ME 371- Section 1EF
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October 25th, 2021
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Project 1: Reverse Engineering and Solid Modelling of a Porcelain Teapot

Overview

The part that will be examined is a porcelain teapot. After observing the shapes of the tea pot and taking the dimensions for as many components as possible, one is able to break it down into multiple parts. This can be seen in the hand-drawn diagram of the porcelain teapot below. When modelling this product, the main SolidWorks features that will be used are: Loft and Sweep. The goal of this project is to utilize such SolidWorks features to reproduce a model of the porcelain teapot so that it closely resembles the original object, so that its measurements are 1:1 with the original teapot's measurements. Layer 1 will be created using a single Loft feature while Layers 2 & 3 will be lofted together in another Loft feature to create the teapot body. The entire teapot body will be shelled to a 3.2 mm thickness. The teapot spout will be created using a Surface Loft feature. And finally, the Handle will be created using a Swept boss/base feature.

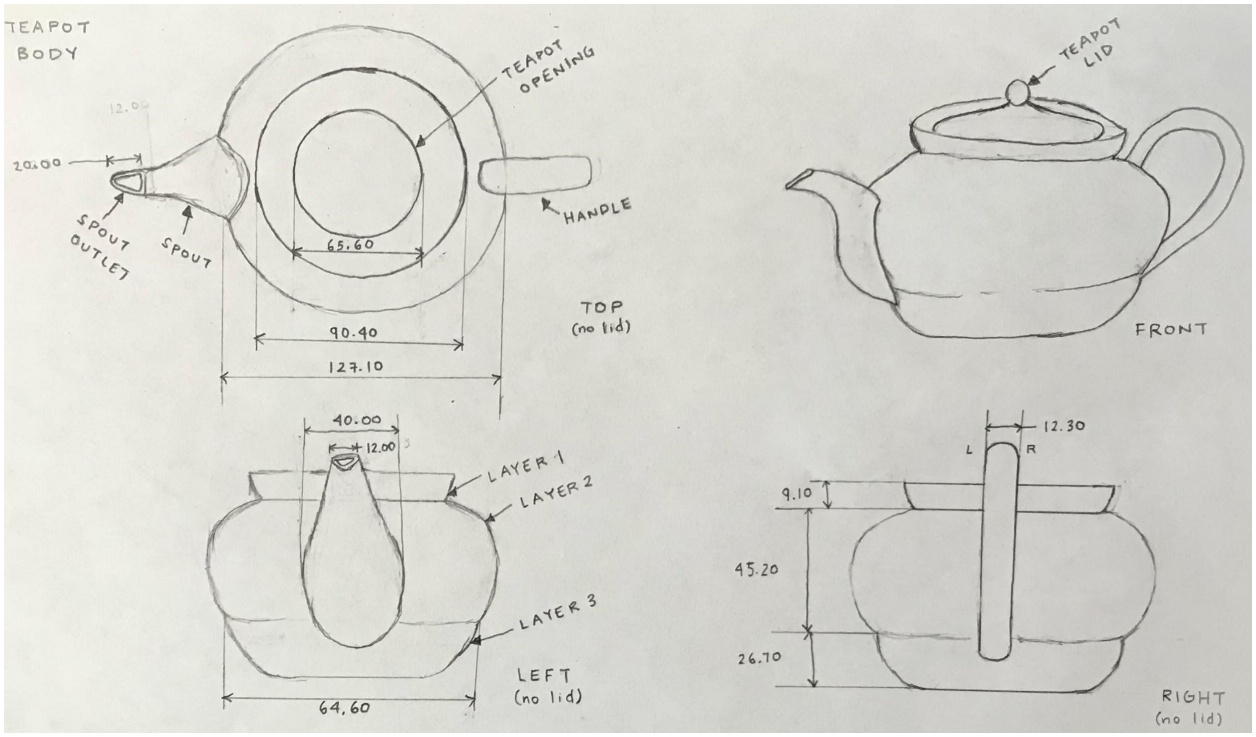


Figure 1. Hand-drafted porcelain teapot with 3 standard views (teapot lid omitted)

Procedure

In order to be able to properly model the teapot in SolidWorks, a caliper and soft measuring tape were used to obtain the dimensions of all of their components. Each of the measurements for the teapot components are placed into a table and labeled with “caliper” and “measuring tape” to indicate which measuring instrument was used to determine their dimensions. For easier reference, look at the sketches above to see what each dimension’s label means.

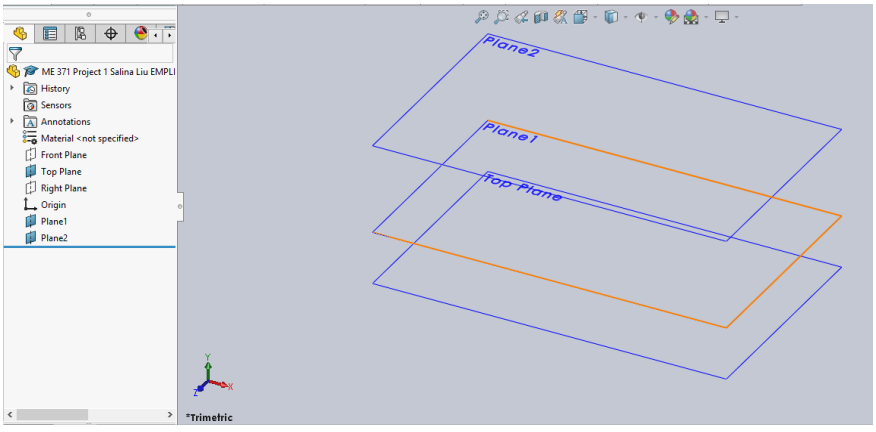
a) Measurements for Teapot:

Teapot section	Component	Dimensions	Measurement Method
Layer 1	Diameter of the top of Layer 1	99.5 mm	caliper
	Diameter of Layer 1’s Opening	65.6 mm	caliper
	Height of Layer 1	9.1 mm	caliper
	Circumference of base of Layer 1 (had trouble measuring because spout and handle is in the way)	284 mm	measuring tape
	Diameter of base of Layer 1	~90.4 mm	calculated from above circumference
Teapot section	**Component**	**Dimensions**	**Measurement Method**
Layer 2	Circumference of widest part of Layer 2	370 mm w/o the spout	measuring tape
	Diameter of the widest part of Layer 2	127.1 mm	caliper
	Height of Layer 2	45.2 mm	caliper
Layer 3	Diameter of widest part of Layer 3	107.6 mm	caliper
	Diameter of Bottom base of Layer 3	64.6 mm	caliper
	Height of Layer 3	26.7 mm	caliper
	Spout outlet diameter	12 mm	measuring tape

