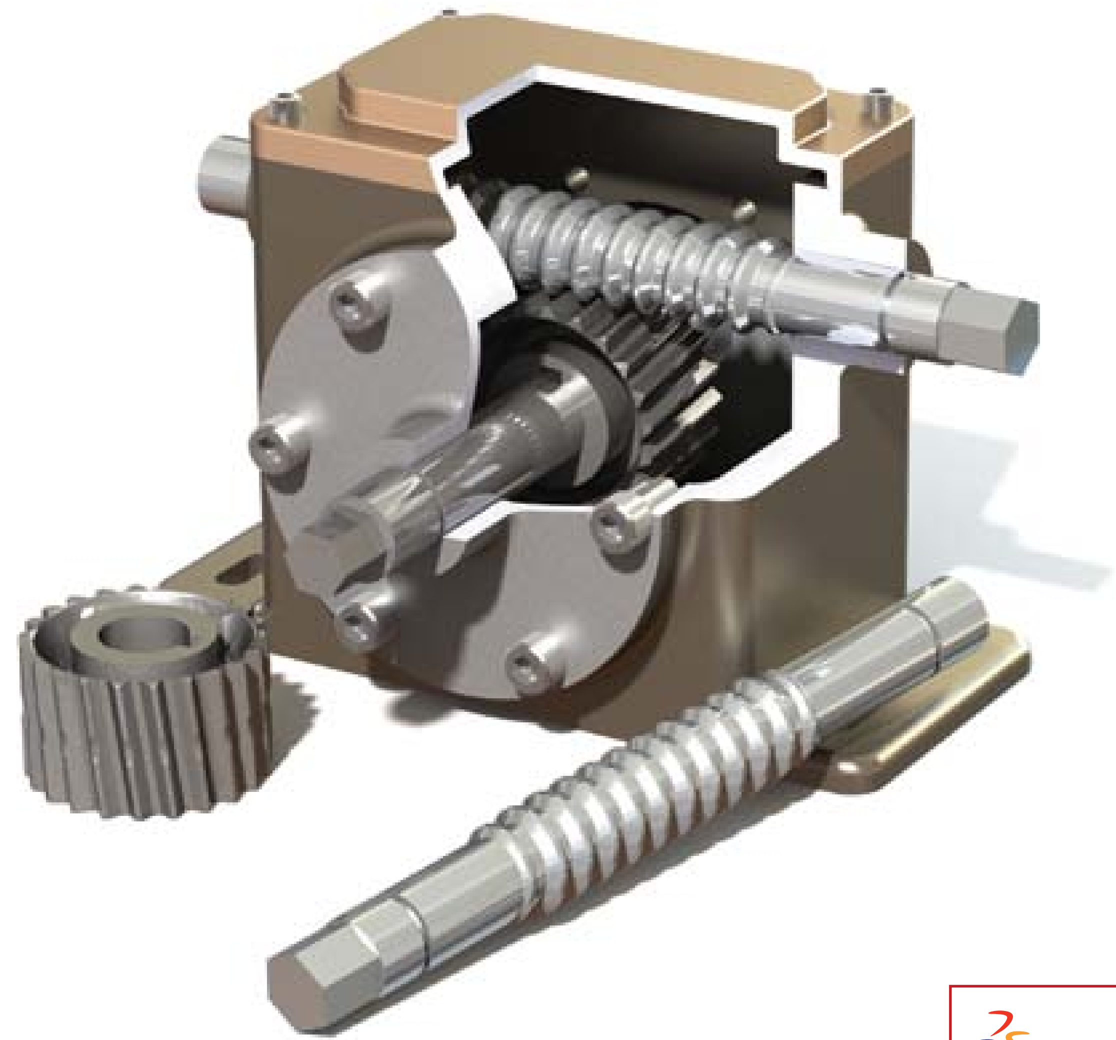
Beginner's Guide to SolidWorks® 2012 – Level I

Alejandro Reyes, MSME, CSWP, CSWI



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Special Features: Sweep and Loft

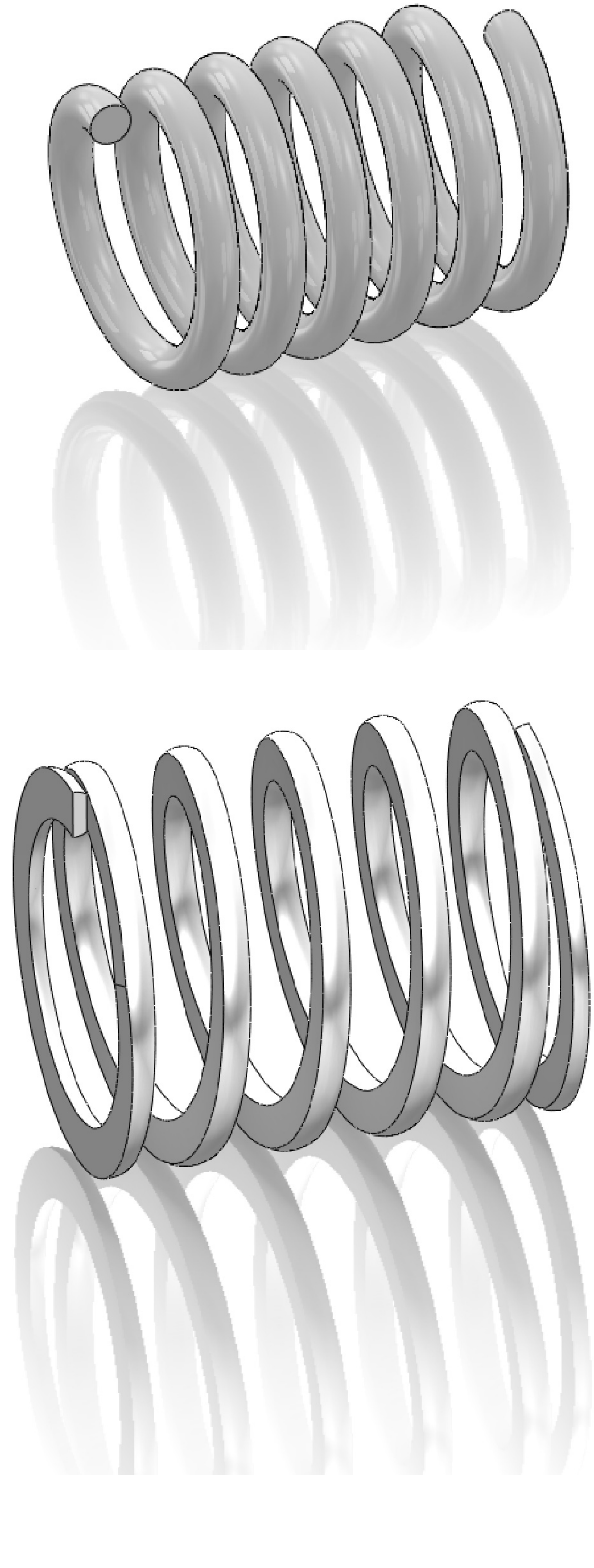
There are times when we need to design components that cannot be easily defined by prismatic shapes. For those features that have ‘curvy’ shapes we can use **Sweep** and **Loft** features, which let us create almost any shape we can think of. These are the features that allow us to design consumer products, which, more often than not, have to be attractive and appealing, making extensive use of curvature and organic shapes. The components can be easily manufactured using a plastic injection process. These products include things like your remote control, a computer mouse, coffee maker, perfume bottles, telephones, etc., and many times the success or failure of these products can be directly attributed to their appearance. They have to look nice, ‘feel’ right, and of course perform the task that they was meant for. Sweeps and Lofts are also prevalent in the automotive and aerospace industry where cosmetics, aerodynamics and ergonomics are a very important part of the design.

Sweeps and Lofts have many different options that allow us to create from simple to extremely complex shapes. In light of the vast number of variations and possibilities for these features, we’ll keep these examples as simple as possible without sacrificing functionality, to give the reader a good idea as to what can be achieved.

Sweeps and Lofts are usually referred to as advanced features, since they usually require more work to complete, and a better understanding of the basic concepts of solid modeling. Having said that, these exercises will assume that any command that we have done more than a couple of times up to this point, like creating a sketch are already understood and we’ll simply direct the reader to create it providing the necessary details. This way we’ll be able to focus more oin the specifics of the new features. These are some examples of consumer product designs made using advanced modeling techniques.

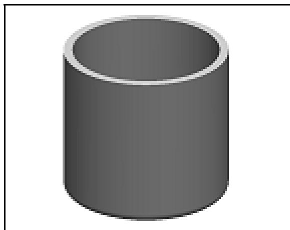
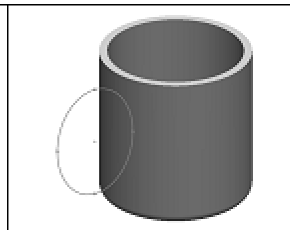
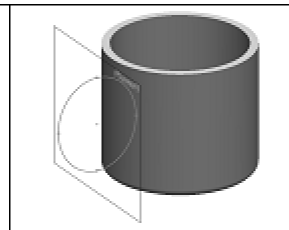
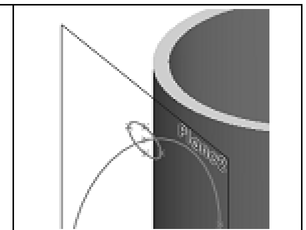
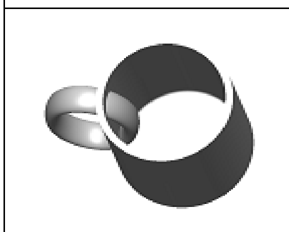
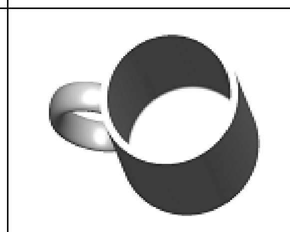
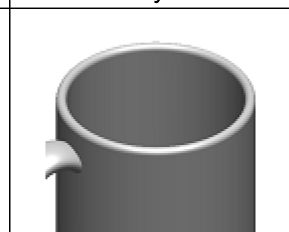
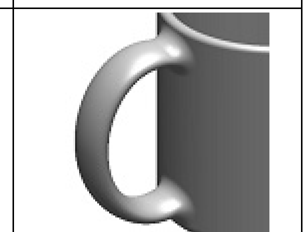
Notes:

Sweep: Cup and Springs

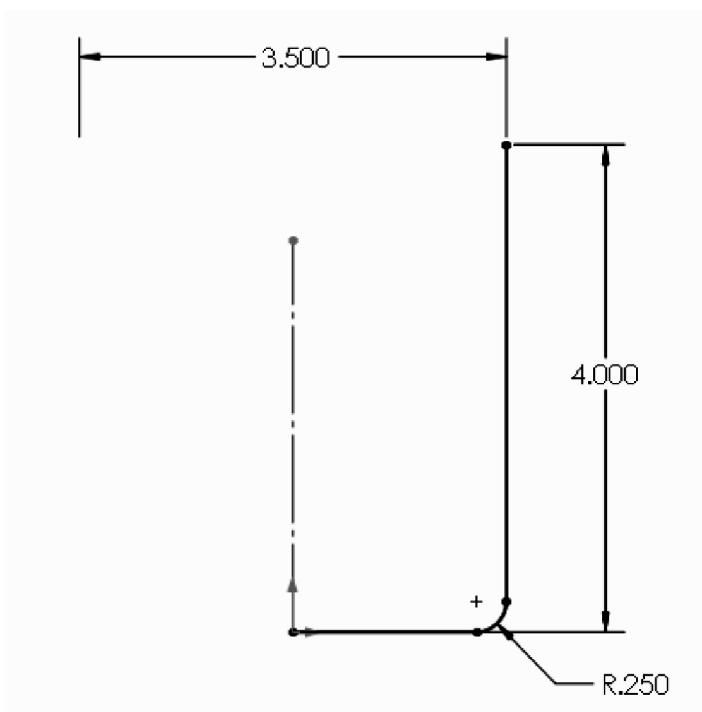


Notes:

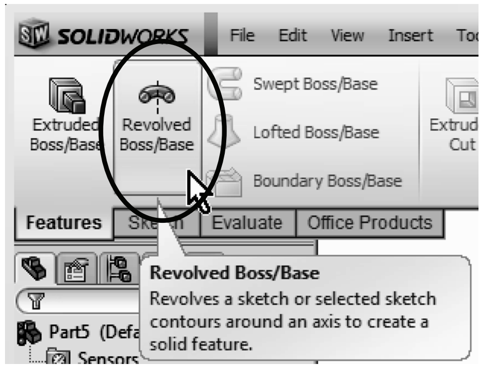
For this exercise we are going to make a simple cup. In this exercise we will learn a new option when creating features called "**Thin Feature**", the Sweep feature, a new Fillet option to create a Full Round fillet, and a review of auxiliary Planes. The sequence of features to complete the cup is:

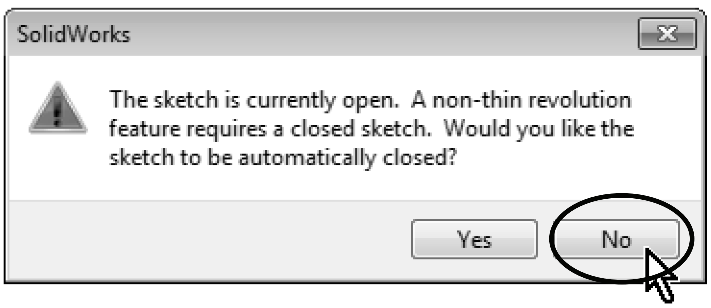
			
Revolve Thin	Create Path Sketch	Auxiliary Plane	Create Profile Path
			
Make Sweep	Cut inside of Cup	Full Round Fillet	Handle Fillet

149. – For the first feature we will create a "**Revolved Feature**" using the option "Thin Feature". This option makes a feature with a specified thickness based on the sketch that was drawn. Select the Front plane and create the following sketch. Notice the sketch is an open profile with two lines, an arc and a centerline. (Remember to make the diameter dimension about the centerline.)



150. - After selecting the "**Revolve Boss/Base**" command we get a warning telling us about the sketch being open. Since we want a thin revolved feature, select "No".

