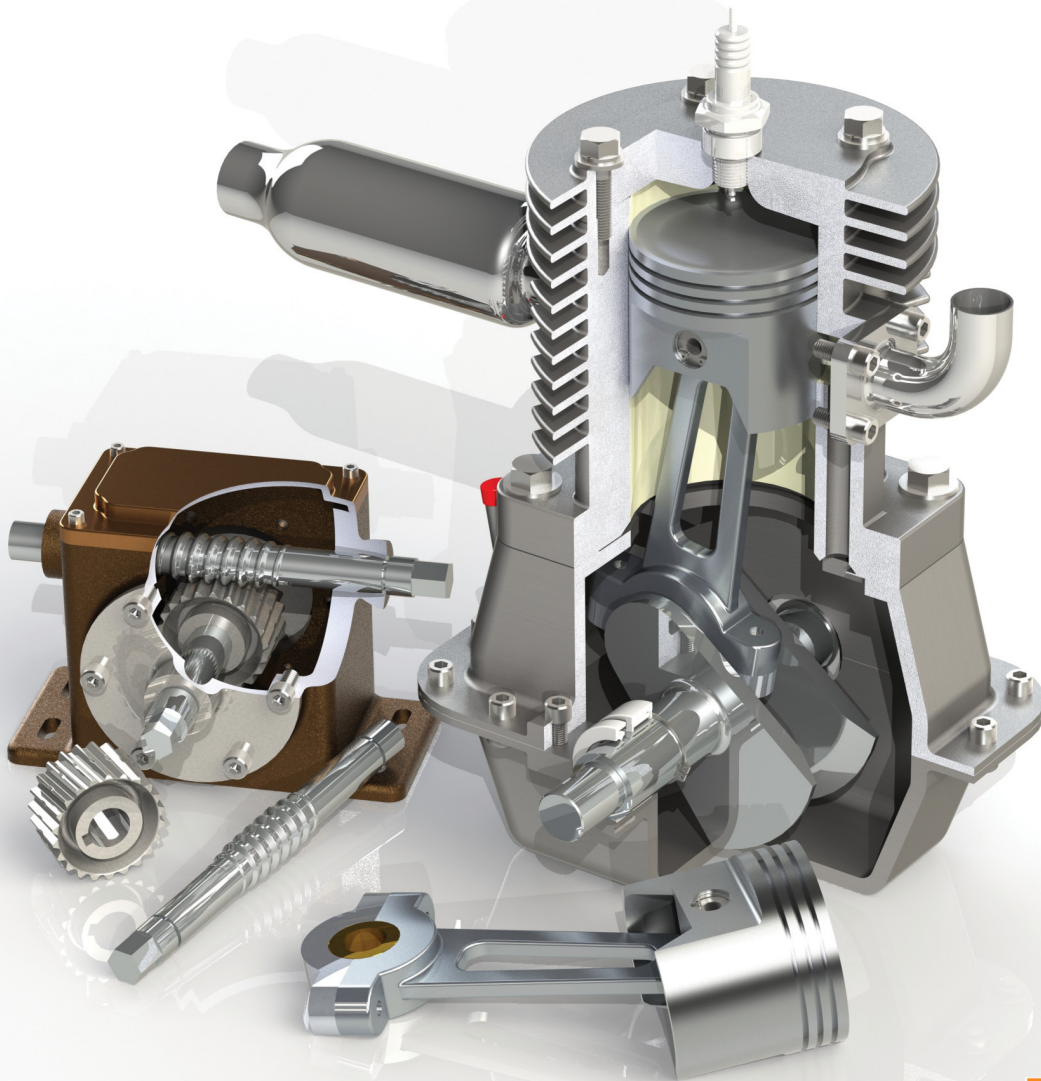


# Beginner's Guide to SolidWorks® 2013 - Level I

Parts, Assemblies, Drawings, Simulation Xpress



Alejandro Reyes MSME, CSWP, CSWI



Visit the following websites to learn more about this book:



[amazon.com](https://www.amazon.com)

[BARNES & NOBLE](https://www.barnesandnoble.com)

[Google books](https://books.google.com)

## ***Special Features: Sweep, Loft and Wrap***

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. 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 in the market can be directly attributed to their appearance. They have to look nice, 'feel' right, and of course perform the task that they were intended 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 process.

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 on the specifics of the new features.