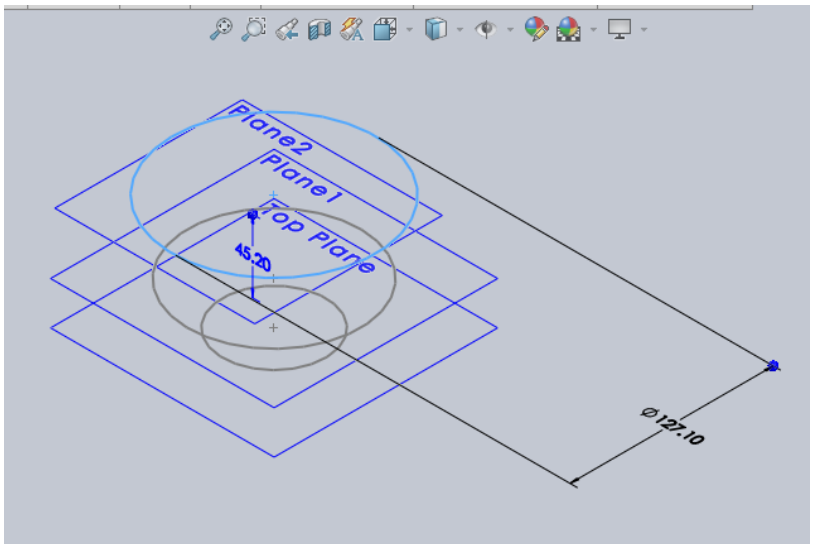
Spout	Diameter of Hole where liquid travels to spout	30 mm	measuring tape
	Widest part of spout (dimension not used)	40 mm	caliper
	Length of spout bottom (curve) (dimension not used)	81 mm	measuring tape
	Length of spout top (curve) (dimension not used)	40 mm	measuring tape
	Spout outlet length (dimension not used)	20 mm	measuring tape
Handle	Handle width (measured from L to R of handle)	12.3 mm	caliper
	Handle thickness (measured from inner part to outer) (dimension not used due to inconsistency)	8.1 mm	caliper
	Length of handle (outer curve) (dimension not used)	165 mm	measuring tape
Overall Teapot Body	Thickness (estimated thickness)	3.2 mm	caliper

b) Modeling:

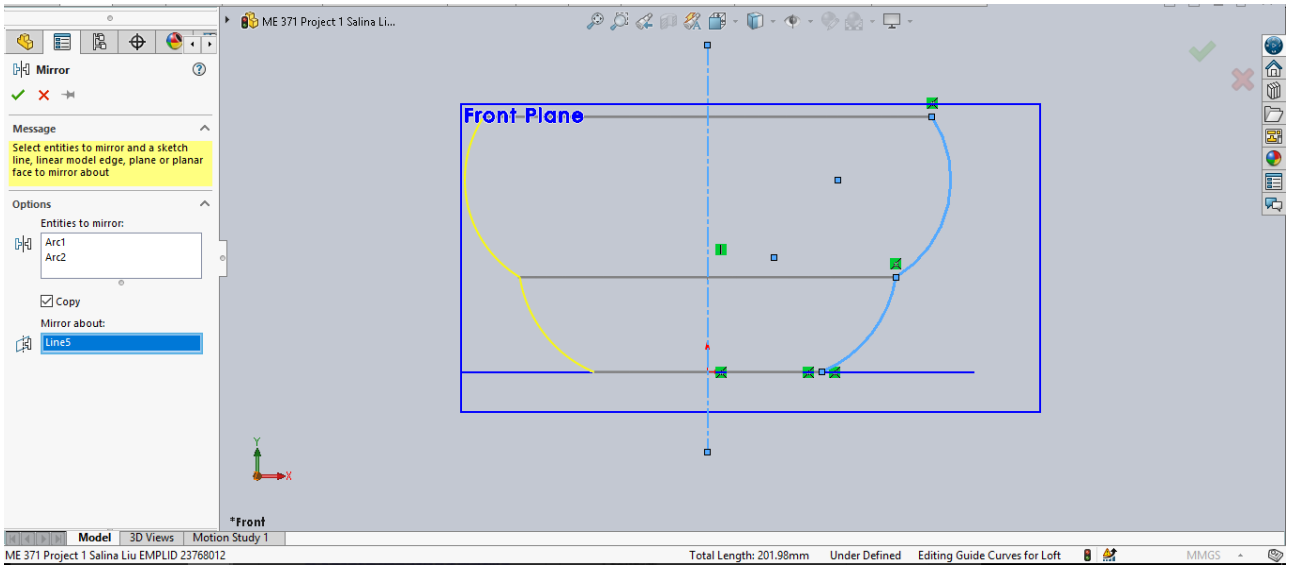
Once all possible measurements were taken, the next step was to create the SolidWorks model of the teapot. One can use the ***Loft feature*** to create the uniquely shaped body of the teapot. The first step for using the ***Loft feature*** is to create reference planes. In this case, there were two planes for the teapot's two "layers"of different circumferences. The spacing of the planes reflected the distances between each "layer": Plane 1 was created with reference to the Top Plane and is 26.70 mm above it. Plane 2 was created with reference to Plane 1 and is 45.20 mm above it.



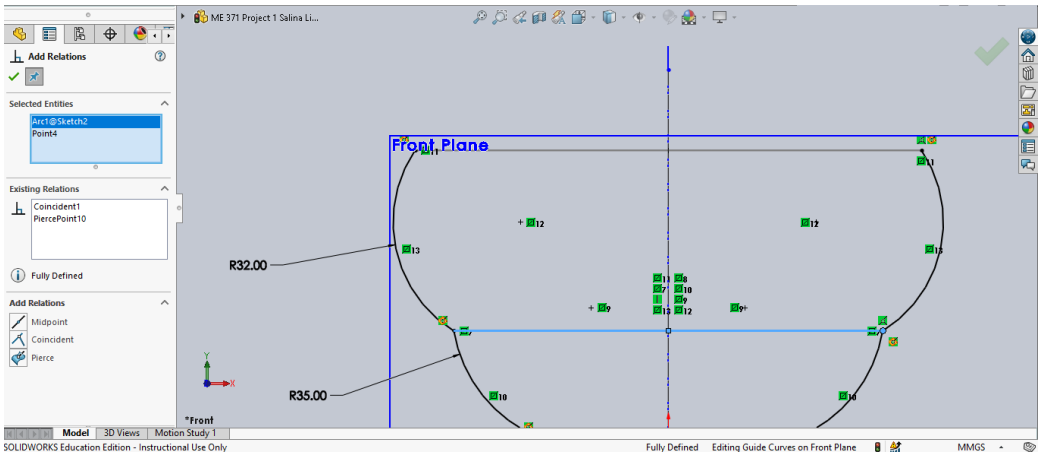
The next step to creating a ***Loft feature*** is to sketch circles on all of these planes. Here, one can see how each circle is dimensioned with the diameter that was measured for each "layer" of the teapot and how the diameters increase as the planes reach higher. The circles' diameters (Top Plane to Plane 2) are 64.60 mm, 107.60 mm, and 127.10 mm, respectively.



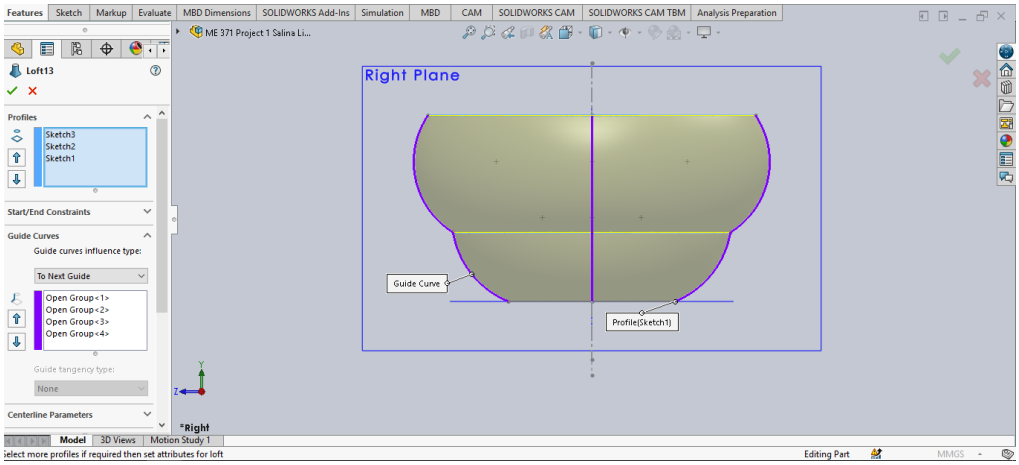
Once the sketches on the surfaces of the planes are created, one must also create Guide Curves for the ***Loft feature*** to be able to recognize that the shape of the teapot is rounded for each layer. The Guide Curves (two connected arcs) must be sketched on both the Front Plane and the Right Plane and they can be reflected across a centerline, using the mirror entities sketch feature, to reduce errors from trying to replicate the arcs on the left side of the sketch. The radii for the larger and smaller arcs are 32 mm and 35 mm respectively.