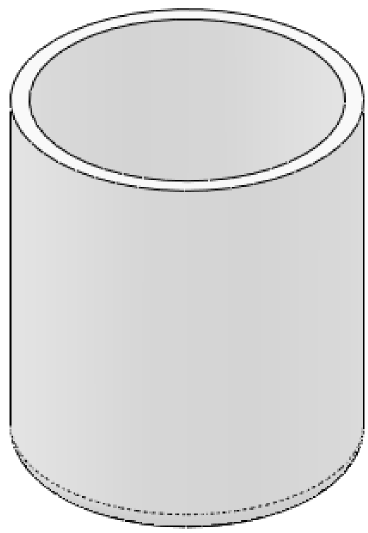




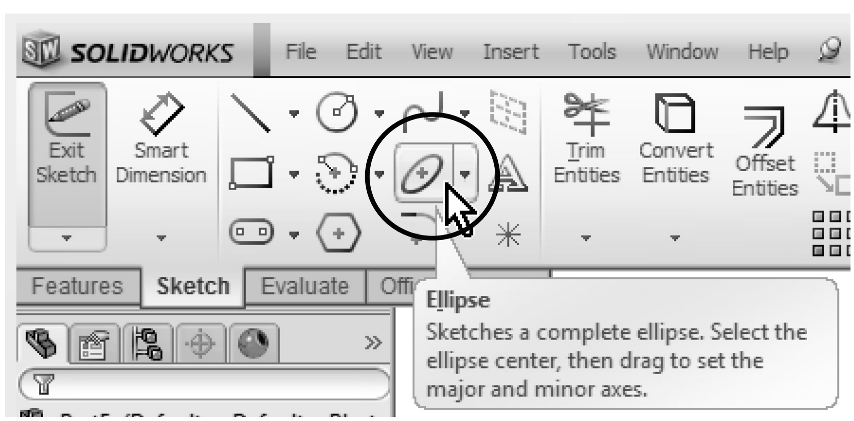
In the "Revolve" options, "**Thin Feature**" is automatically activated. For the cup we want the dimensions we added to be external, so we select the "**Reverse Direction**" option. Notice the preview shows the change adding material *inside*.

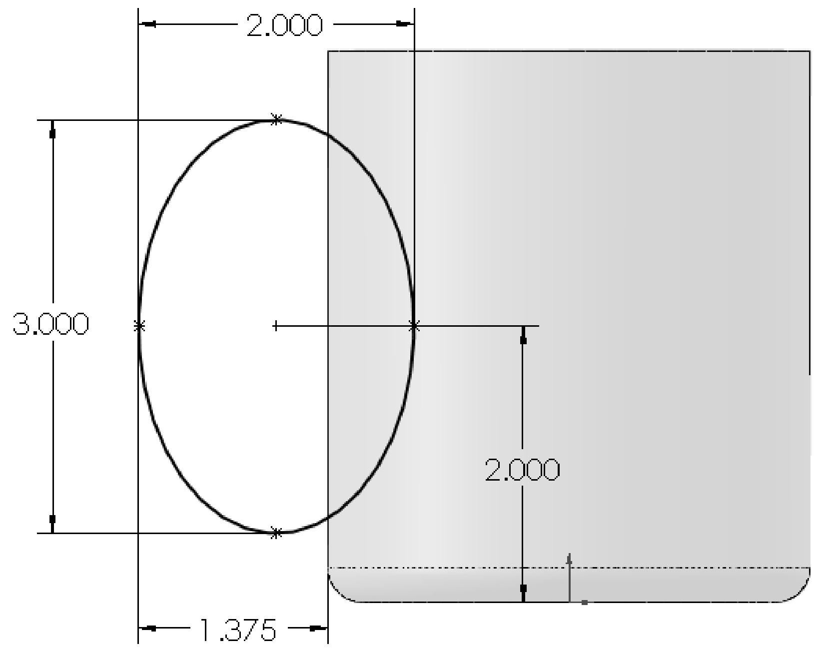
In the value box we typed 3/16; we can add a fraction and SolidWorks changes it to the corresponding decimal value when we click OK. We can also type simple mathematic expressions including addition, subtraction, multiplication and division in any value box where we can type a value.

Our thin feature should now look like this:

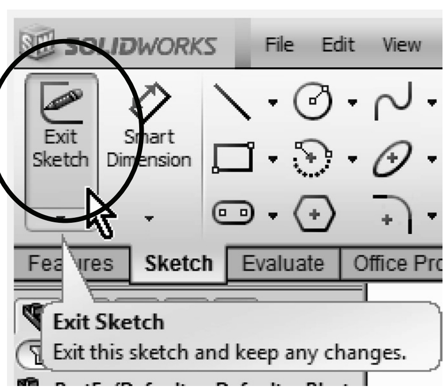


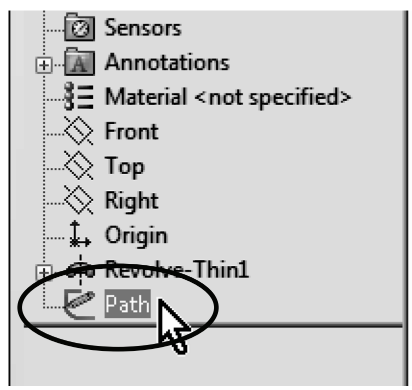
151. - Select the Front Plane and create the following sketch using an ellipse. Be sure to add a "Vertical" relation between the top and bottom points of the ellipse to fully define it. Select the "**Ellipse**" command from the Sketch tab in the CommandManager or from the menu "**Tools, Sketch Entities, Ellipse**". Add the corresponding dimensions to and from the ellipse points at the major and minor axes.



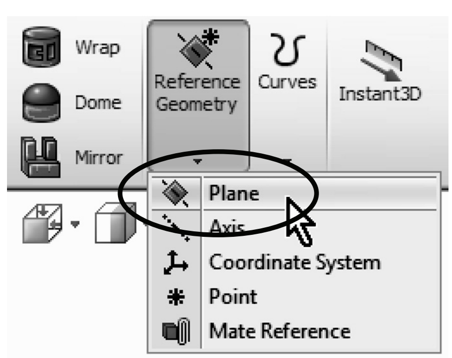


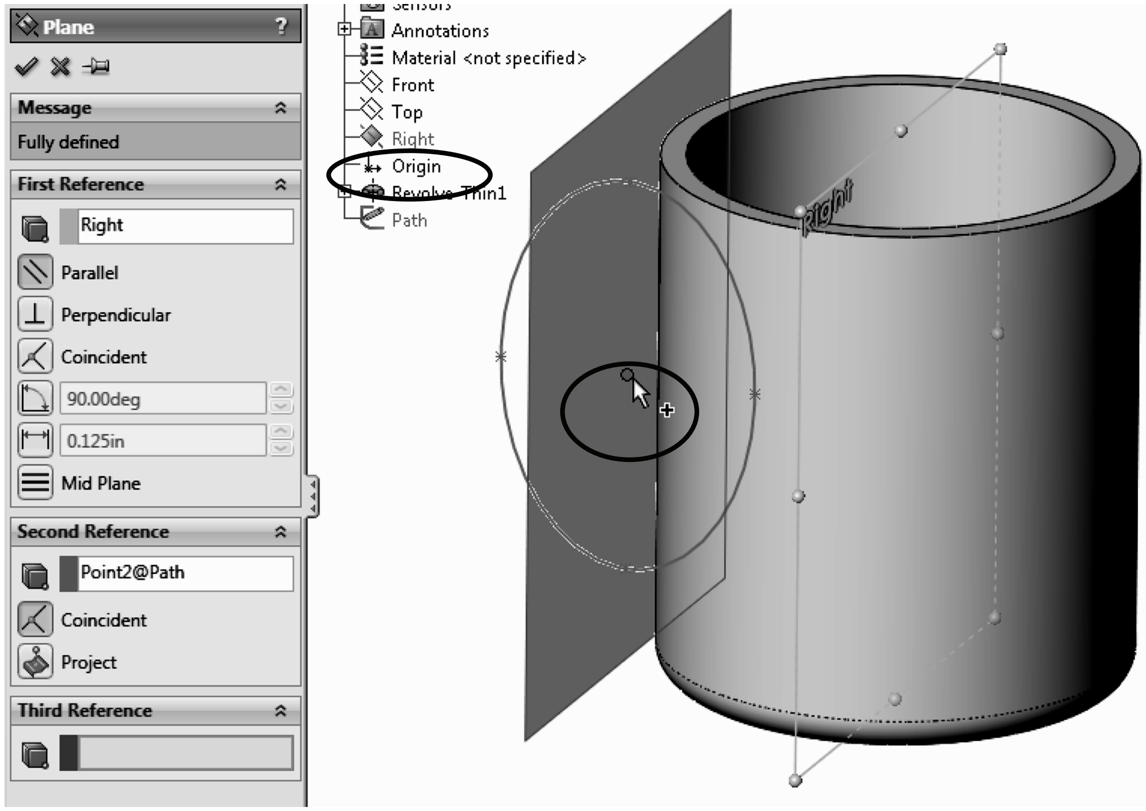
Exit the sketch and rename it *'Path'*. We will not use the sketch for a feature just yet.



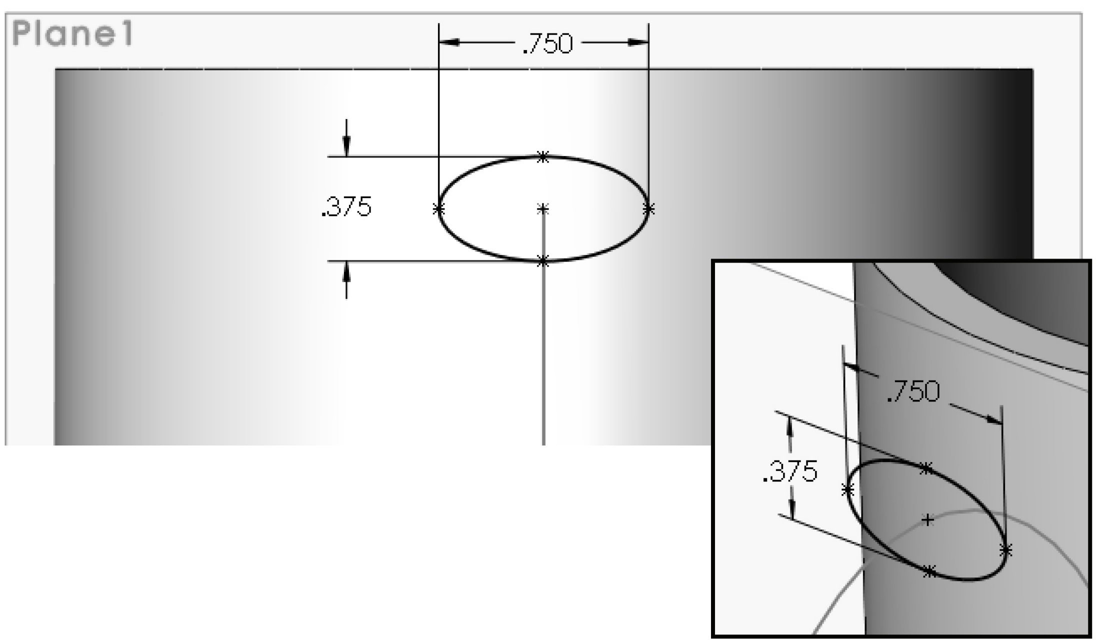


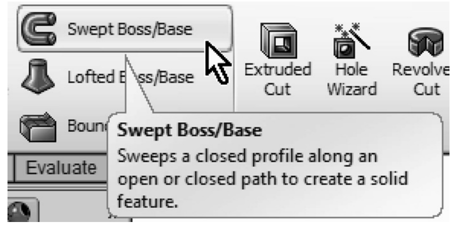
152. - Create an Auxiliary plane parallel to the Right plane using the center of the ellipse as a second reference as shown.



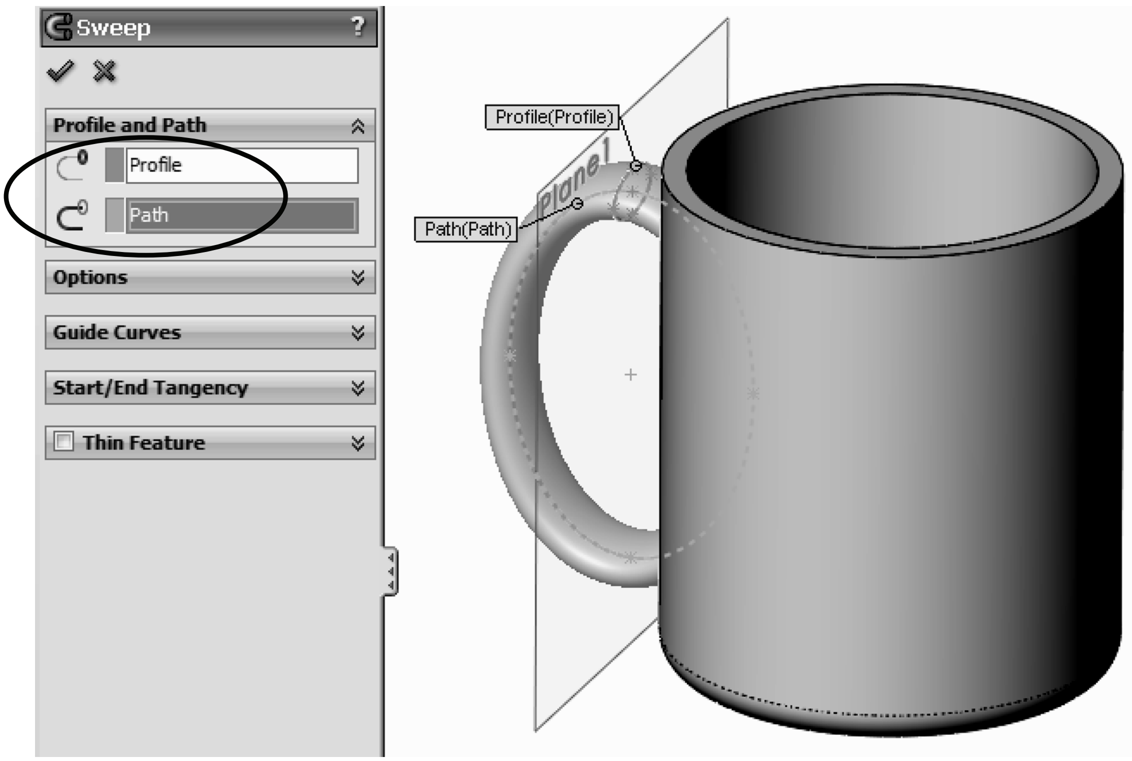


153. - Select the plane just created and draw the next sketch. Start the center of the ellipse at the top point of the previous sketch's ellipse. Add a coincident relation if needed. Remember to add a horizontal relation between the points of the major (or vertical to the minor) axis. Exit the sketch and rename it *'Profile'*.

