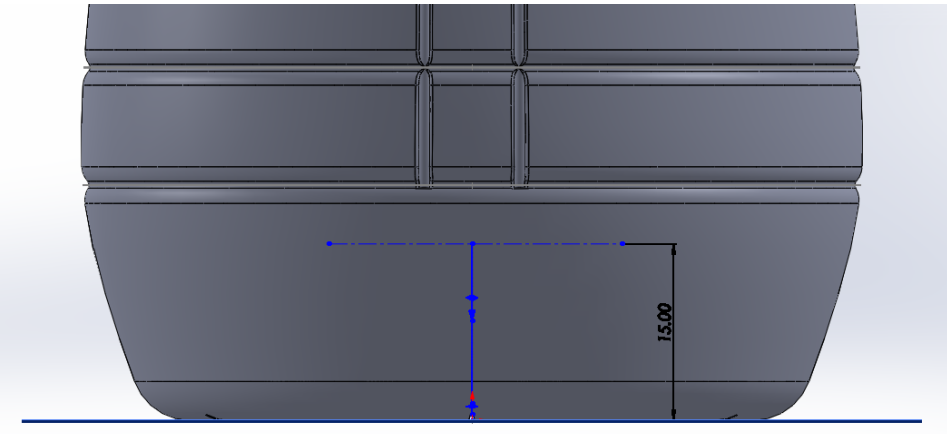

Fig. 11 – Sketch of the spline curving the base.

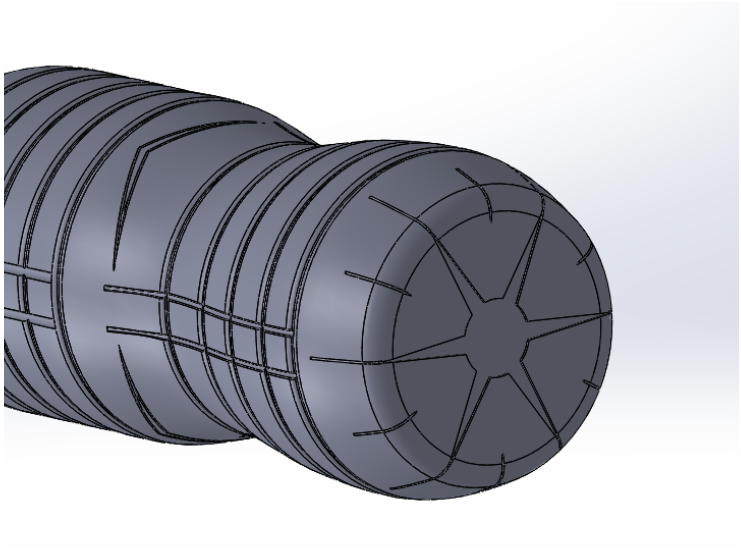

Fig. 12 – Completed bottom of bottle base.

III. Results

After the solid modeling was completed on SolidWorks, the following results were obtained:

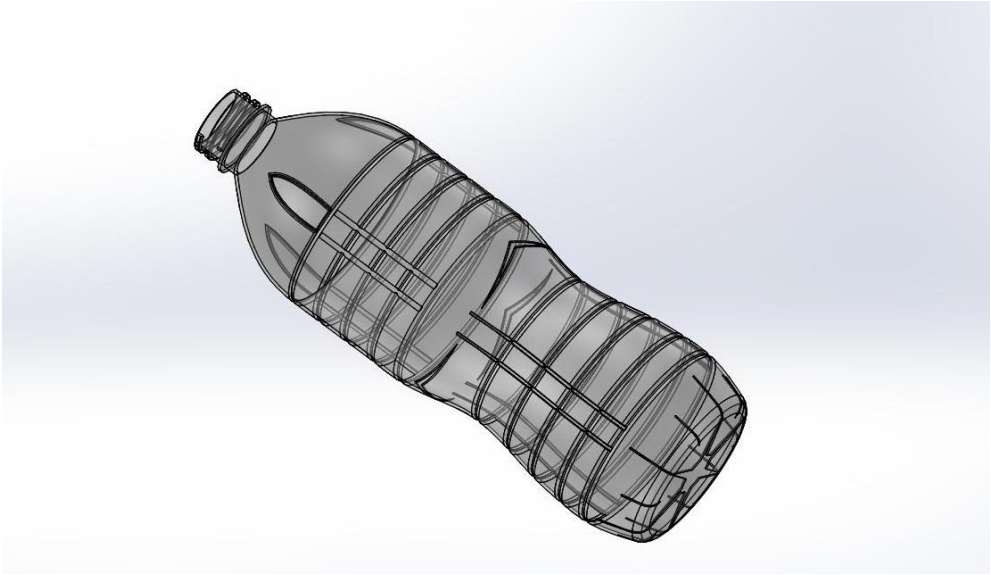

Fig. 13 – Final result of solid modeling the water bottle.

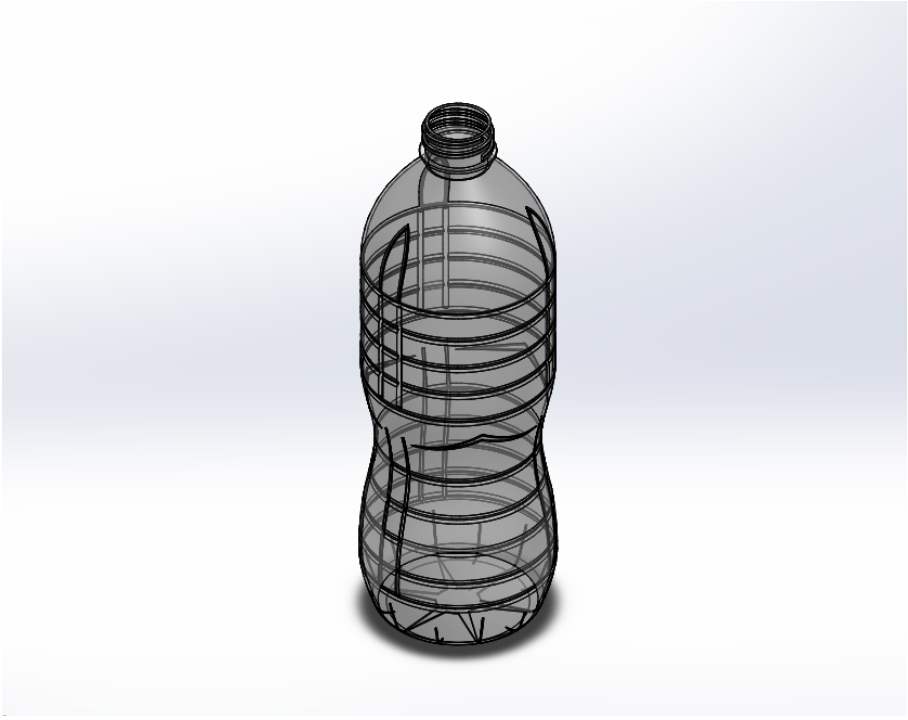

Fig. 14 – Isometric view of the finished model.

III. Discussion

This project taught me how to use new features such as the spline tool, loft, and sweep. I didn't have any experience prior with using these features so it took some practice in learning how to use them correctly. Since this object required the modeling of the curved indents, many splines needed to be constructed by using 3D sketch. The design of the object itself is very complex and difficult to follow through at first. I thought the curved indents on the bottle would be nearly impossible to replicate in SolidWorks, however the spline tools sweep cuts made it more than possible. It was a little difficult to sketch in 3D on the program, so a suggestion would be to improve this feature by it having more options to accurately sketch in 3D for whatever it is you are modeling.

The engineering drawing from the SolidWorks part file did not let me dimension everything I wanted to. One example would be the diameters of the circles used for the initial skeleton of the bottle. Since these sketches were used for lofting, SolidWorks could not define a line or curve from the final sketches of the model. One very difficult thing I was not able to do was to replicate the wavy indents around the circumference of the bottle found towards the bottom half of the body. I tried using the spline feature to sketch a wave over the body however the cut sweep feature would not be able to cut through the model as precisely as I wanted it to. One way in which I would have done this project differently would be to use alternative methods of capturing the curves. Wire could have been useful for the wavy curves around the bottle, and it would have provided a more accurate dimension for the indents.