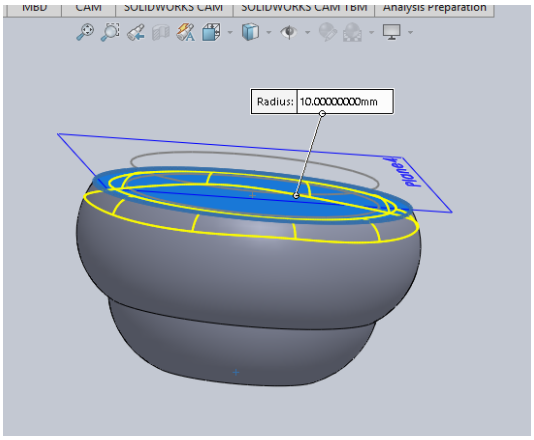


For these Guide Curves to serve their purpose, all lines need to be properly connected to the sketches of the three circles on the Top plane, Plane 1, and Plane 2. In order to ensure this, one can use the “Add Relations” feature and pierce the points of each arc with the three sketched circles.



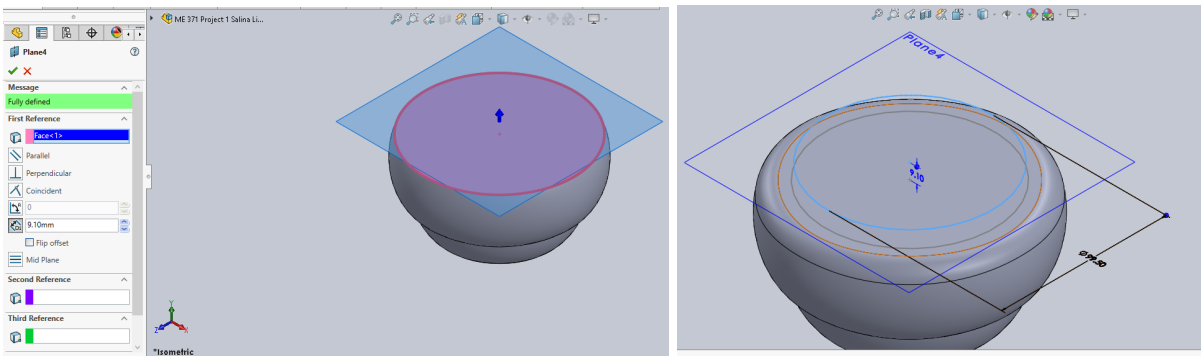
Once the sketches for the Loft profiles and Guide Curves have been completed, one can apply the ***Loft feature***, making sure to select the Guide Curves separately (rather than selecting the entire sketches).





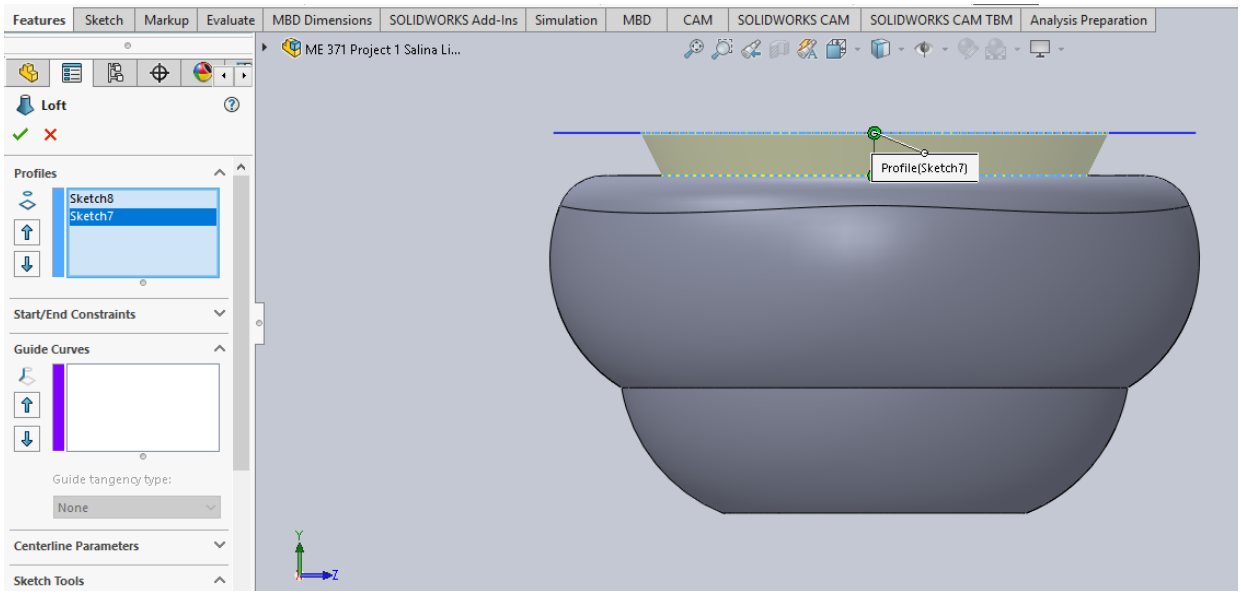
After completing the first ***Loft feature***, one should fillet the top surface of the teapot (10 mm) in order to make the surface more rounded and smooth, which is what the top of the real teapot looks like. This will also help prepare the surface before another ***Loft feature*** is created on top of it.

To create the top most region of the teapot, it will require another ***Loft feature***. To prepare for it, one must create another reference plane, using the face of the first lofted feature as reference. The distance between the plane and the face is 9.10 mm, which was the distance that was measured for the space between the teapot opening and its base (see image on left, below).

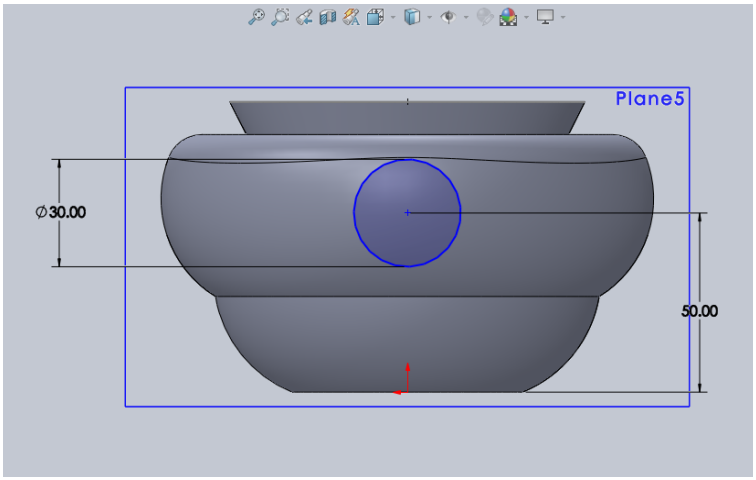


Once again, two circle sketches are made. The circle sketched on the face of the ***Loft feature*** has a diameter of 90.40 mm, and the circle sketched on the reference plane has a diameter of 99.50 mm (see image on right, above).