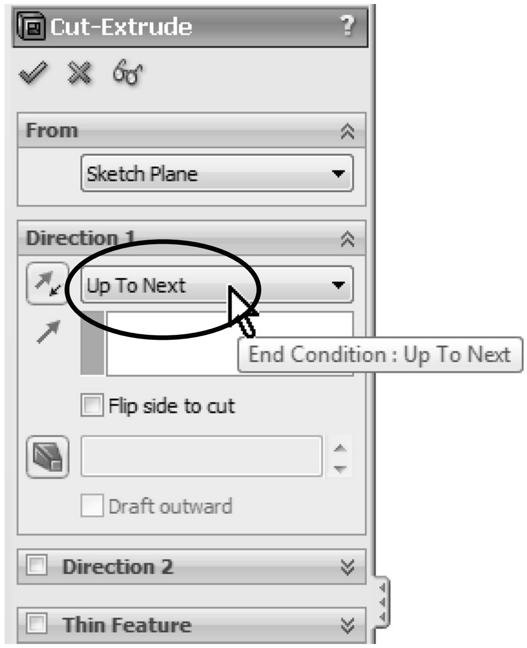


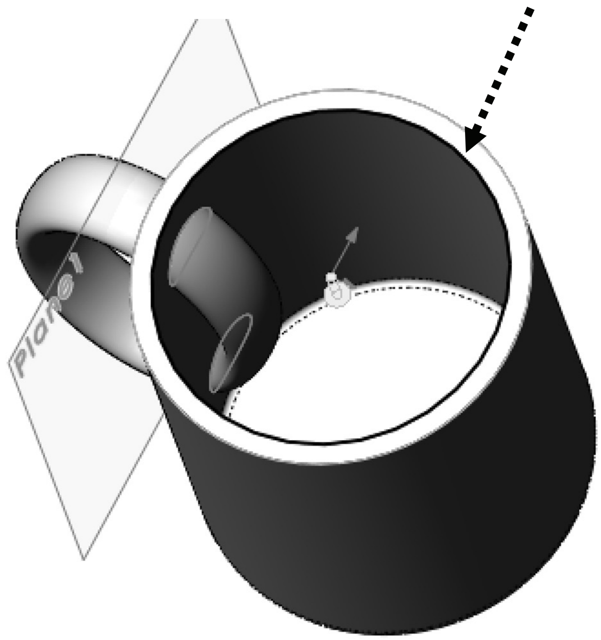


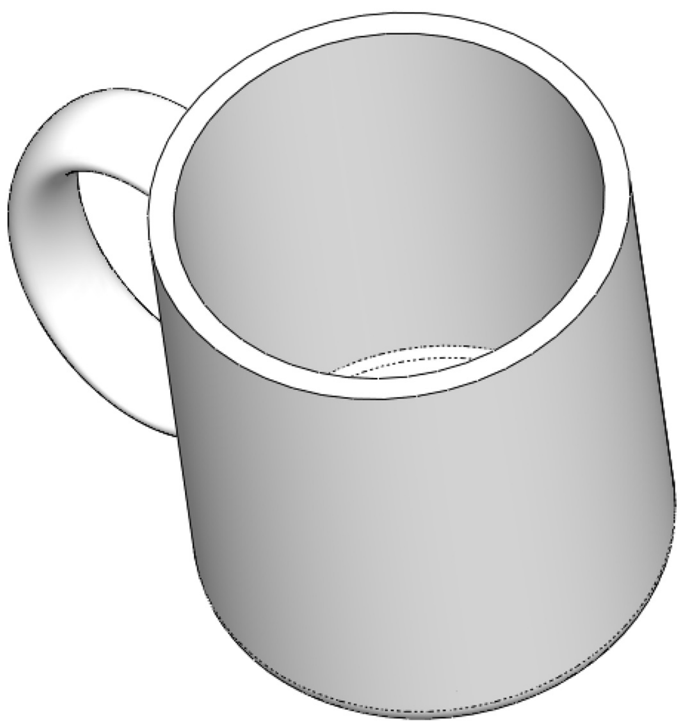
154. - Select the "**Sweep**" icon from the Features tab in the CommandManager. The sweep is a feature that requires a minimum of two sketches: one for the sweep profile and one for the path (the path can also be a model edge or a defined curve). In the "**Sweep**" properties, select the *'Profile'* sketch in the Profile selection box, and the '*Path*' sketch in the Path selection box. The Sweep can optionally have guide curves and other parameters to better control the resulting shape, but we are using only the basic options. Notice the preview and click OK when done. Hide the planes using the "Hide/Show Items" toolbar if so desired.



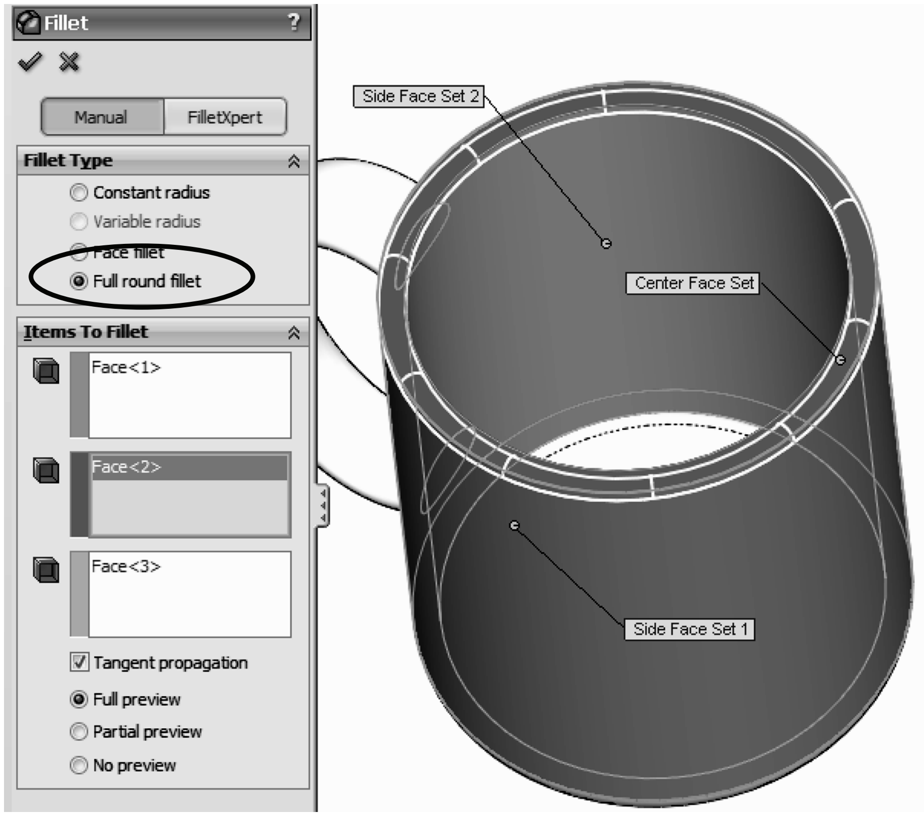
155. - Notice the sweep is also inside the cup. We will make a cut to fix this. Create a sketch in the flat face at the top of the cup. Select the <u>inside</u> edge at the top and use "**Convert Entities**" from the Sketch tab to convert the edge to sketch geometry. Select the "**Extruded Cut**" icon and use the option "Up to next" from the end condition drop down selection.



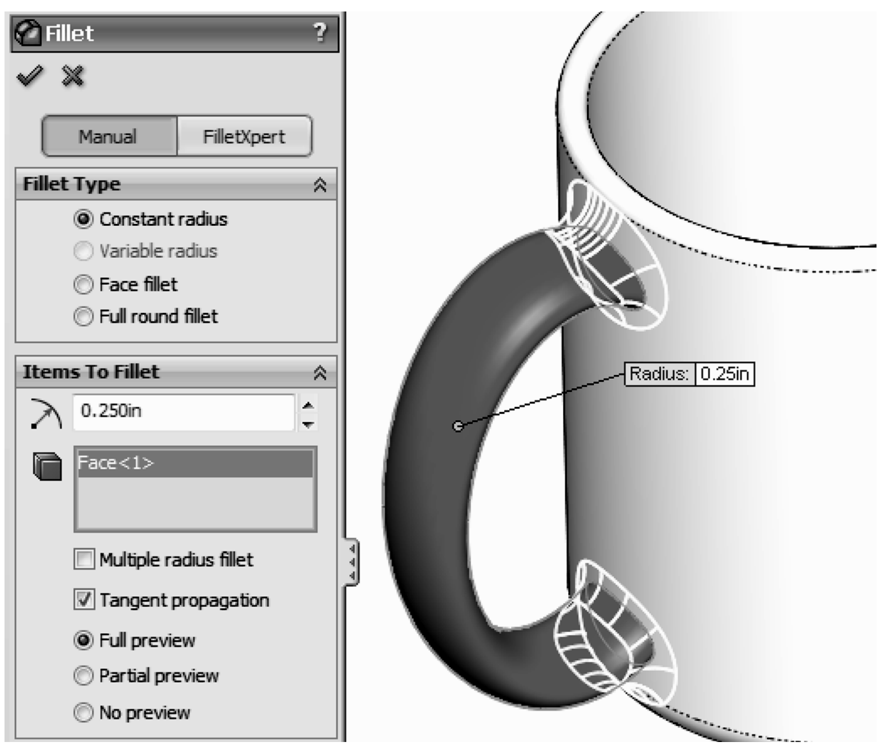




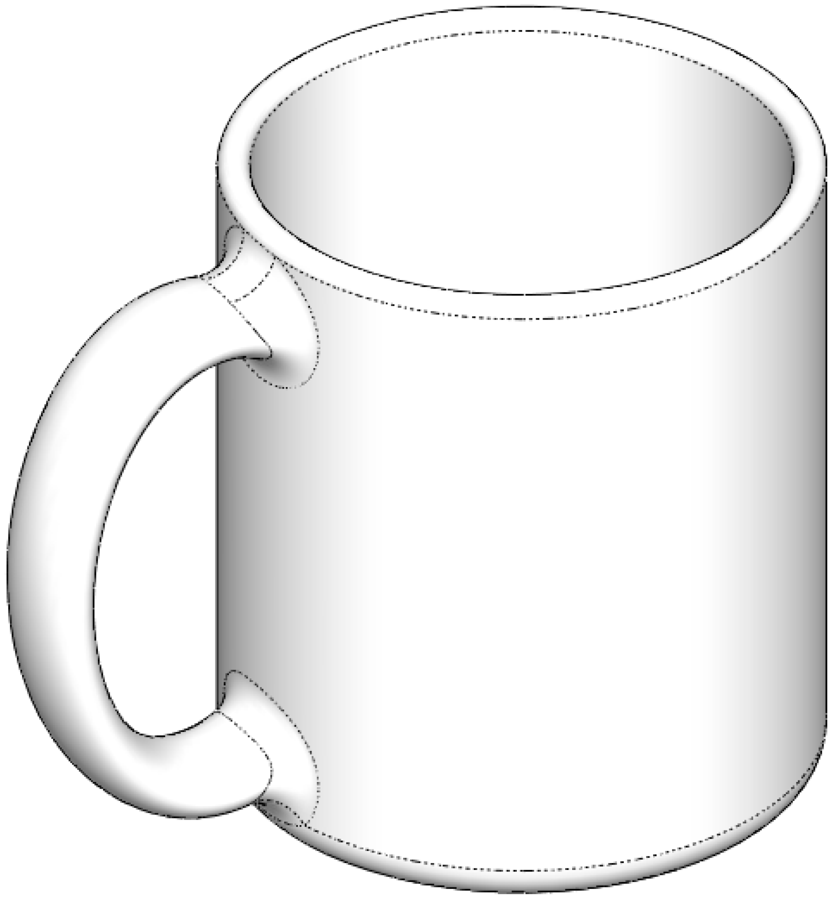
156. - Now we need to round the top lip of the cup; it is currently a flat face. To do this we'll add a "**Fillet**" using the "**Full Round Fillet**" option. There are three faces that we need to select, the middle face will be replaced by the fillet. The first selection box is active; select the outside face of the cup. Click inside the second selection box (Center Face) and select the top flat face of the cup. Click inside the third selection box and select the inside face of the cup. Click OK to finish.



157. - To finish the cup add a fillet to round the edges where the handle meets the cup. Select the fillet command with the "Constant Radius" option, select the handle's surface and change the radius to 0.25″. Click OK to finish.



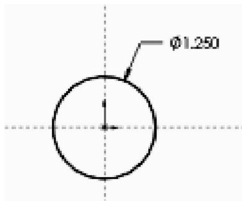
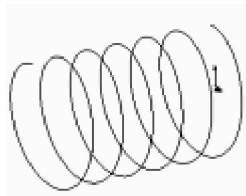
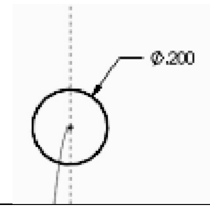
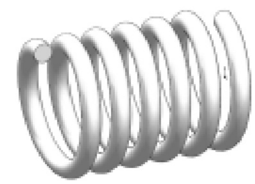
158. - Save the finished part as *'Cup'* and close.