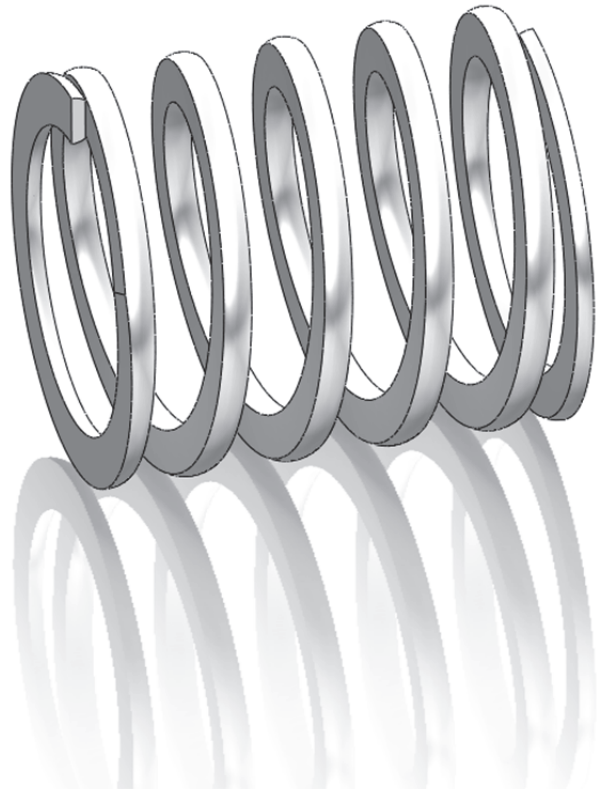
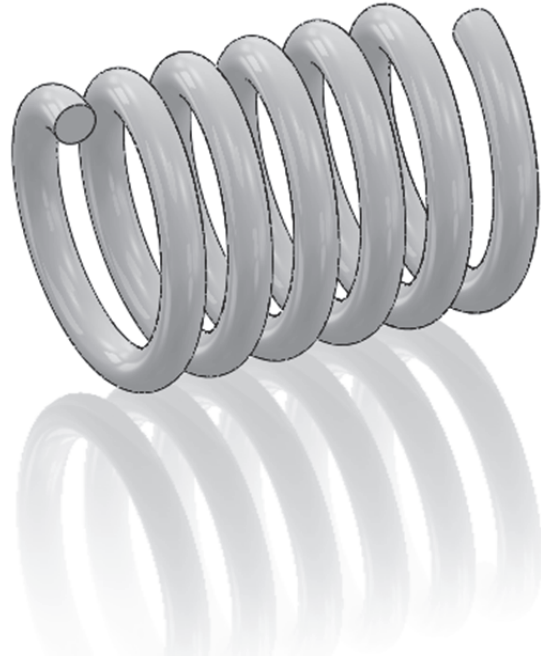
The Wrap feature is a special tool that helps us, as the name implies, to 'wrap' a sketch around a surface, this tool helps us create features like cylindrical cams, slots on cylinders or cylindrical surfaces, etc.

These are some examples of designs made with advanced modeling techniques.




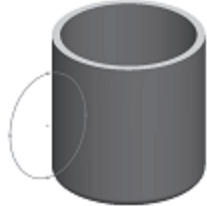
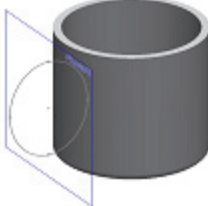
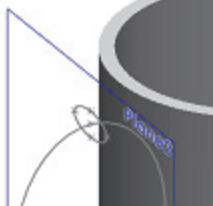