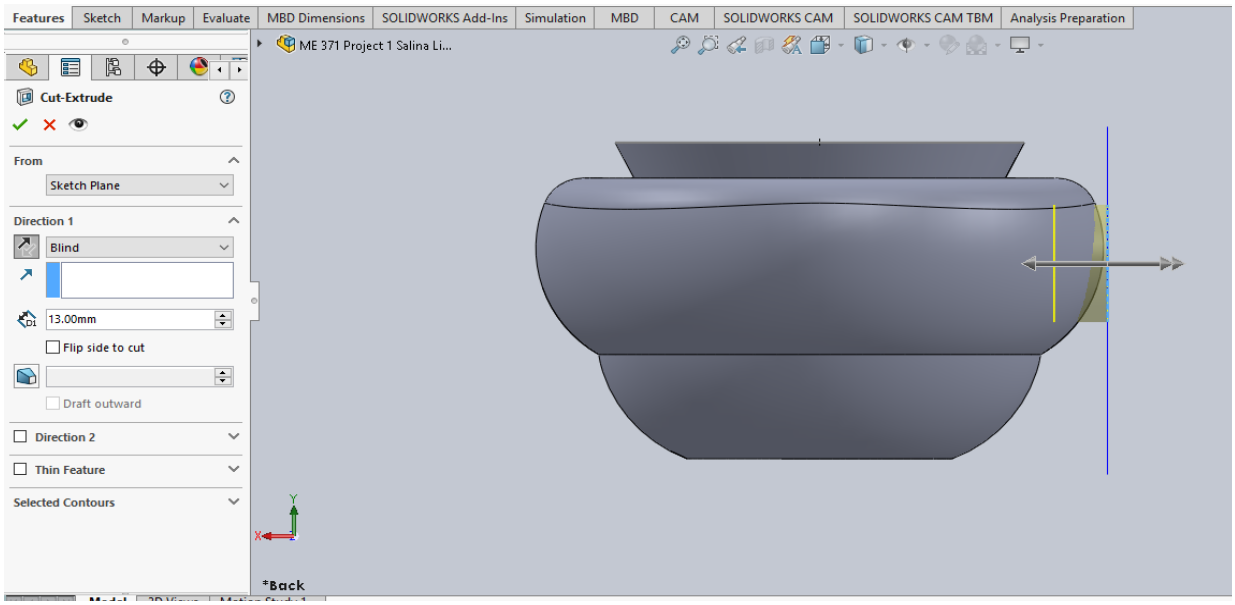
After sketching the circles on the two surfaces, one can create a second ***Loft feature*** by selecting the two circle sketches for the profiles. Selecting the sketches in the proper profile order is important because the ***Loft feature*** can sometimes loft the profiles to make an hourglass-like 3D shape rather than one that was intended.



Once the body of the porcelain teapot is complete, one can start looking at the detailed components of the object. One of them is the teapot spout inlet, which is a circular hole created in the body of the teapot (pictured below). To create this inlet, one can make a plane using the Right plane as reference and adjust the distance of the plane so that it is located very close to the surface of the porcelain teapot body. Then, one can create a sketch of a circle with a 30.00 mm diameter (which was measured using a tape measure), and approximate its distance away from the bottom of the teapot body.

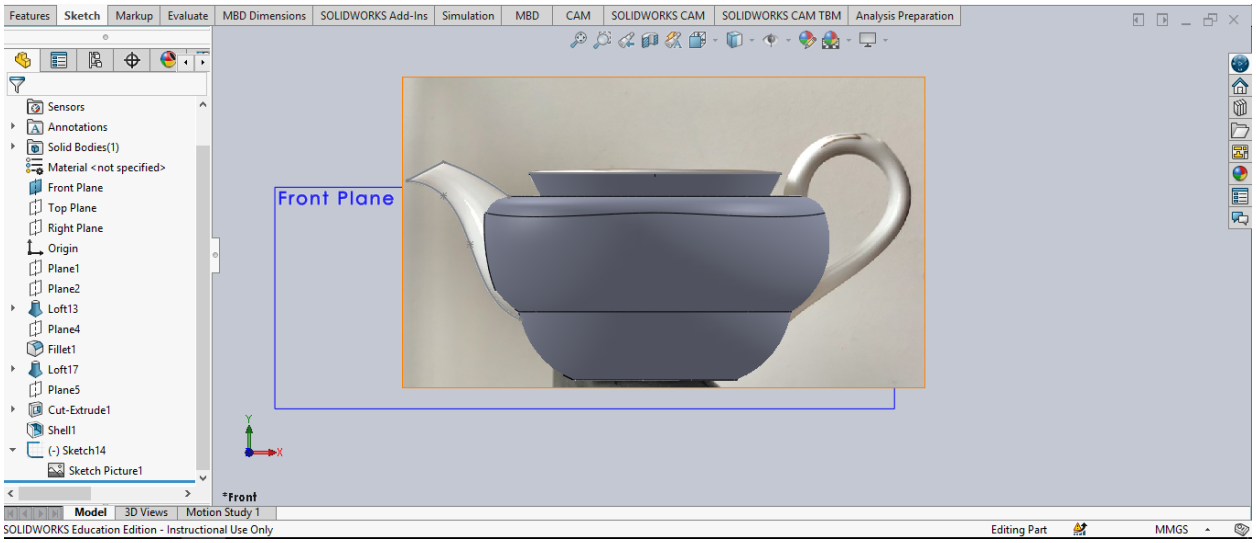


Once the sketch is created, one can use Extruded Cut to create a circular hole for the inlet. And to determine the depth of the cut, one can adjust it accordingly so that it cuts through just enough of the surface of the teapot body. For this object in particular, the cutting depth was 13.00 mm and it had to be re-oriented towards the object when deciding on cut direction.

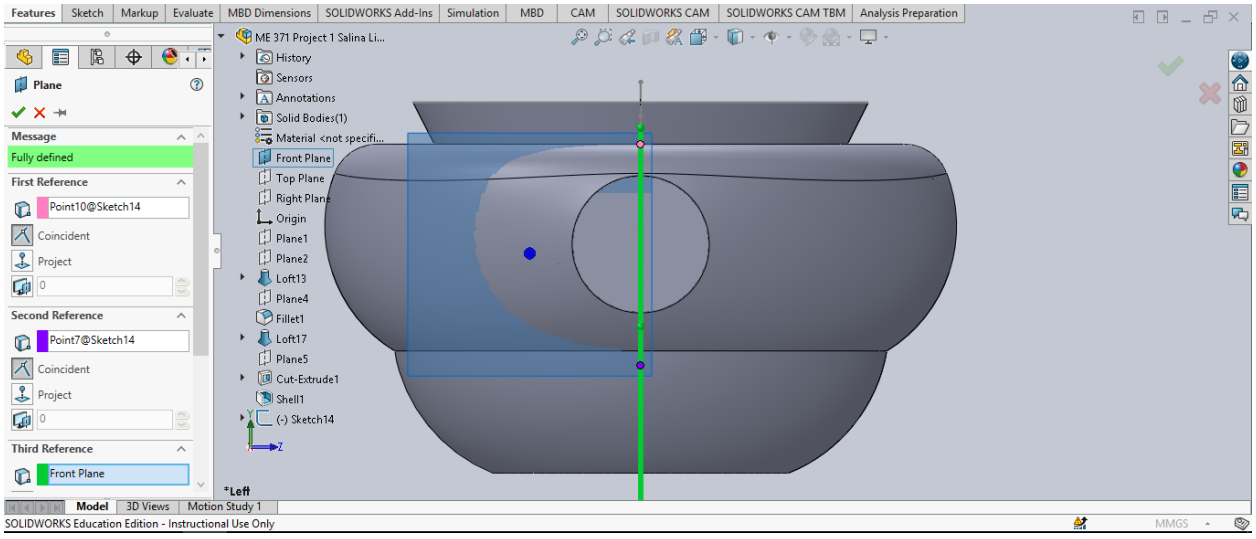


With the teapot spout inlet complete, the next component that needs to be made would be the actual teapot spout, which will be a ***Surface Loft feature***. This is due to the spout being a thin feature, and due to the fact that it is difficult to measure the thickness of this part of the teapot.