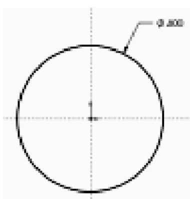
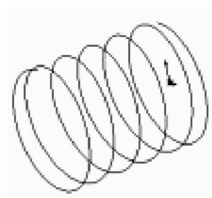
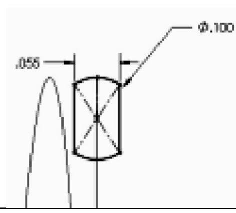
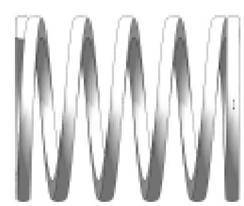
Notes:

In this exercise we are going to show how to make a simple and a variable pitch spring. In order to make these springs we'll have to learn how to make simple and variable pitch helixes to be used for the sweep paths. The sequence of features to complete the springs is:

Simple Spring

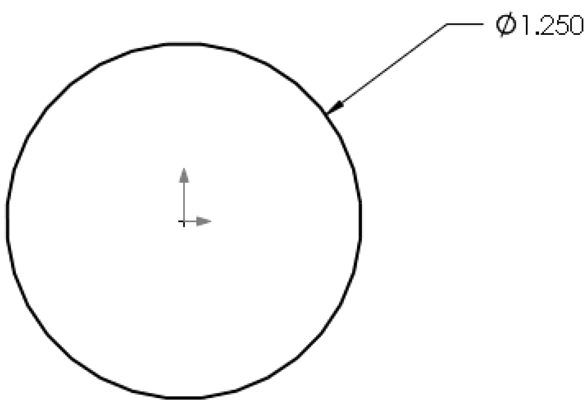
			
Draw circle	Make Helix	Draw Spring Profile Sketch	Make Sweep

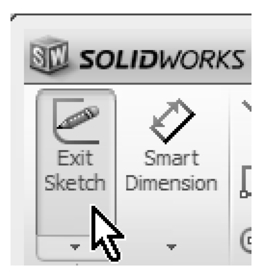
Variable Pitch Spring

			
Draw circle	Make Variable Pitch Helix	Make Spring Profile sketch	Make Sweep and cut ends

Simple Spring

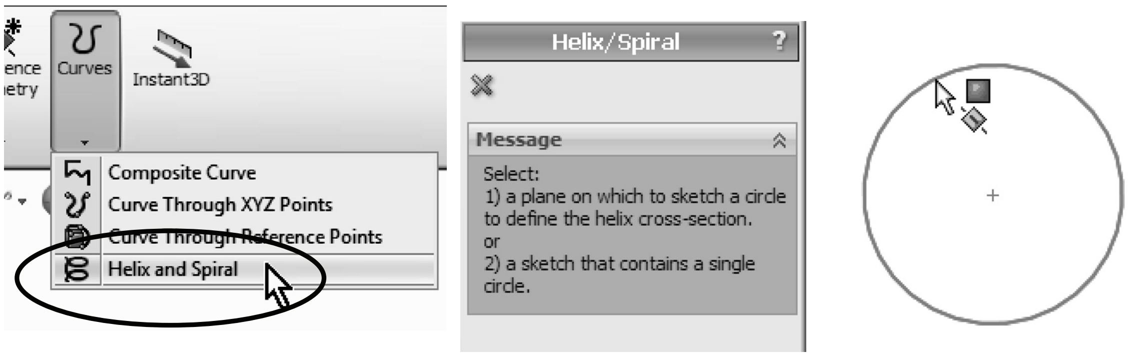
159. - In order to make a helix, first we need to make a sketch with a circle. This circle is going to be the helix's diameter. Select the Front Plane and make a sketch using the following dimensions. Exit the sketch when done.





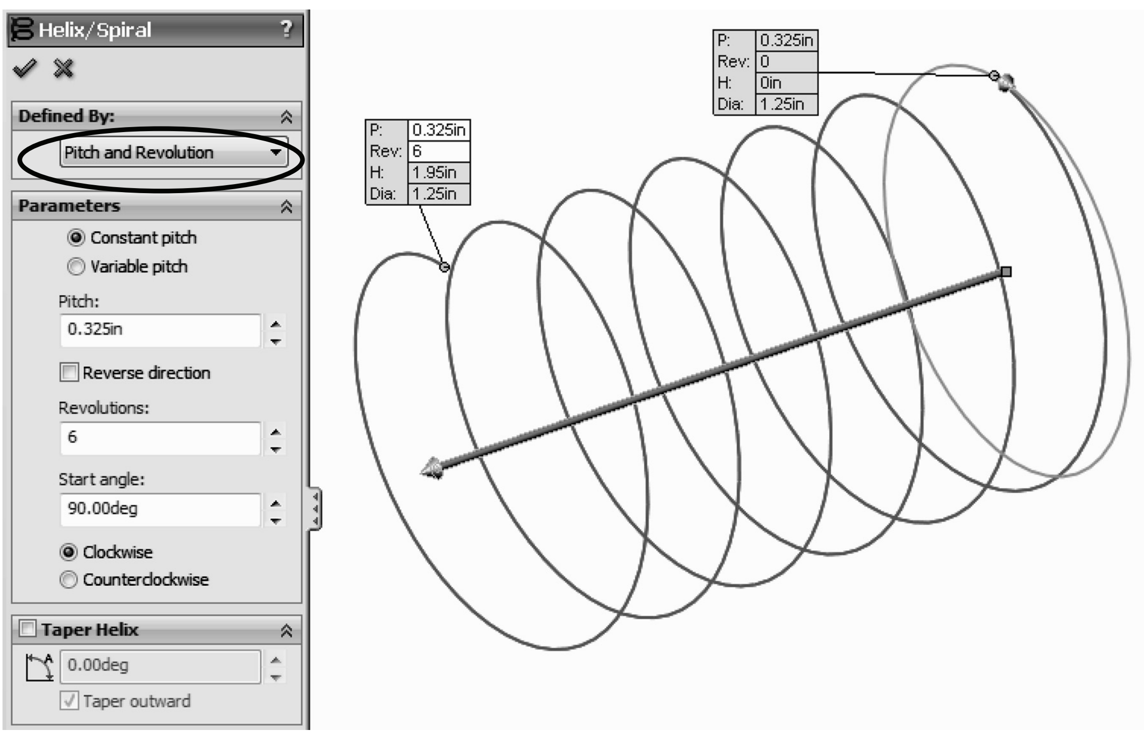
If we don't exit the sketch, we'll only have the "Helix and Spiral" option from the "Curves" command.

160. - In the Features tab select "**Curves**, **Helix and Spiral**" from the drop-down menu. When asked to select a plane or a sketch, select the sketch we just drew.



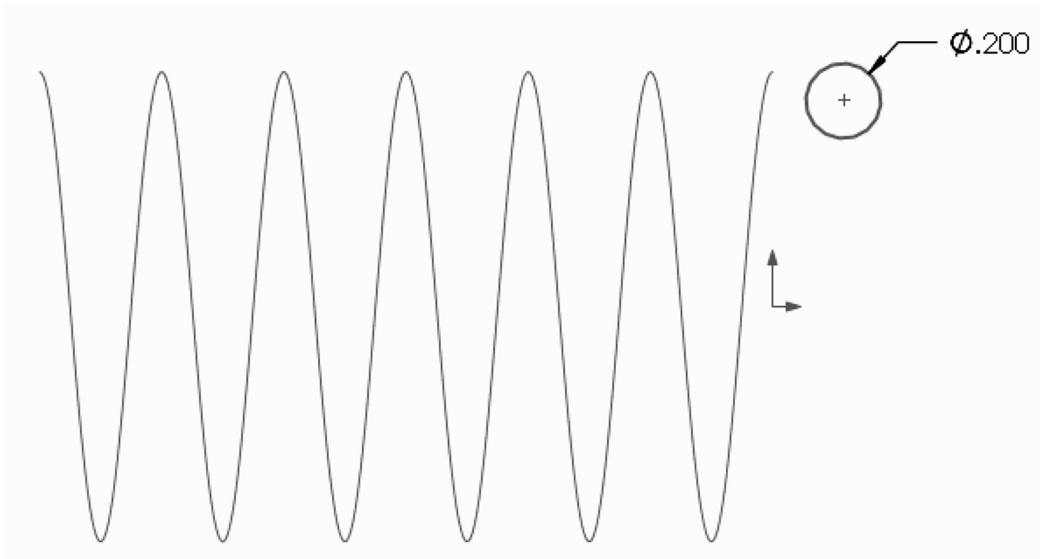
If we select the "**Helix and Spiral**" command before exiting the sketch, SolidWorks will use that sketch automatically for the helix.

161. - The helix can be defined by the combination of any two parameters between Pitch, Revolutions and Height. For this example select "Pitch and Revolution" from the "**Defined By:**" drop down menu, and make the pitch 0.325" and 6 Revolutions. The "Start Angle" value will define where the helix will start. By making it 90 degrees it will start coincident with the Front Plane.

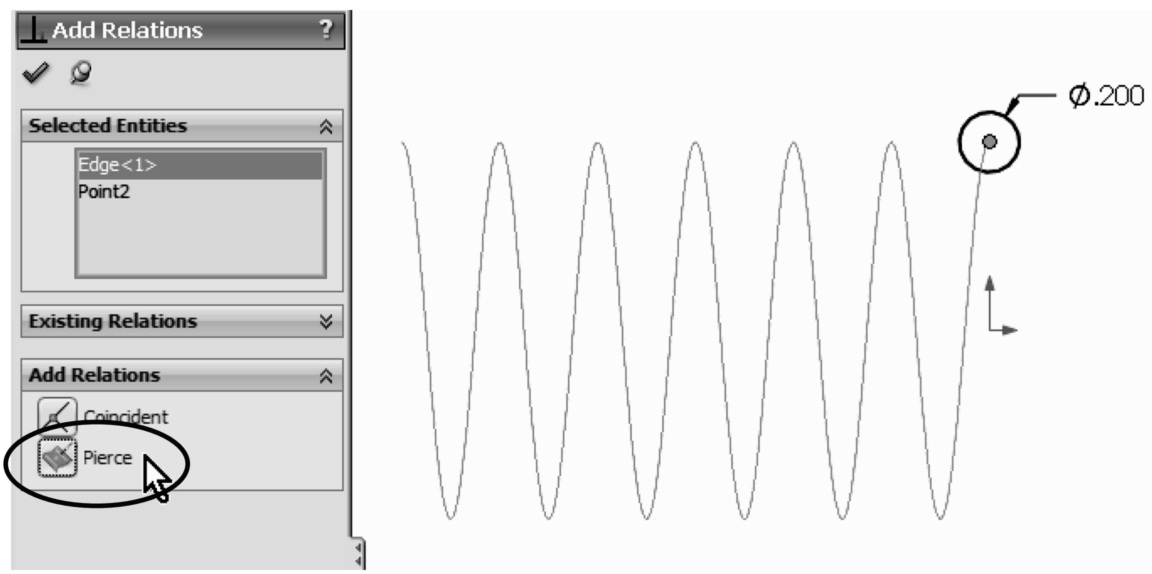


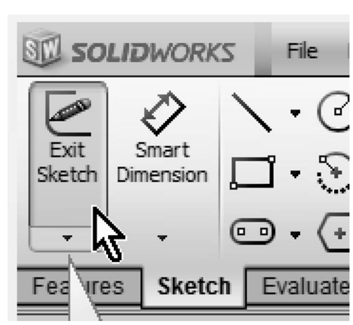
Note that the Helix can be made Clockwise or Counterclockwise, Tapered, Constant or Variable pitch and can also be reversed (going right or left).

162. - Once the Helix is done, we need to make the Profile sketch for the sweep. Switch to a Right view, and add a new sketch in the Right Plane as shown. Make the circle close to the Helix…

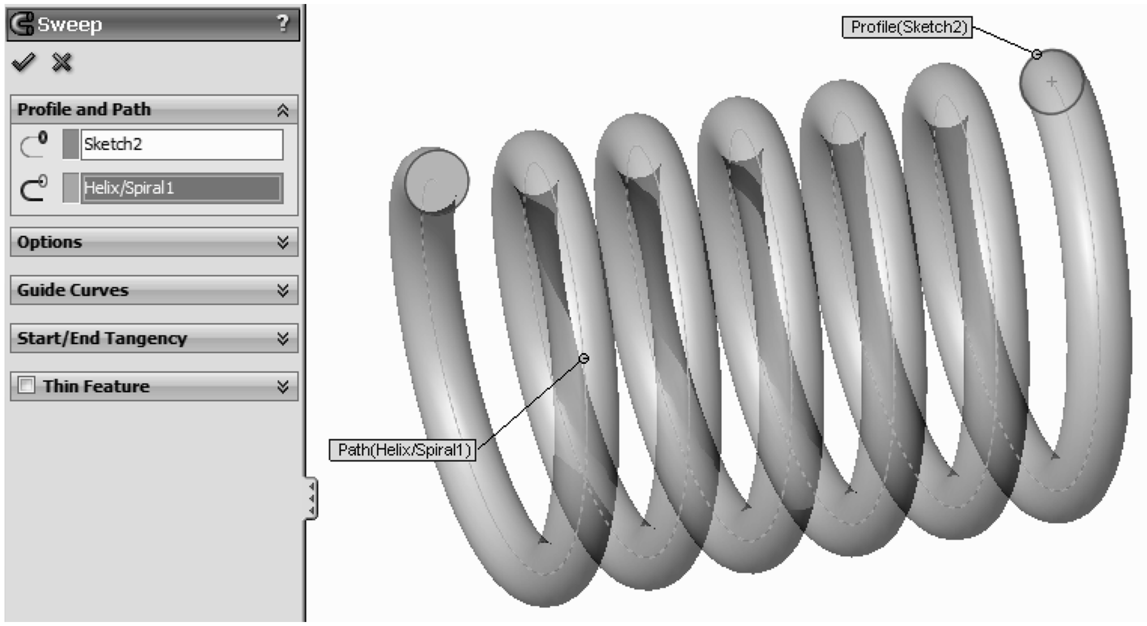


… and add a "**Pierce**" geometric relation between the center of the circle and the helix. This way the path will start at the beginning of the helix. This relation will make the sketch fully defined. Exit the Sketch when done.

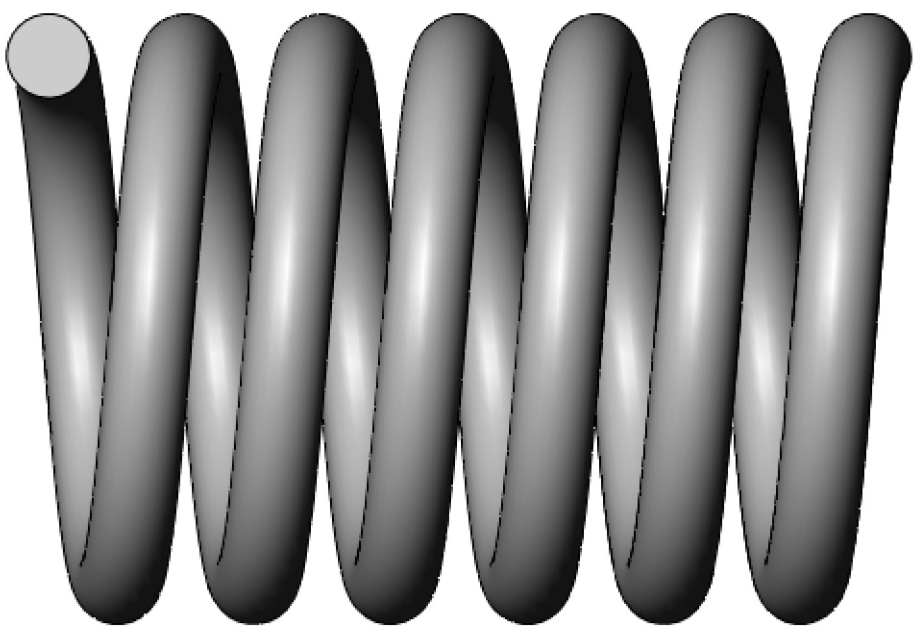




163. - Select the Sweep command and make the sweep using the last sketch as a profile and the Helix as a Path. Notice the Preview.