## ***Sweep: Cup and Springs***

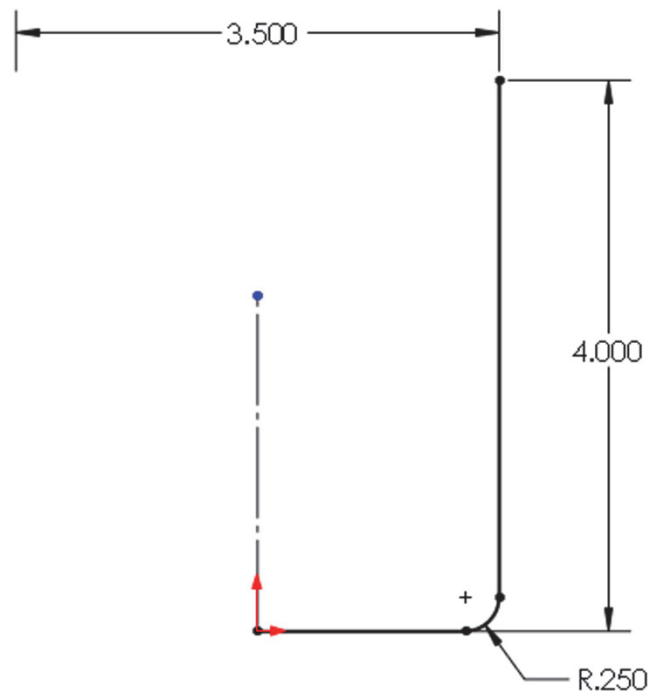


**Notes:**

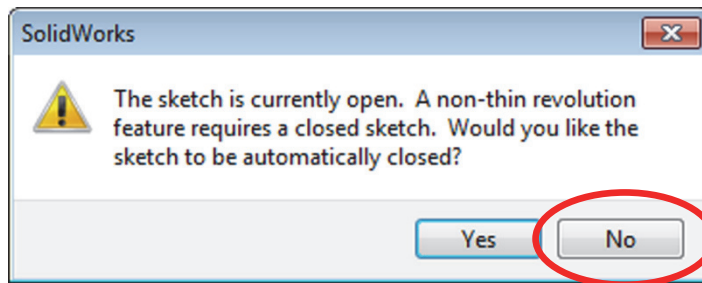
For this exercise we are going to make a simple cup. In this exercise we will learn a new option when creating features called “**Thin Feature**”, the Sweep feature, a new Fillet option to create a Full Round fillet, and a review of auxiliary Planes. The sequence of features to complete the cup is:

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

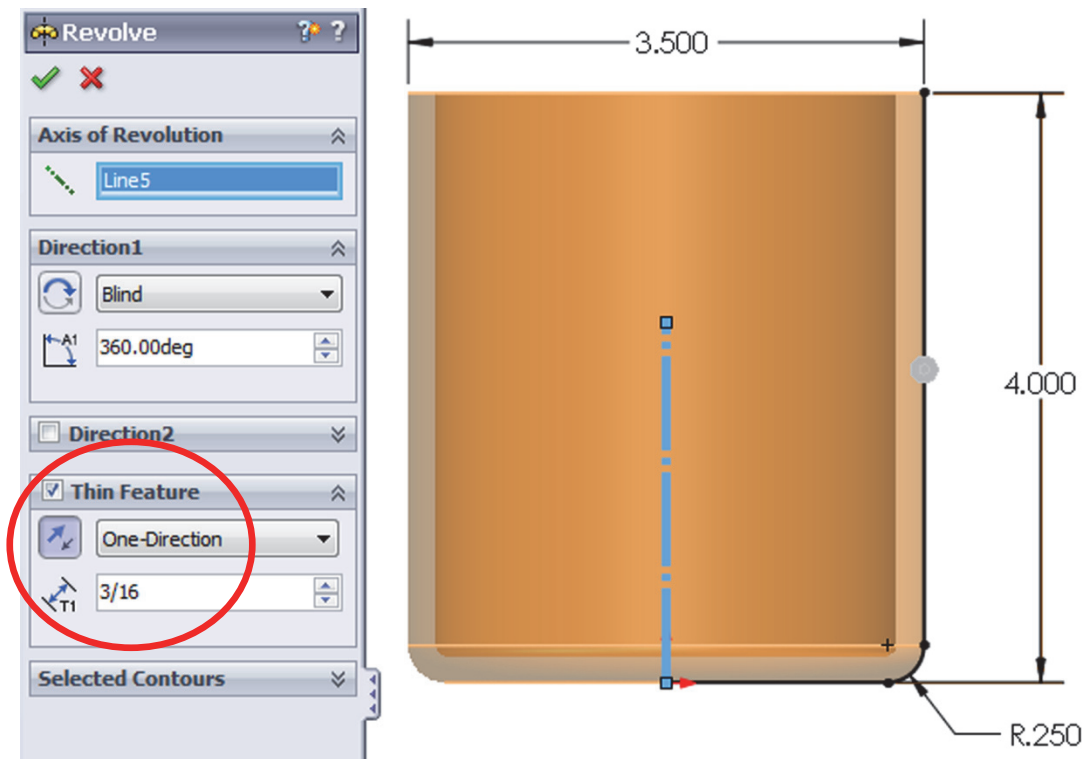
**149.** – For the first feature we will create a “**Revolved Feature**” using the “Thin Feature” option. This option makes a feature with a specified thickness based on the sketch that was drawn. Select the “*Front Plane*” and create the following sketch. Notice the sketch is an open profile with two 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.”