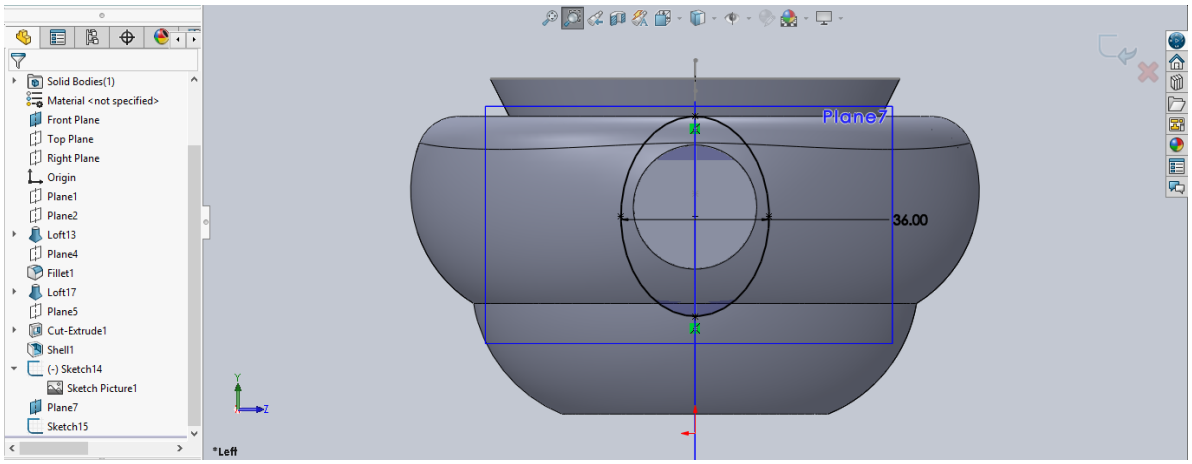
The first step to creating this feature would be to create the Guide Curves for the ***Surface Loft feature***. Because the teapot spout is made up of many complex curves, utilizing the Picture Sketch feature in SolidWorks would allow for tracing on the Front Plane and the best sketch tool to use would be the straight line (for spout outlet) and spline curves, which can be adjusted to fit along the shape of the teapot spout.



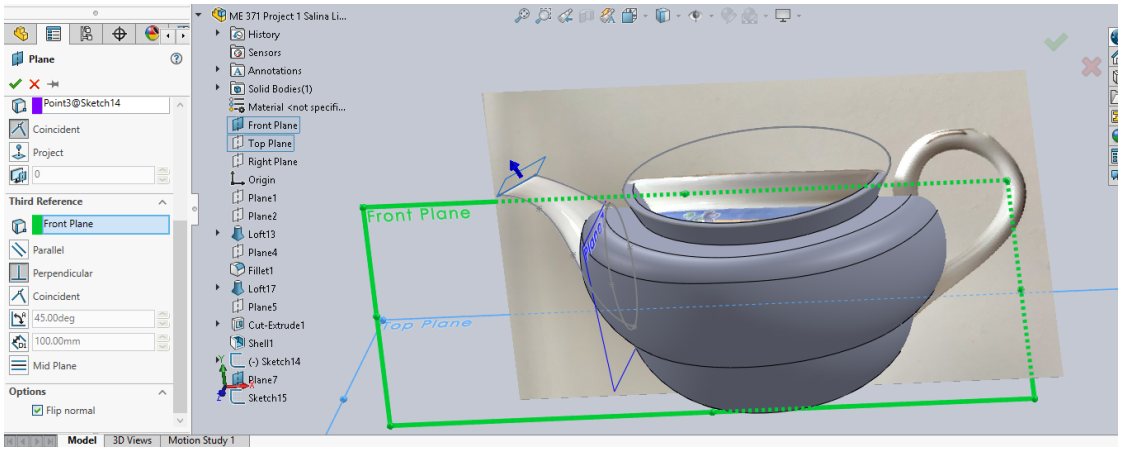
Then, to make the first profile for the ***Surface Loft feature***, one can use the endpoints of the Guide Curves (splines), and the Front plane as references to create a coincident plane.

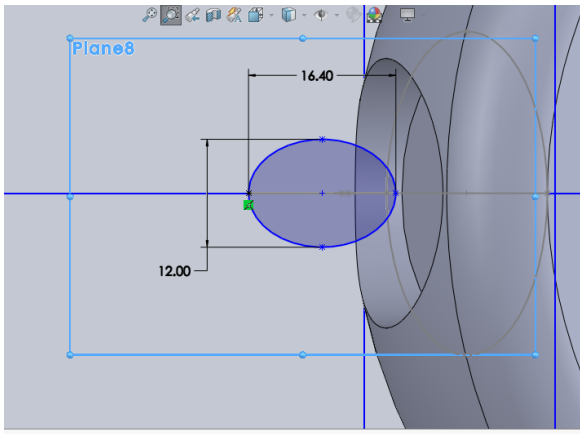


On the coincident plane (Plane 7), one can sketch an ellipse that has vertices coincident to the two spline curves (sketched on Front Plane and used Add Relations) and has a minor axis diameter of 36.00 mm (which is an approximated value from looking at the real teapot spout's placement).



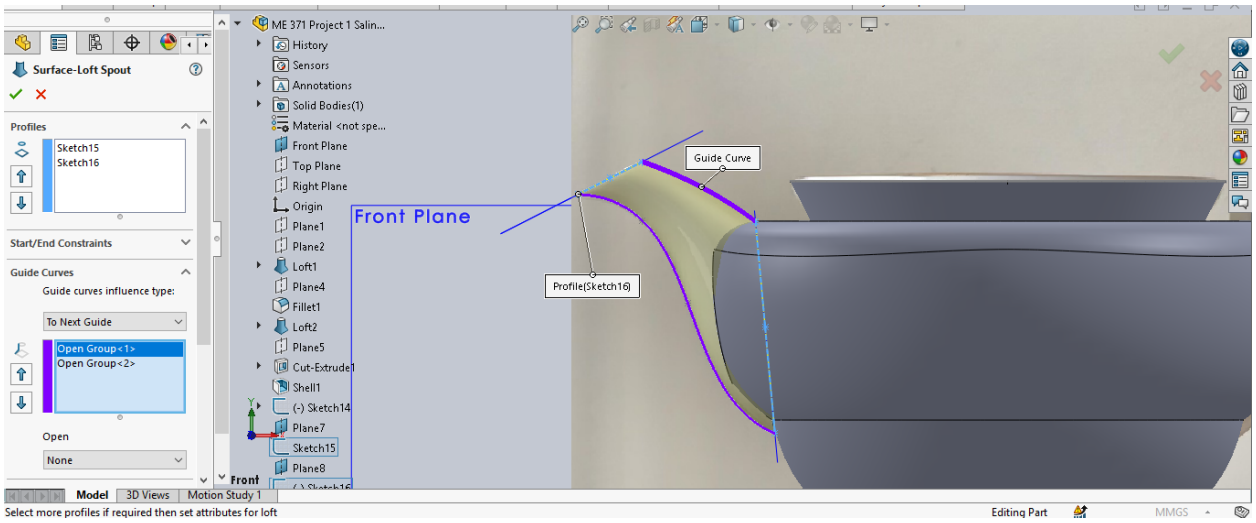
After creating the first profile for the ***Surface Loft feature***, one also needs to create a plane with reference to the two points on the line sketch for the spout outlet and the Front plane. This plane (Plane8) will be used to create the second profile for the ***Surface Loft feature***.