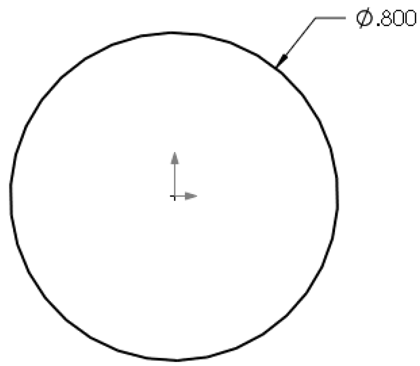


Finished Spring.

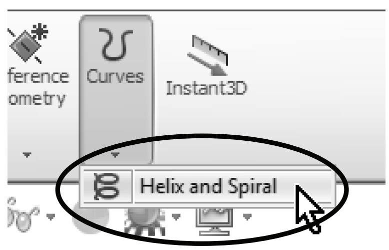


Variable Pitch Spring

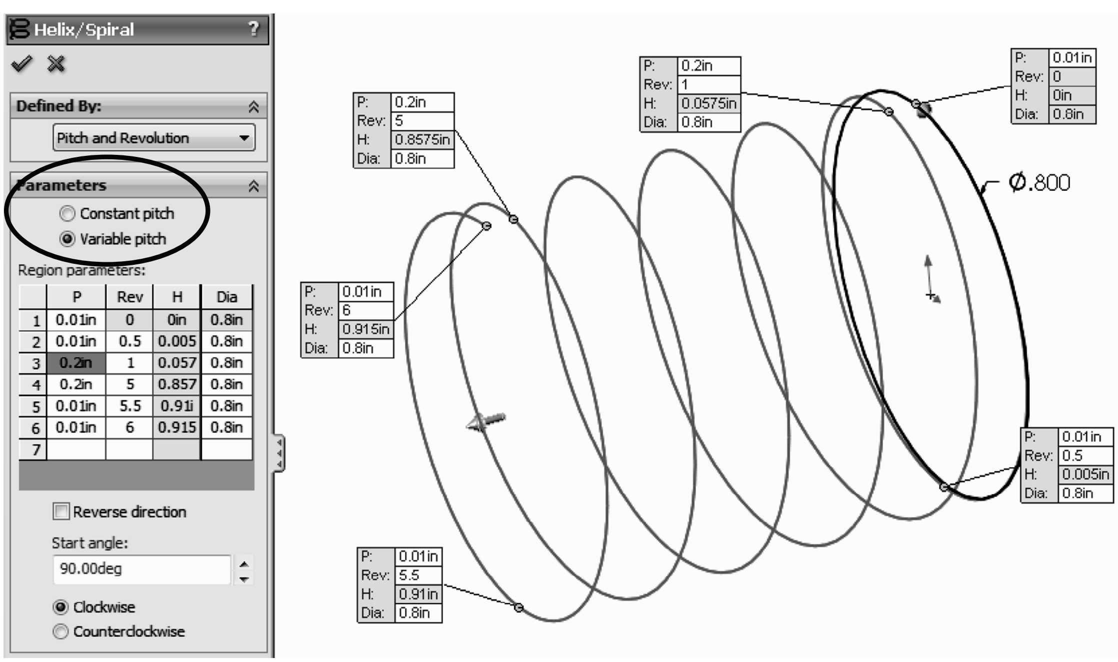
164. - For the variable pitch spring we'll start the same way and make the following sketch in the Front Plane. This will be the spring's outside diameter.



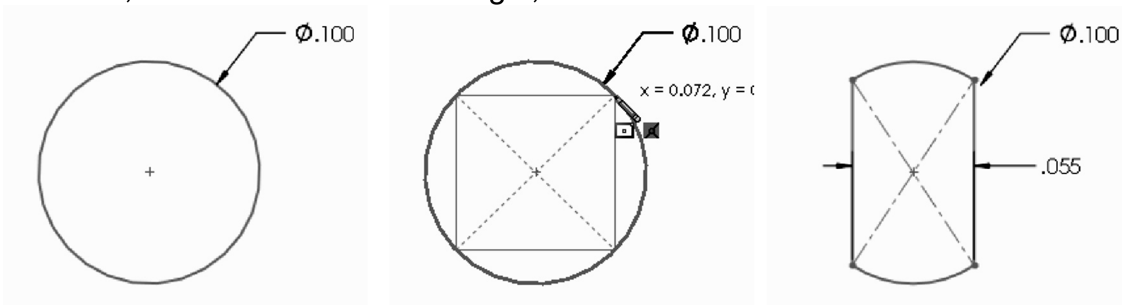
165. - While still editing the sketch, from the Features tab select the "**Curves**" icon and select the "**Helix and Spiral**" command; notice it is the only option available.



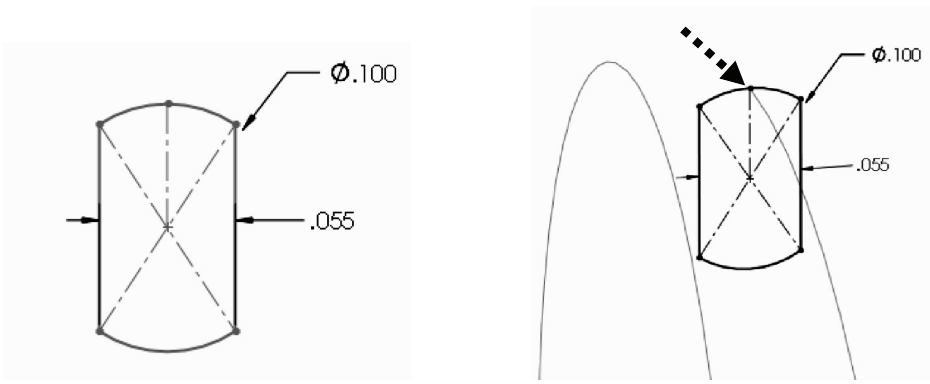
166. - In the **Helix/Spiral** command select "**Pitch and Revolution**" from the "**Defined by**:" selection box and "**Variable Pitch**" in the Parameters box. After selecting it we are presented with a table to fill in the values for the Helix. Fill in the following values. Notice the helix preview. Click OK when finished.



167. - After the helix is complete, add a new sketch in the Right Plane. Draw the circle first, then add a center rectangle; trim and dimension as needed.

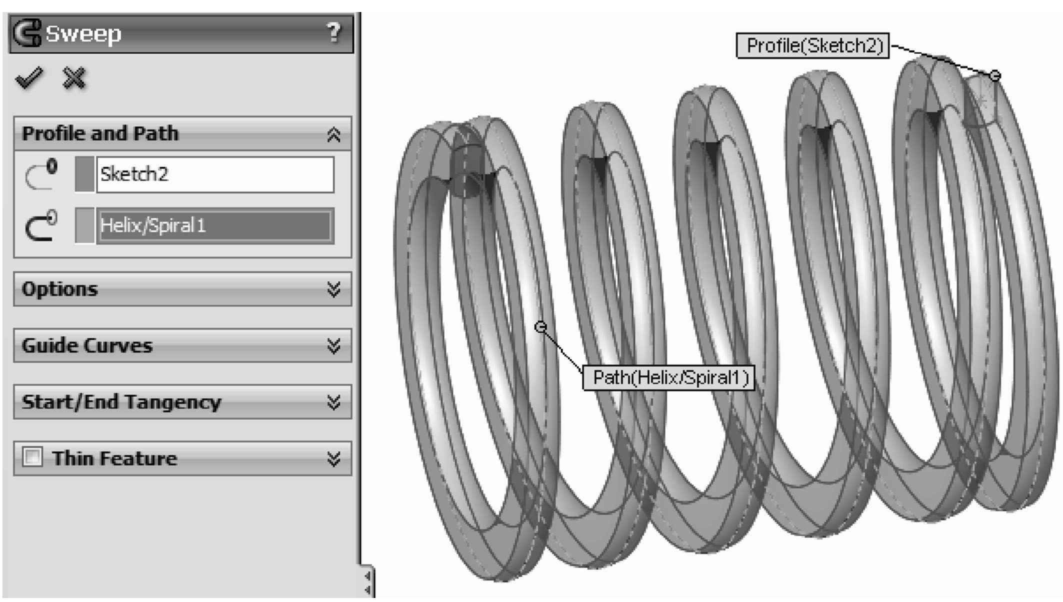


168. - Add a centerline from the center to the top line (make sure it is coincident). Select the endpoint of the centerline and the helix to add a "**Pierce**" geometric relation to make the sketch fully defined. Exit the sketch and optionally rename it *'Profile'*.

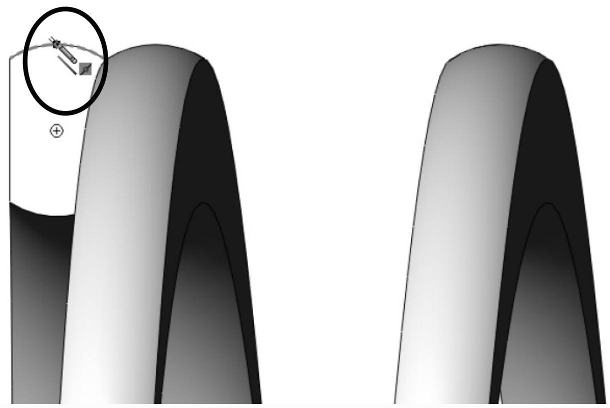


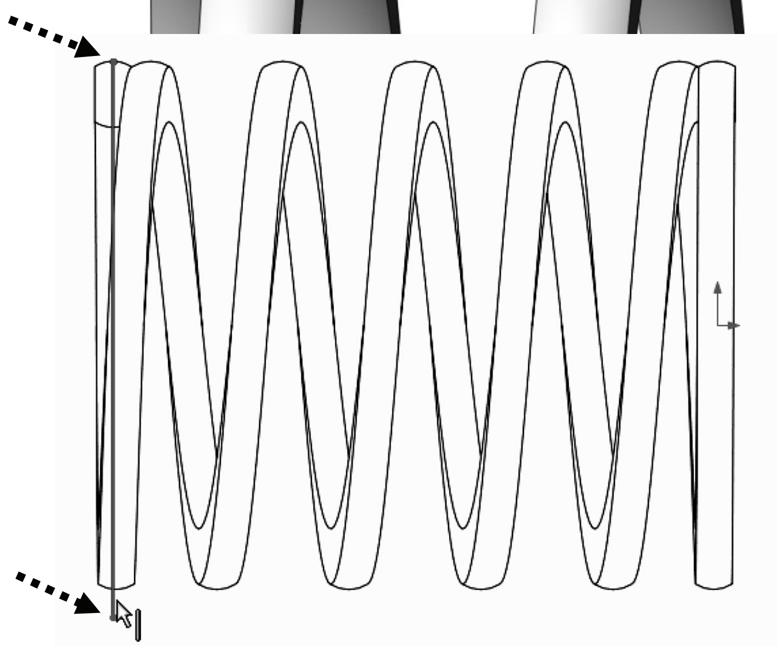
The Pierce relation allows us to fix the profile to the path, and can be added to any part of the sketch. We chose to add it to the top because the original sketch used for the helix is the outside diameter of the spring.

169. - Just as we did before, select the Sweep command and add the path and profile. Click OK when finished.

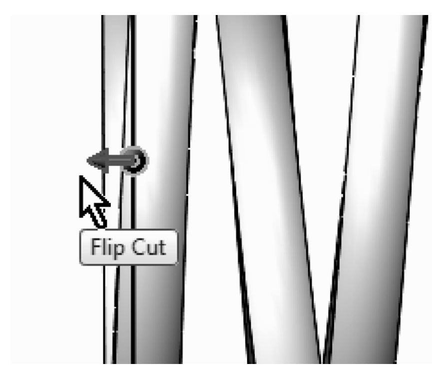


170. – To finish we'll make a cut to the sides to make the ends flat. Change to a Right view and add a sketch in the Right Plane. Draw a single line starting at the midpoint of the edge indicated, and down long enough to cross the part.