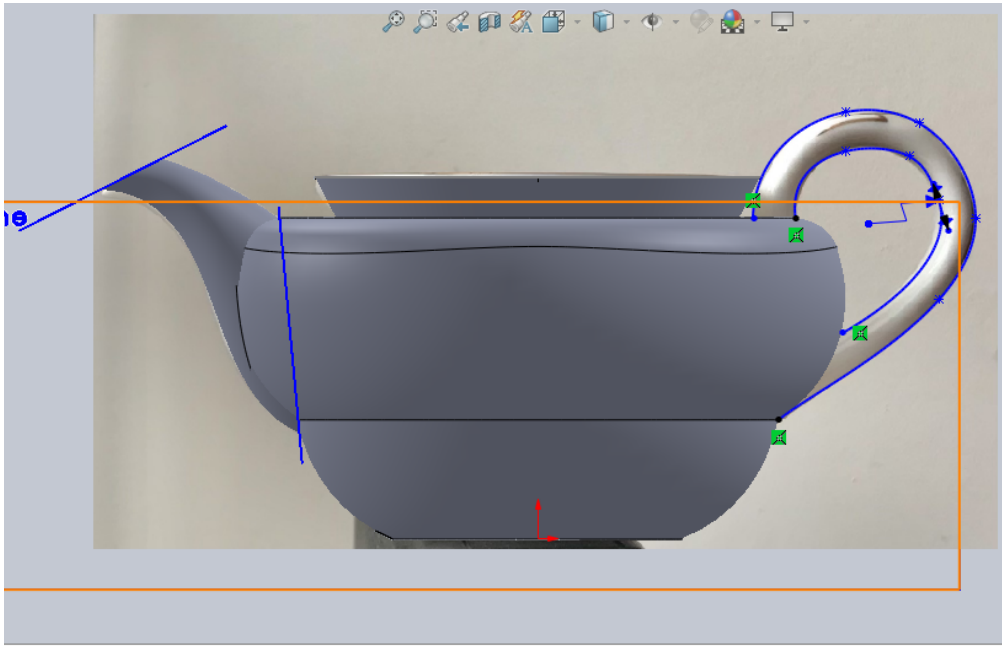


Once Plane 8 is created, one can sketch an ellipse that has vertices coincident with the two end points of the spout outlet line sketch (on Front Plane). And according to the measurements taken for this spout outlet, its minor axis will have a diameter of 12.00 mm.

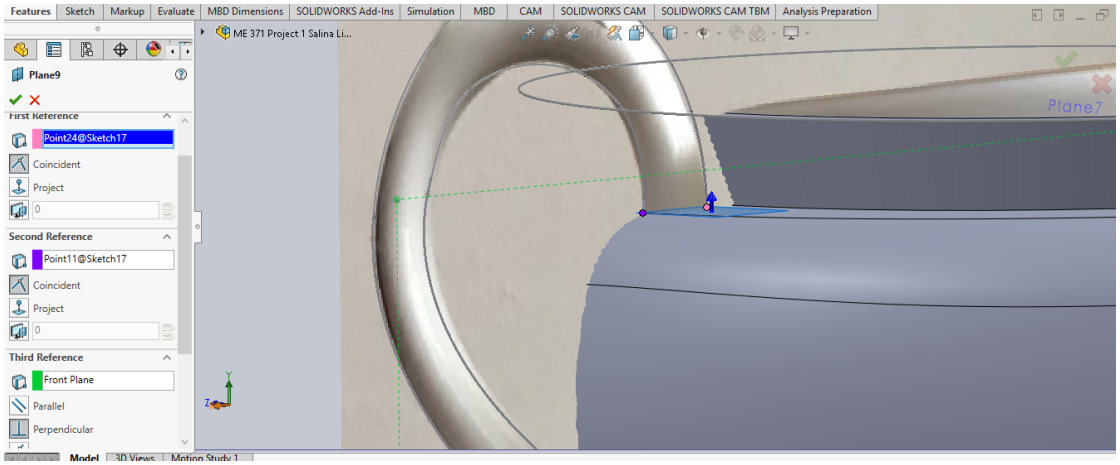
With the two ellipse profiles and the two guide curves, one can create the ***Surface Loft feature*** by selecting the profile sketches and by separately selecting the guide curves (so that SolidWorks recognizes the shape of the loft).

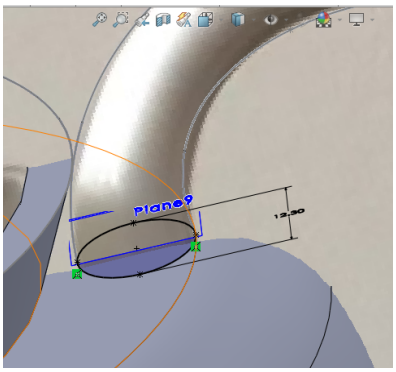


Since the teapot spout is a ***Surface Loft feature***, it does not require an extra "shelling" step and one can move on to creating the handle for the teapot. The technique that will be used will be the ***Swept boss/base feature***, which requires a profile as well as guide curves. The first step to create this swept feature is to create a sketch on the Front plane and utilize Picture Sketch to help trace out the handle's curves. The best sketch tool to use in this case would be the spline tool, as it allows for one to adjust the curves to fit the handle.



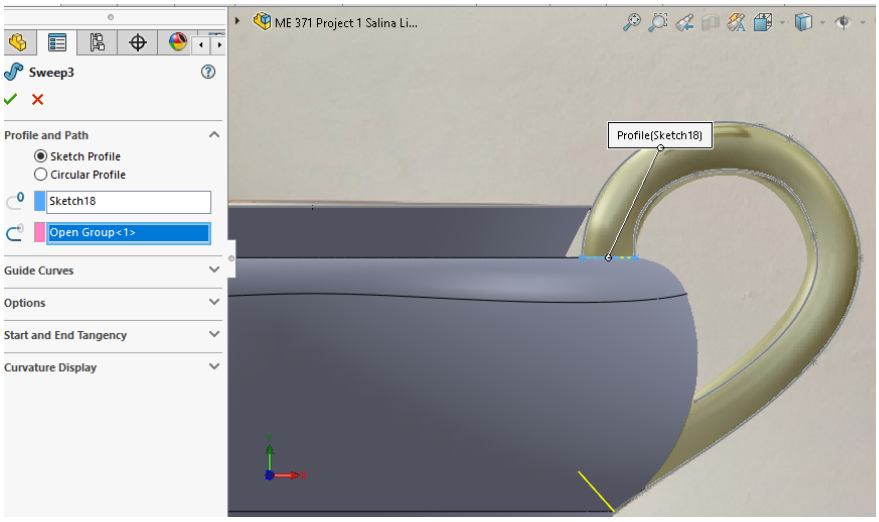
Once the Guide Curves for the handle are created, one can use the endpoints of these two splines and the Front plane as reference, to create a plane for the Sweep profile.