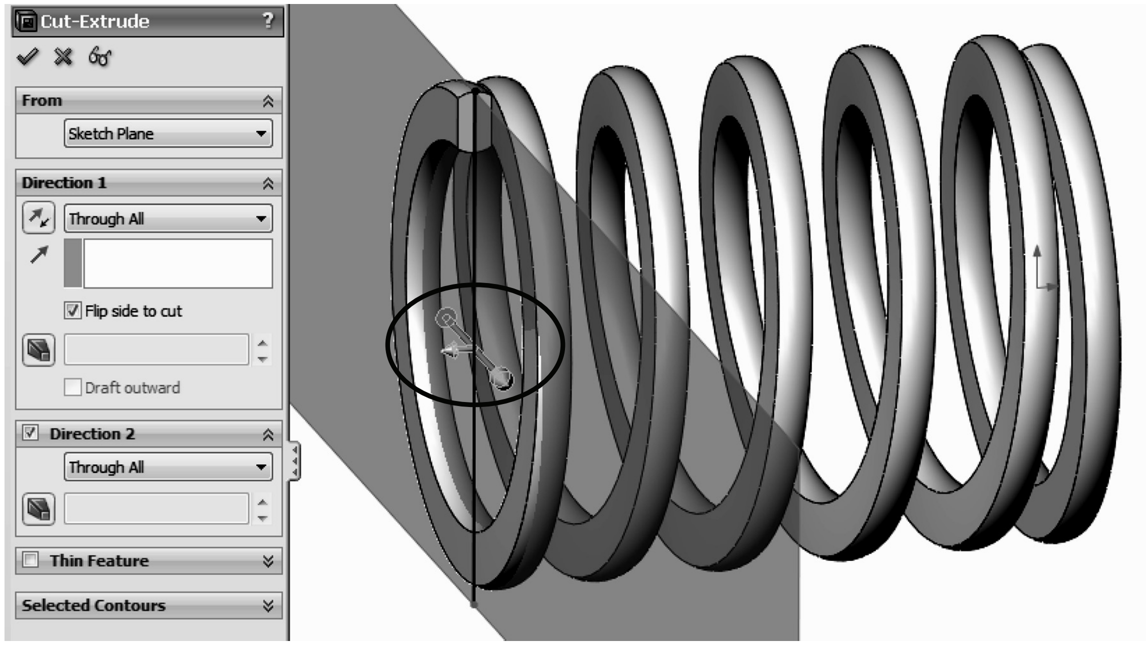


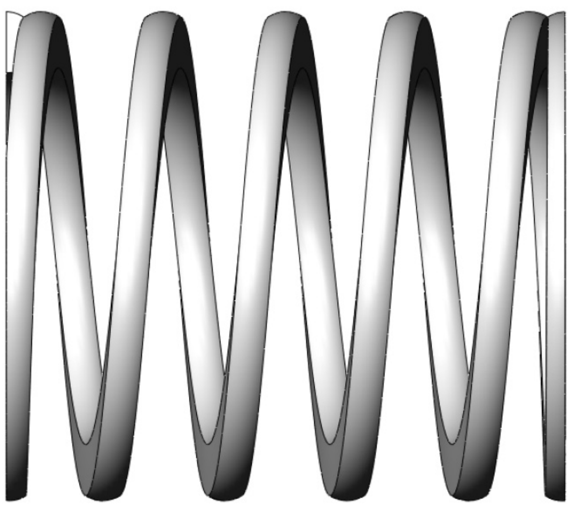


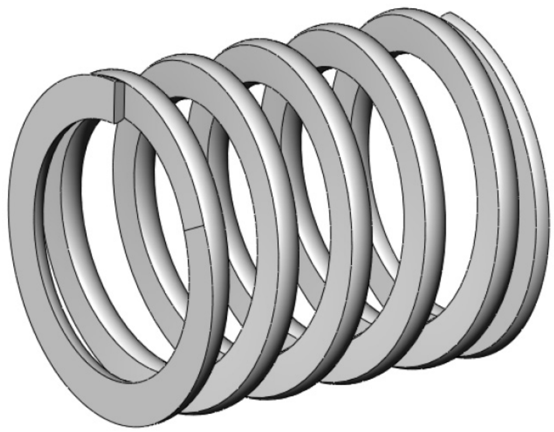
171. - Select the "**Extruded Cut**" command. When we use an open sketch to make a cut, the "Through All" option is automatically selected, and one side of the model is cut using the open sketch. The small arrow located at the center of the line indicates which side of the model will be cut, and we can see the plane used to trim the model. Use the "**Flip side to cut**" option to cut the left side, otherwise the biggest part of the spring will be cut. Click OK to complete the Cut Extrude and repeat it in the right side.



When using an open sketch to cut a model, we can use only a single open profile. If we have multiple open profiles we cannot make the Cut Extrude.





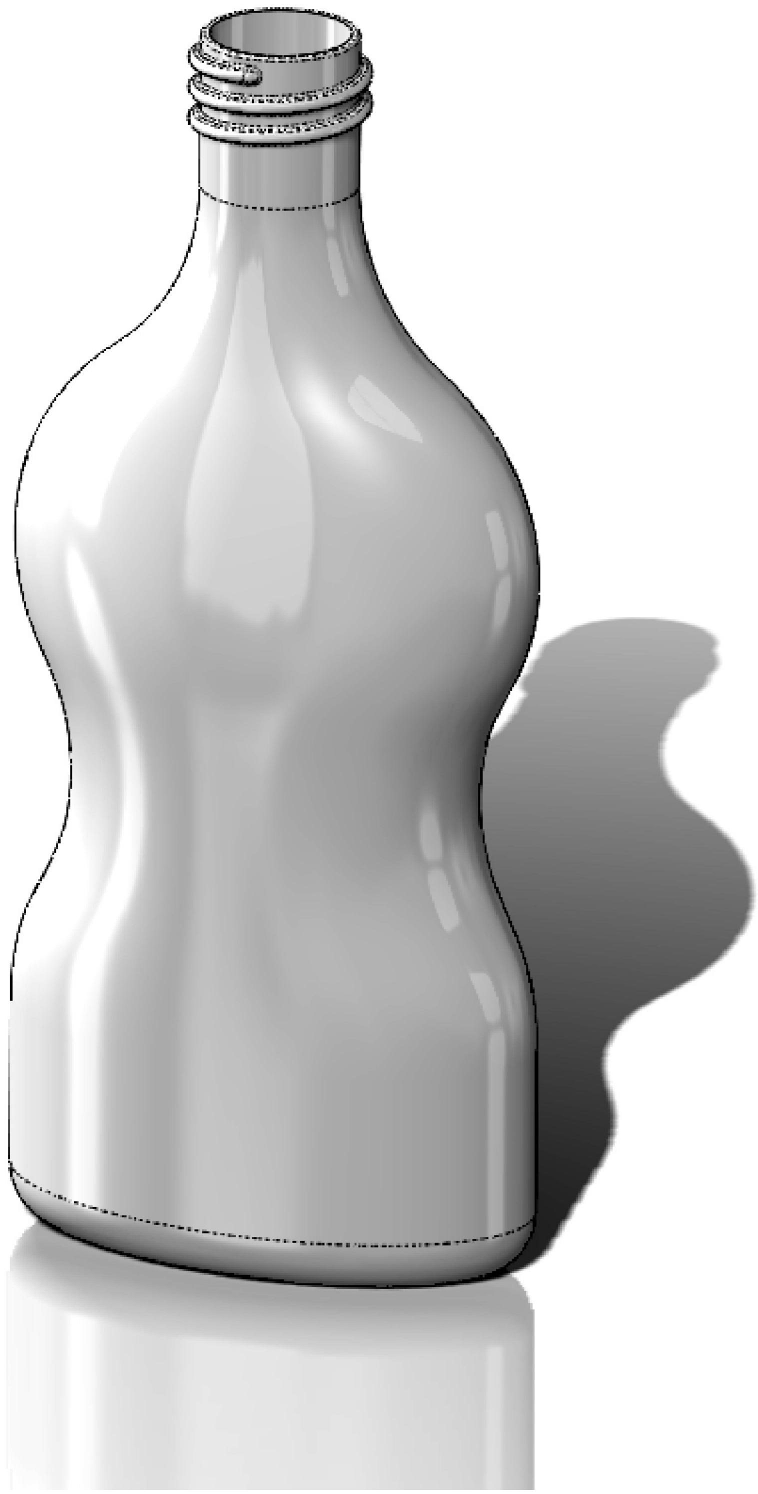


Save the model as "Variable Pitch Spring".

Threads can be added to a model using this approach, or a spiral cut made using the menu "**Insert, Cut, Sweep**" which will remove material. When modeling screws and fasteners in general, it is almost always unnecessary to add a helical thread, as it uses a lot of computing resources, and a simple representation of it usually suffices. It is strongly advised to only add threads when required by the model, as in the bottle example next.

Exercise: Build the following bottle using the knowledge acquired in this lesson. Download the complete instructions from our website at

http://mechanicad.com/download.html



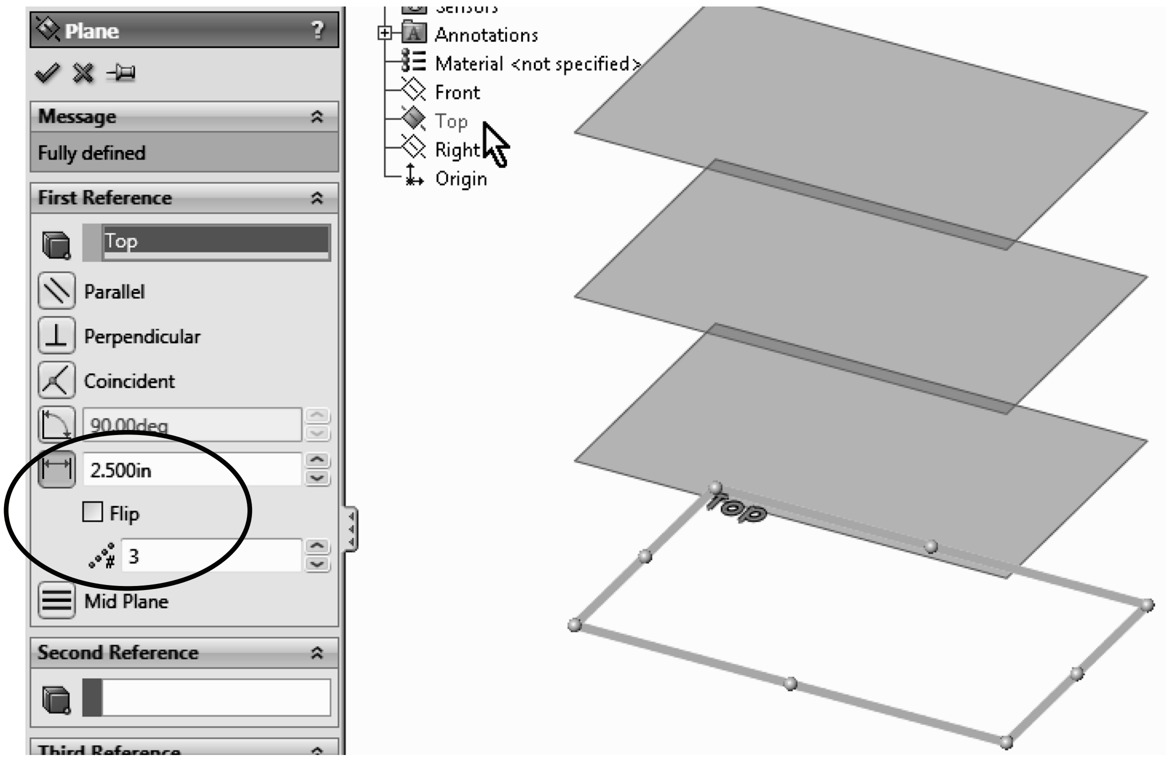
Notes:

Loft: Bottle

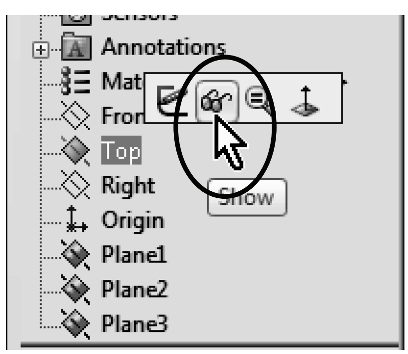
Notes:

The "**Loft**" feature requires at least two different sketches and/or faces and optionally guide curves to more accurately define the final shape. The **Loft** helps us design more complex shapes. In this exercise we will make a bottle using a loft with four different sketches.

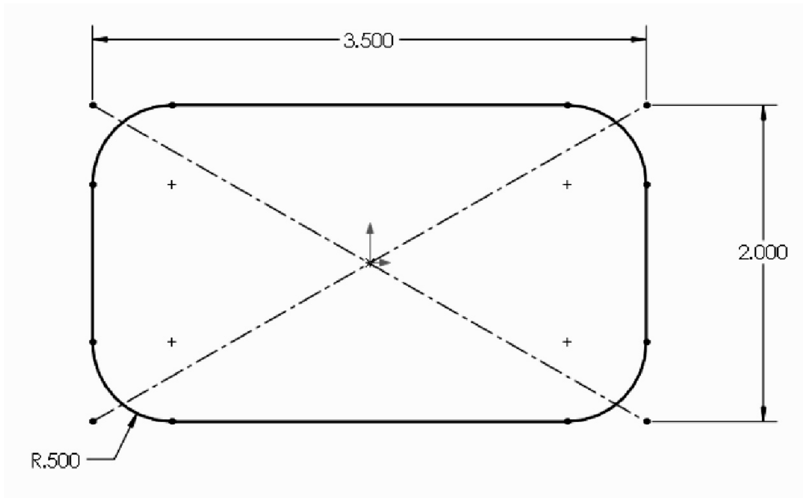
172. - Make a new part document and create three auxiliary planes using the "**Plane**" icon from the Reference Geometry drop-down menu in the Features tab. Select the Top Plane as reference, change the number of planes to 3 and space them 2.5″ as shown. Click OK when done.



173. - For clarity, select the Top Plane in the FeatureManager and show it using the "Hide/Show" command from the pop-up toolbar.

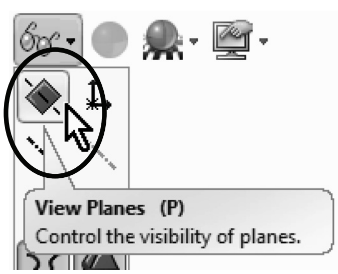


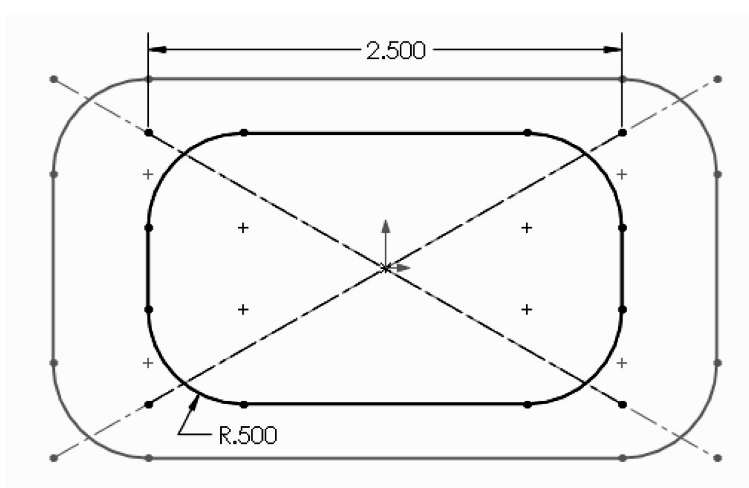
174. - Switch to a Top view, select the Top Plane and draw the following sketch using the "**Center Rectangle**" and "**Sketch Fillet**" tools. Exit the sketch when finished.



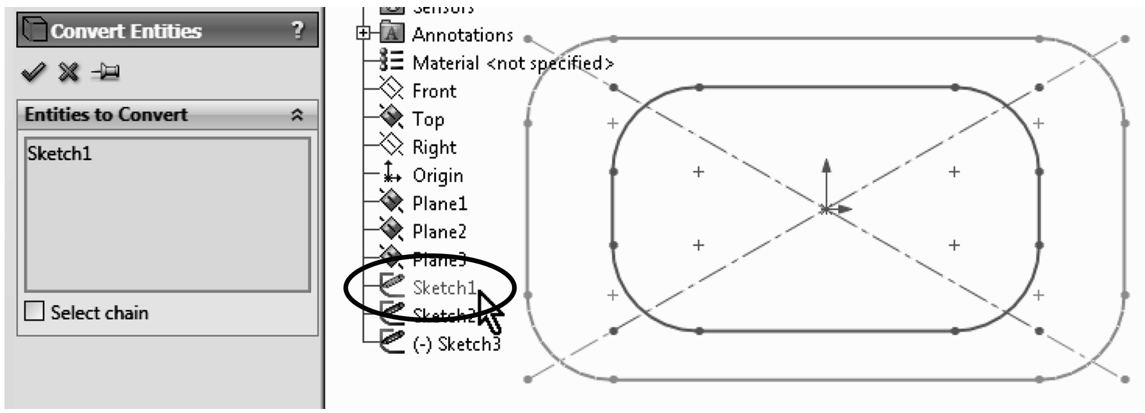
175. - Still in the Top view, select *'Plane1'* from the FeatureManager and create the second sketch. Use the "**Center Rectangle**" tool; be sure to start in the origin and make the rectangle's corner coincident to the previous sketch's diagonal line. Add a 2.5″ width dimension and the 0.5″ "**Sketch Fillet**" to fully define the sketch. Exit the second sketch when finished.

Turn off the display of Planes for clarity with "Hide/Show Items".

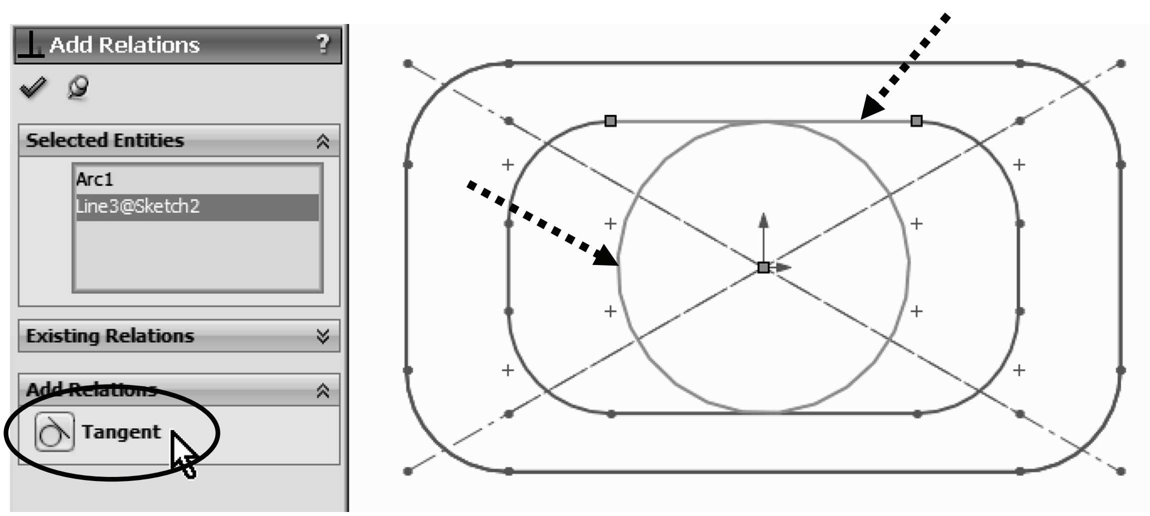




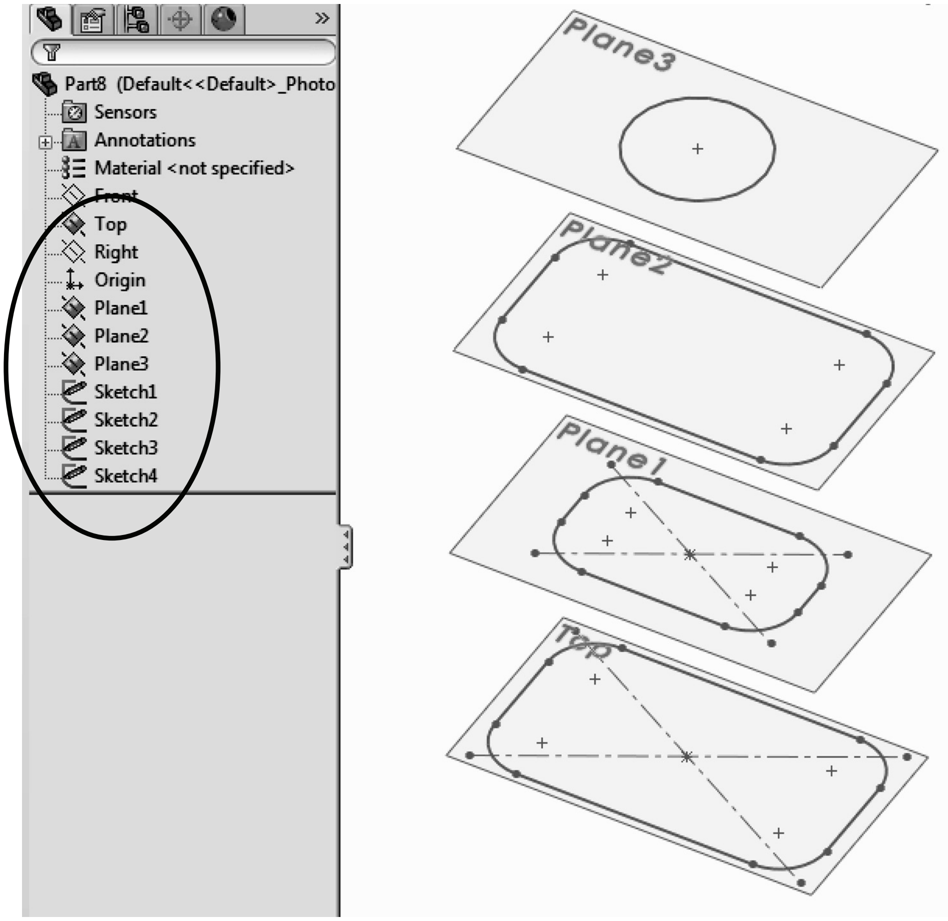
176. - For the third profile, select *'Plane2'* from the FeatureManager and create a new sketch in it. This sketch will be exactly the same as the first one. To help us save time and maintain design intent, we'll use the "**Convert Entities**" tool. In the "**Convert Entities**" selection box, select the entire *Sketch1* from the fly-out FeatureManager and click OK to project Sketch1 in the new sketch. Exit the sketch when done.



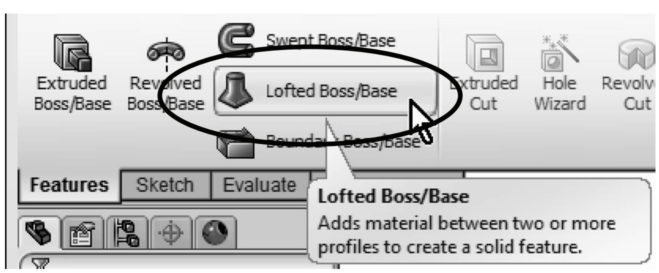
177. - For the last profile select 'Plane3' from the FeatureManager and create a new sketch. Draw a circle and add a geometric relation to make it "Tangent" to the horizontal line in *Sketch2* as indicated by the arrows. This relation will fully define the sketch. Exit the sketch to finish.



178. - The finished sketches will look like this with the planes visible:

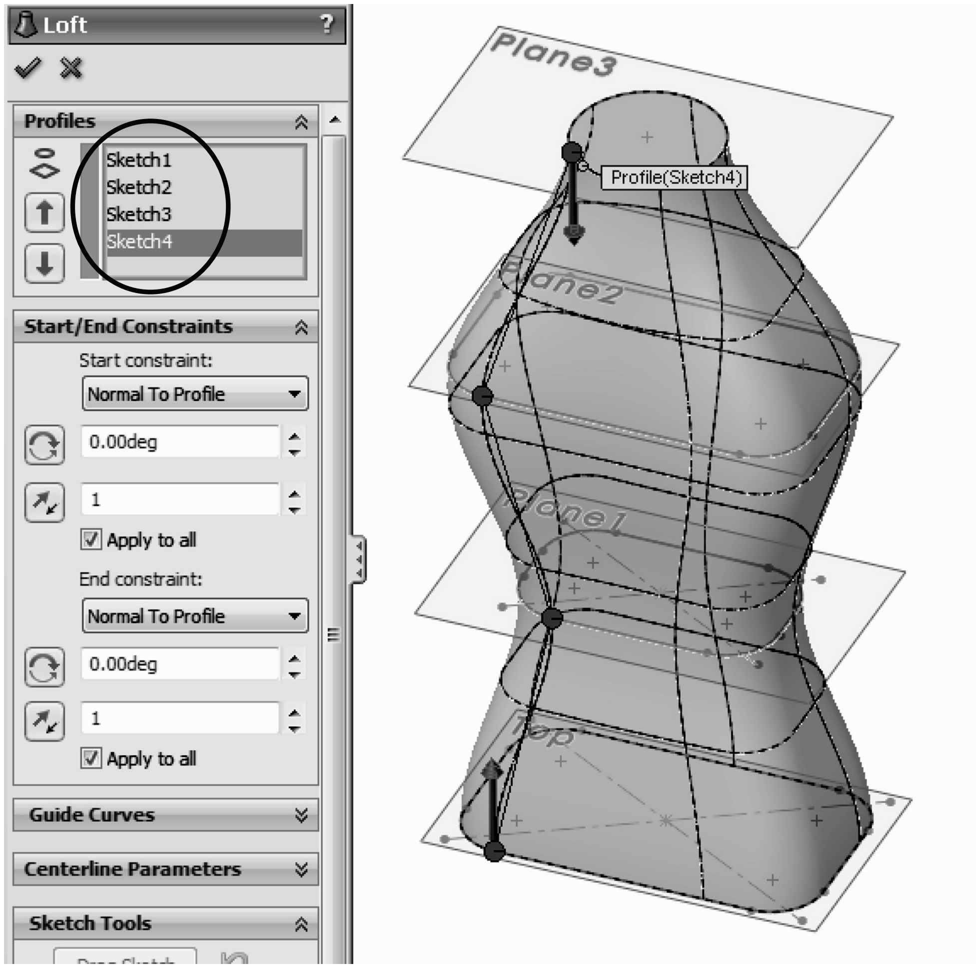


179. - Now we are ready, select the "**Loft**" icon from the Features tab. The loft feature requires two or more sketches and/or faces, and we'll use the four sketches we just made to build the bottle.

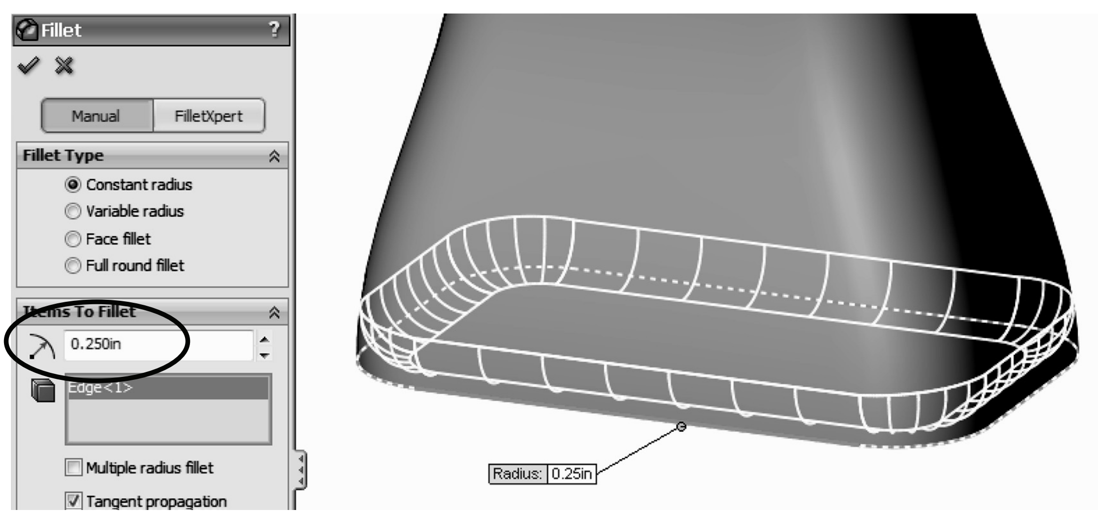


We will select the profiles starting with the first one we made at the bottom and finishing with the last one at the top. It is important to select the sketches thinking that where we select the profile will affect the result. Click in the graphics area near the indicated 'dots', this line indicates the segment of one profile that will be connected to the next profile. If we select points randomly in the profile, the loft could twist and produce undesirable results. Optionally, guide curves can be added to improve control of the resulting shape.

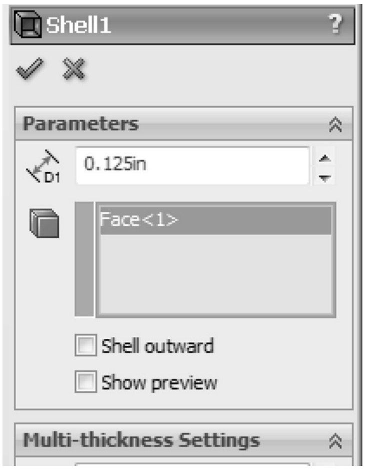
From the "Start/End Conditions" select "Normal to Profile" for both the "Start" and "End" constraint. Notice the difference in the preview after selecting the start and end constraints. Click OK when done.



180. - Add a 0.25″ radius fillet in the bottom edge of the part to round it off.



Finish the part using the "Shell" tool; select the top face to remove it and make the wall thickness 0.125″.