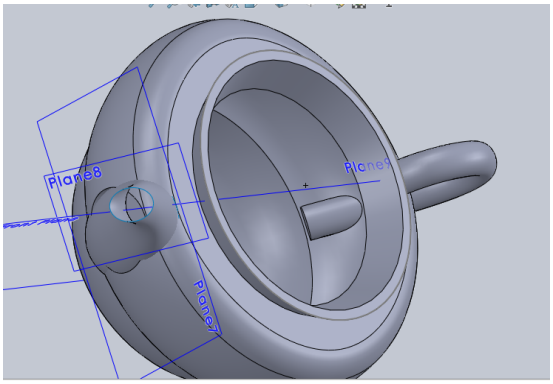


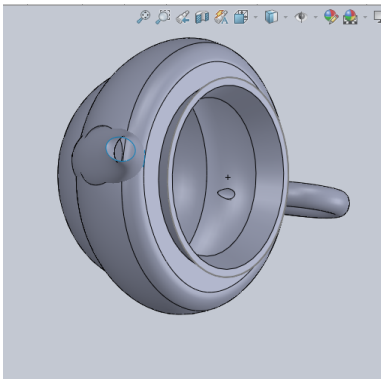
Then, one can sketch an ellipse using the same approach used before, where one adds relations between the co-vertices and the spline curves so that they are coincident with one another. Due to the handle being rounded, the only measurement that could be taken was the diameter of the major axis (width of the handle), which was 12.30 mm; and this was used to dimension the size of the handle sweep profile.

Once this profile is complete, one can create the ***Swept boss/base feature*** using this profile and the guide curves that were created. And it turned out that only one guide curve was needed to create this feature.

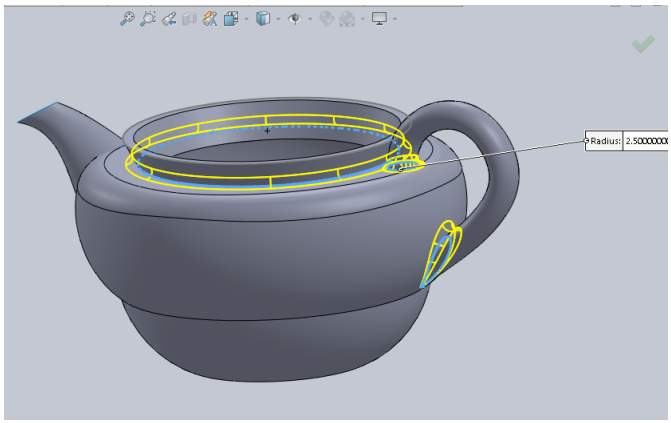


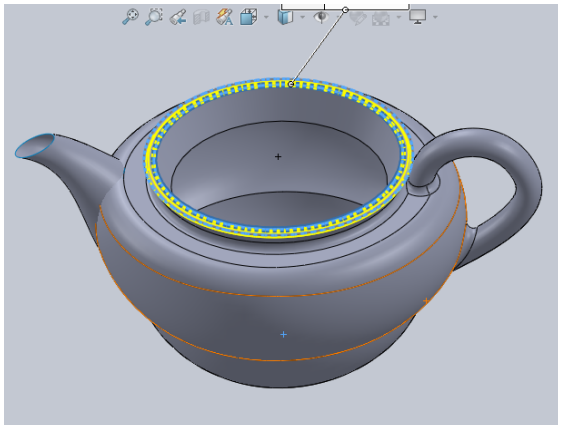
Since the ***Swept boss/base feature*** created a protrusion into the surface of the teapot body (see left), one can use a ***Shell feature*** to eliminate anything that goes beyond the surface (see right).

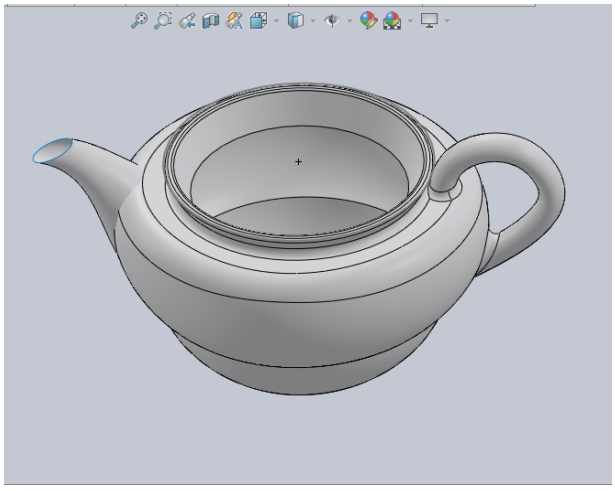




The last two features that will be used for the teapot body would be the ***fillets***. One can create three 2.50 mm ***edge fillets*** on the ends of the handle and the base of the opening to further smooth out those regions (pictured on the left, below). And one can create 1.00 mm ***edge fillets*** on the inner and outer rims of the opening (pictured on the right, below).



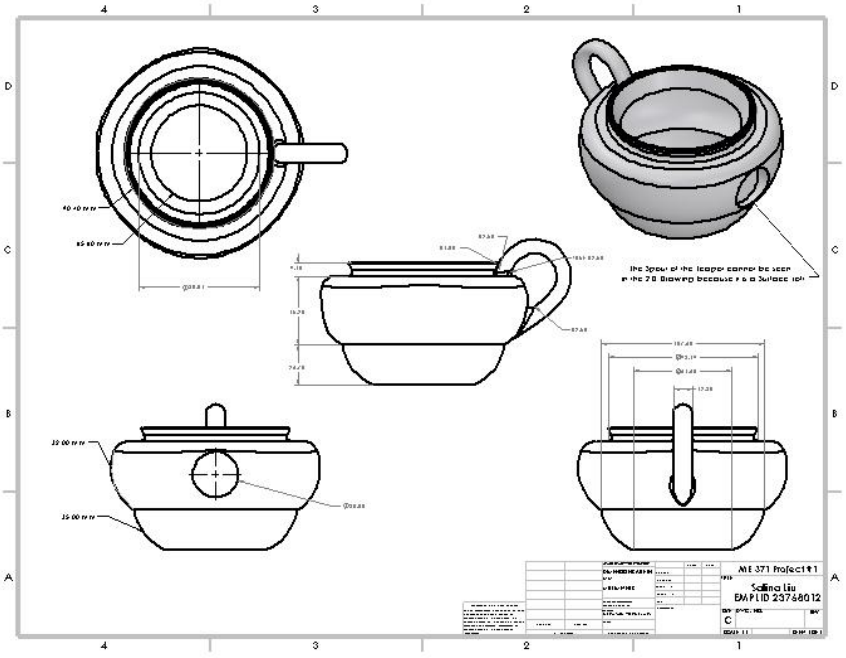




The final step for modelling the porcelain teapot body would be to change the material to ceramic porcelain, which would make it change to a white/grey color, and provide a better visual of what the teapot looks like in real life.

Results

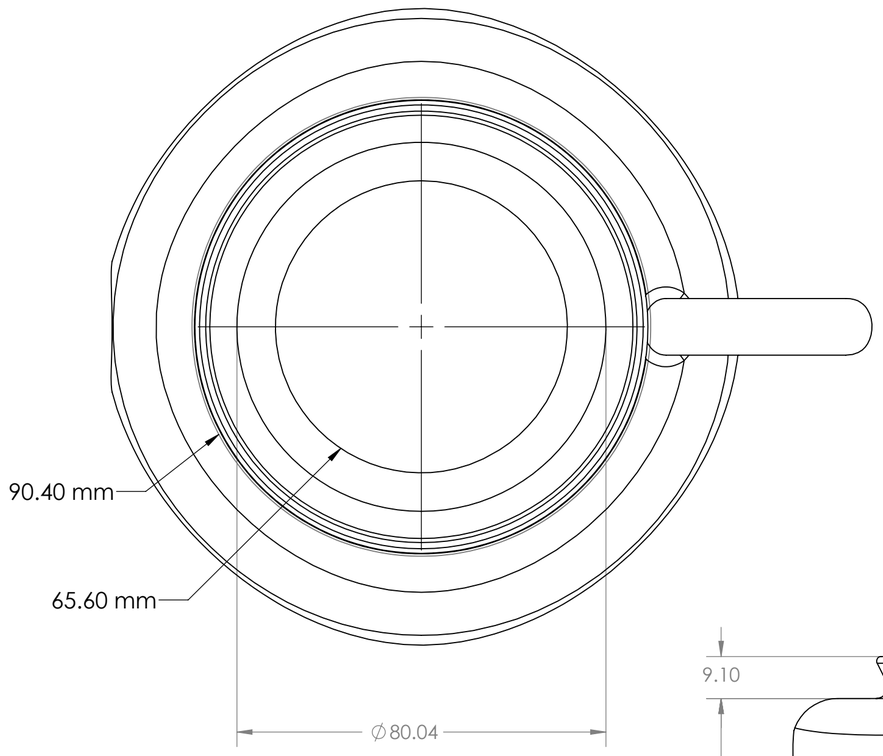
This is the completed solid model and the 2D engineering drawing detailing the dimensions of the porcelain teapot (For a clearer look, see last page of this PDF document).