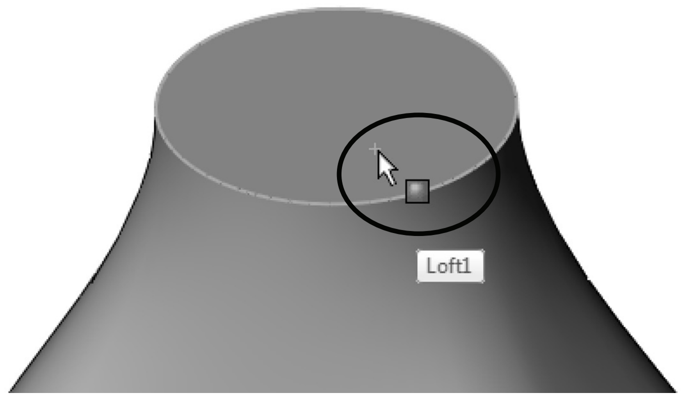
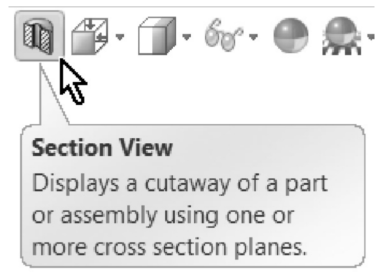
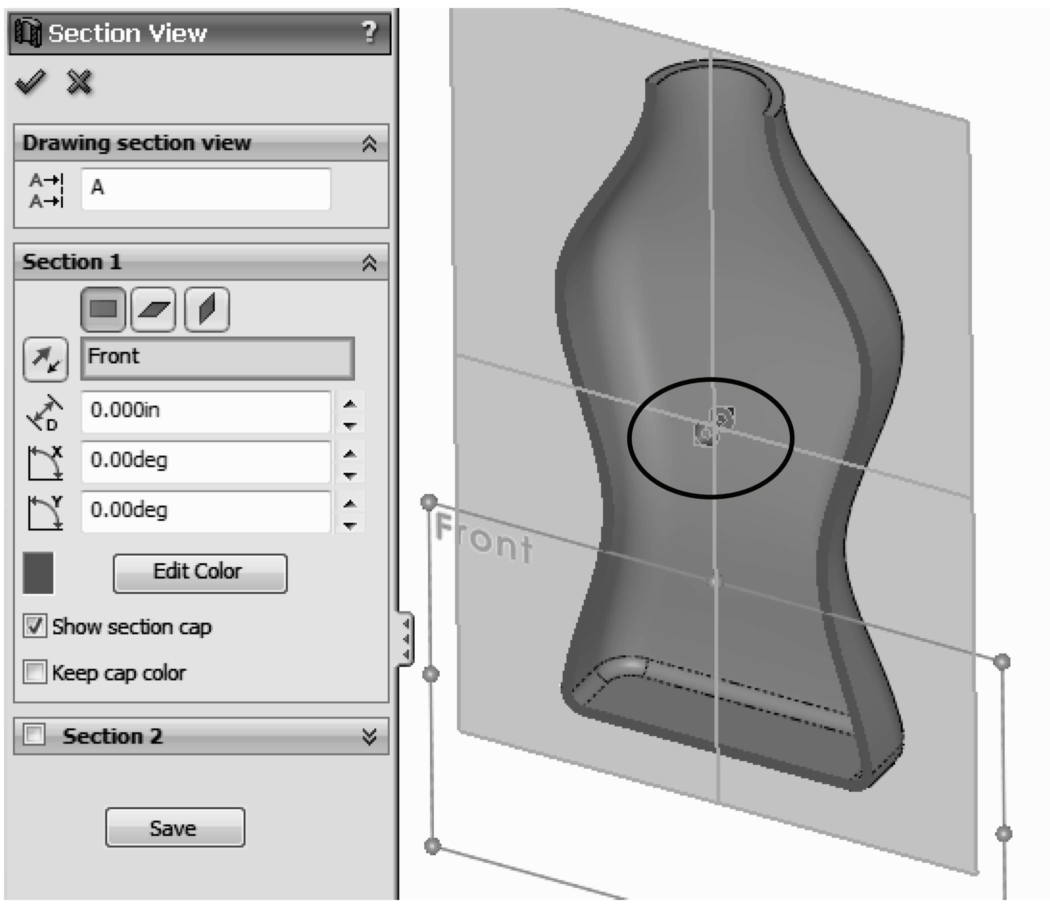


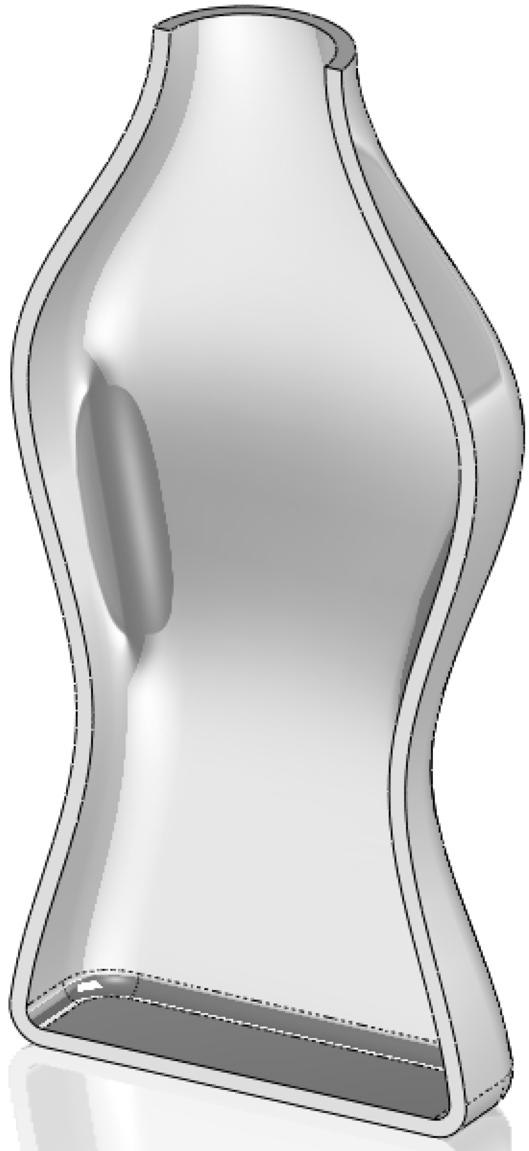
Image shown using Real View

181. - To view the inside of the part, select the "**Section View**" icon from the View toolbar or the menu "**View, Display, Section View"**. We can define which plane to cut the model with, the depth of the cut and optionally add a second section and third section. If we click OK in the Section View, the model will be displayed as cut, but this is only for display purposes; the part is not actually cut. To turn off the Section View, select its icon or menu command again. This section view can be used along with the "Measure" tool to inspect the part.



We can also change the depth of the cut by dragging the arrow in the center of the plane.





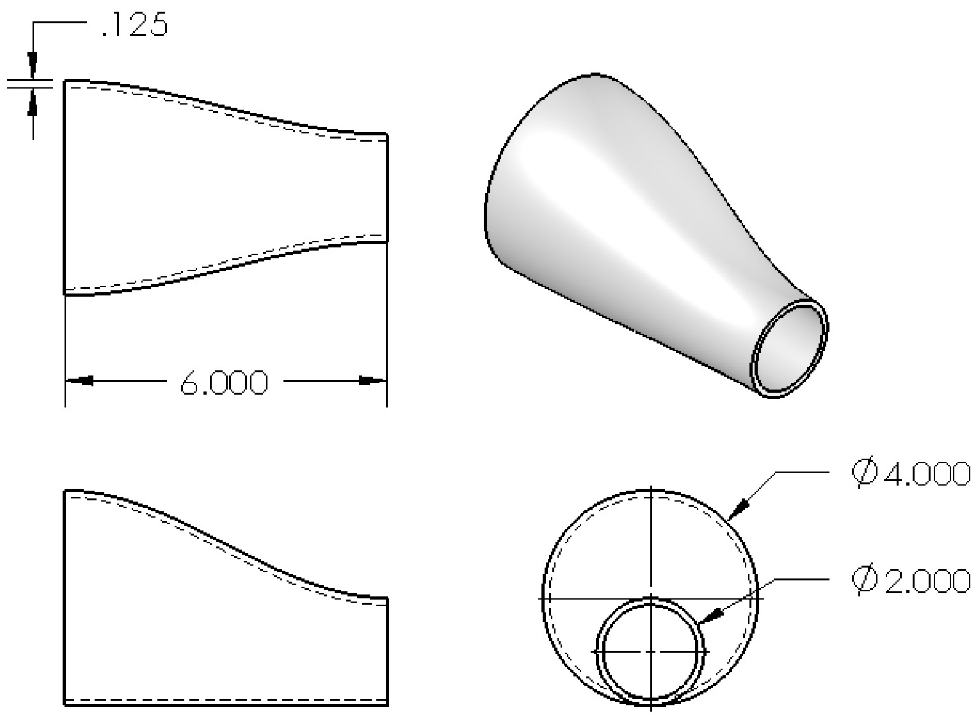
Save the part as "*Bottle Loft*" and close.

<u>Exercises:</u> Build the following parts using the knowledge acquired in this lesson. Try to use the most efficient method to complete the model.

Eccentric Coupler

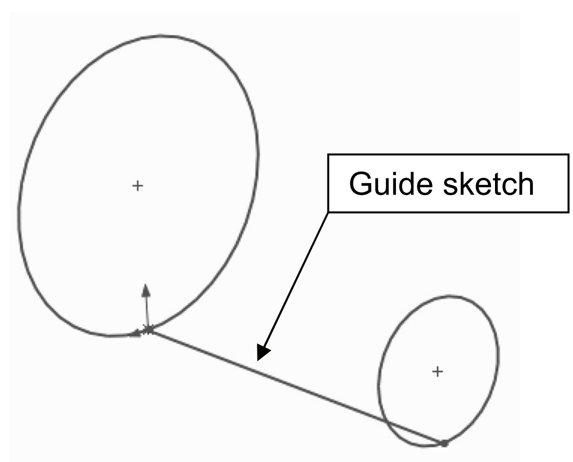
Notes:

- Both circles are centered horizontally (Right view).
- Add a guide sketch at the bottom.
- Set the Start and End Conditions for the loft to “Normal to Profile”.
- Make as Thin Feature *or* Shell it after making the loft feature.



HINTS:

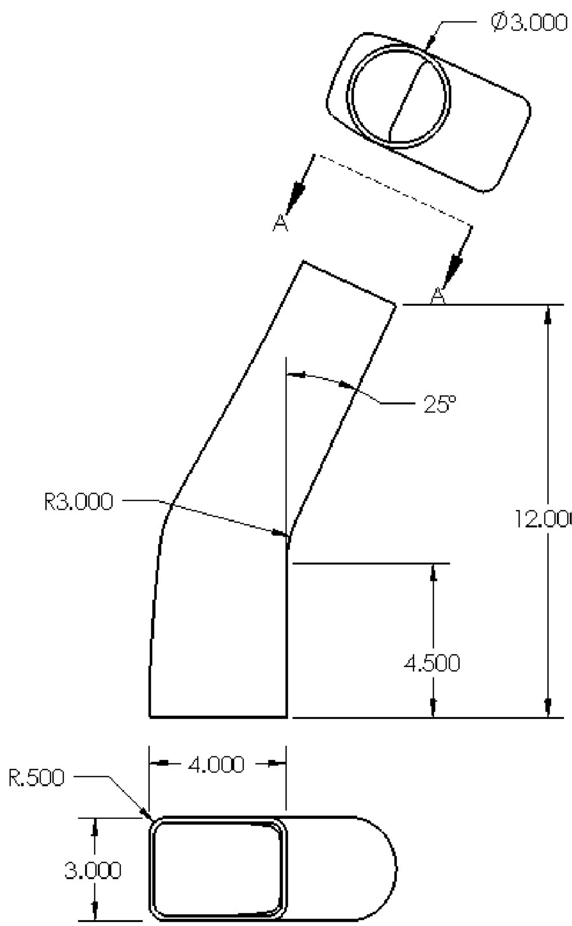
- Draw the 1" circle in the Right Plane.
- Make an Auxiliary Plane 6" parallel to the Right Plane.
- Draw a 2" circle in the Auxiliary Plane.
- Draw a sketch in the Front Plane to make the guide curve.
- Select the guide sketch in the "Guide Curves" selection box.

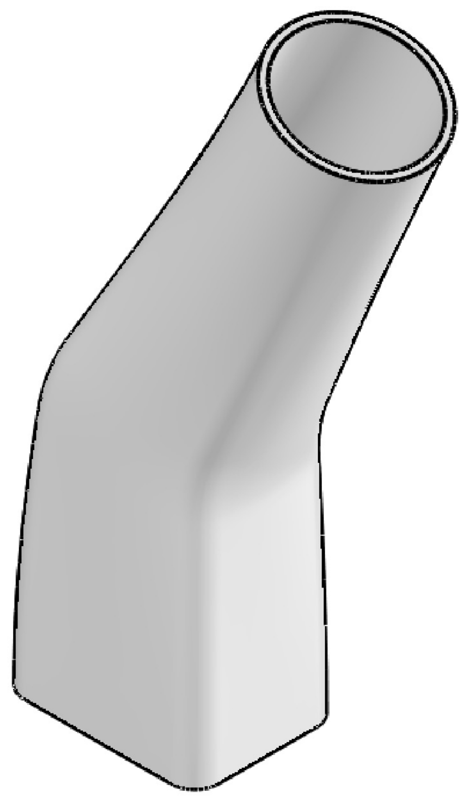


Bent Coupler

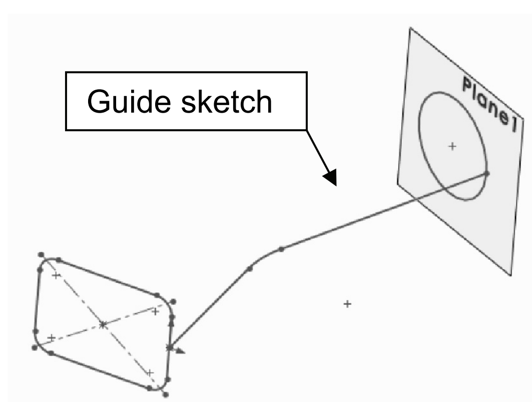
Build the following part using a Loft feature and a shell. The part is 0.15″ thick.

Notes:
- Add a guide sketch along the right side of the part.
- Start and End Conditions "Normal to Profile".
- Make as Thin Feature or Shell after loft.





HINTS:
- Make Rectangular sketch first.
- Make Guide sketch.
- Make Auxiliary Plane perpendicular to Guide sketch and the end point.
- Make circular sketch in Auxiliary Plane.
- Make Loft using Guide sketch as a guide curve.
- Shell the part.



Challenge Exercises:

In order to save space (and trees), visit our web site to download the complete details for the gas grill project as well as the finished parts for this book, higher resolution images of the exercises and some extra topics not covered in the book. Build these parts and after the Assembly lesson make the Gas Grill assembly.

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