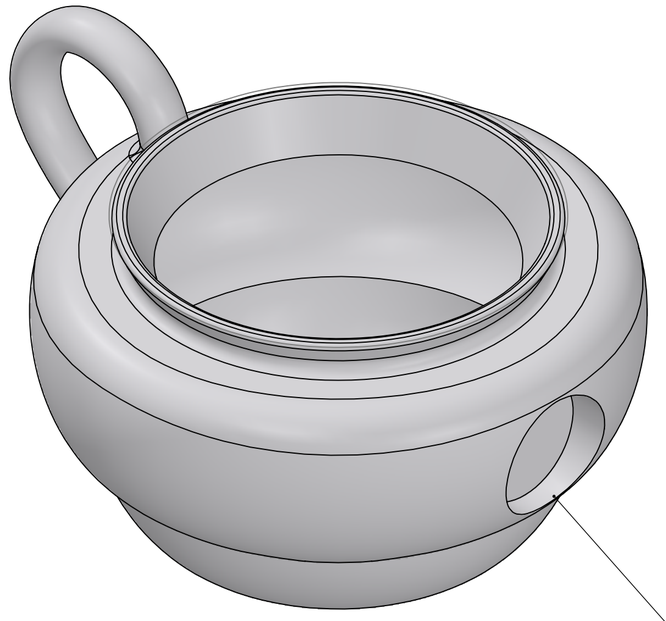


[SolidWorks does not show Surface Loft features, please refer to my hand-drawing on pg.1 for the details on the teapot spout]

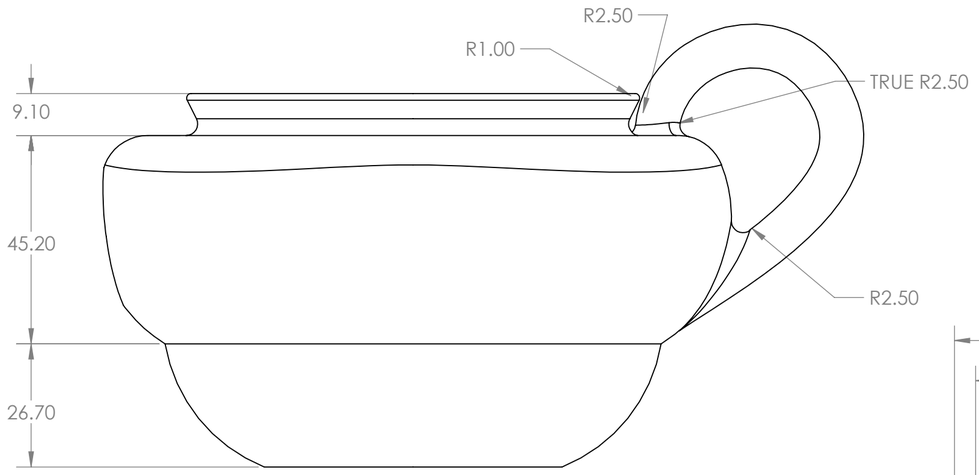
Discussion

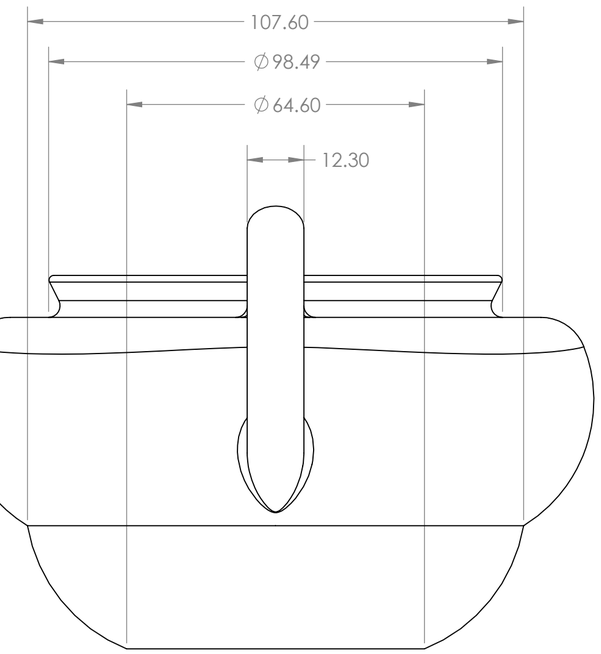
Overall, the modelling of the porcelain teapot had been successful and I was able to express every part of the teapot in the model. However, some ways in which I may have done things differently would be that I could have used a Surface Loft to create the teapot body so that I would not have to use the Shell feature to create the thickness of the teapot. This Surface Loft would have also prevented an issue I had with the lack of shell-ing for the teapot's spout inlet. I also think that it would have been better if I started to use the Picture Sketch earlier into my modelling process because the handle of the teapot turned out to be a little too close to Layer 1 of my teapot body. One thing that would have solved this problem with the handle would be that SolidWorks should allow me to use two guide curves I made for the Swept boss/base rather than only letting me use one of the splines I made. Through this project I was able to better learn how to approach an object and determine how it can be made via the Loft and Swept boss/base features in SolidWorks.

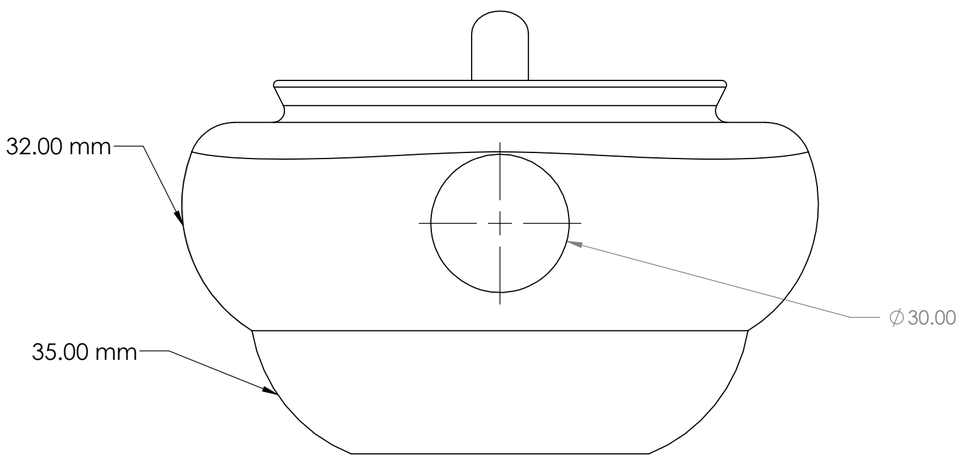




The Spout of the Teapot cannot be seen
in the 2D Drawing because it is a Surface Loft







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ME 371 Project #1

TITLE:

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EMPLID 23768012

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