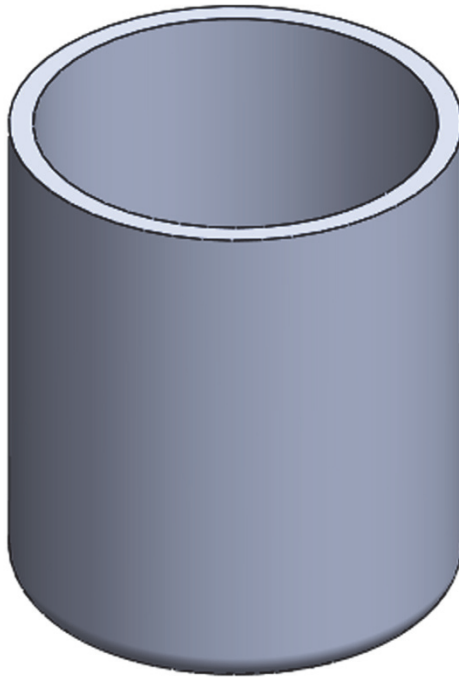


In the “Revolve” options, “**Thin Feature**” is automatically activated. Since we want the dimensions we added to be external dimensions, select the “**Reverse Direction**” option to add the material inside the cup. Notice the preview showing the change.

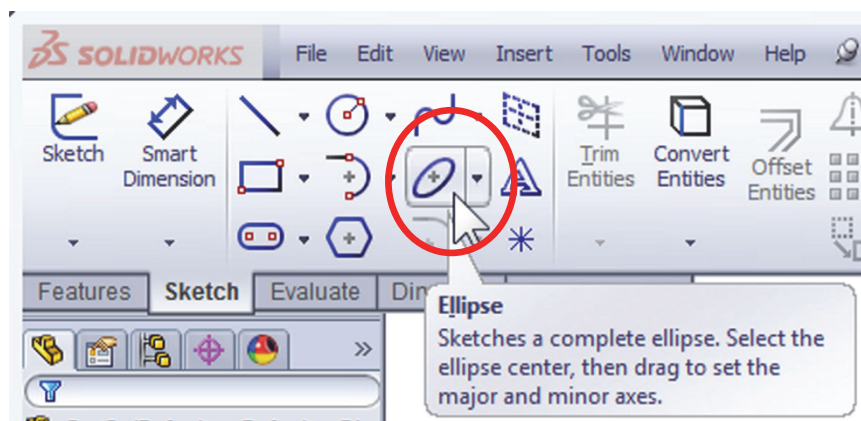


In the value box we typed 3/16; we can add a fraction and SolidWorks changes it to the corresponding decimal value when we click OK. We can also type simple mathematic expressions including addition, subtraction, multiplication and division in any value box where we can type a value.

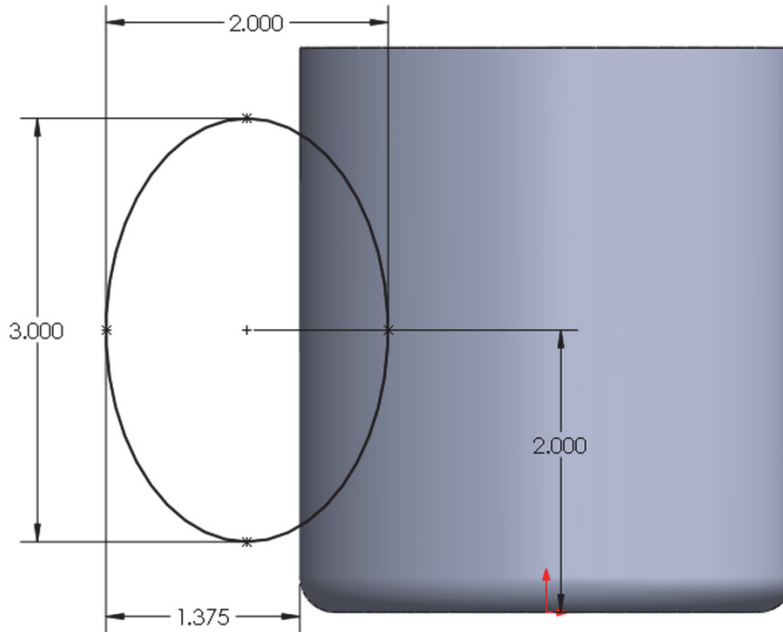
Our thin feature now looks like this:



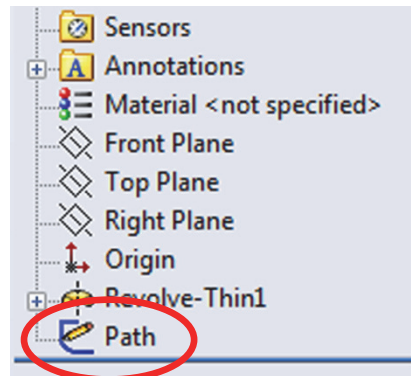
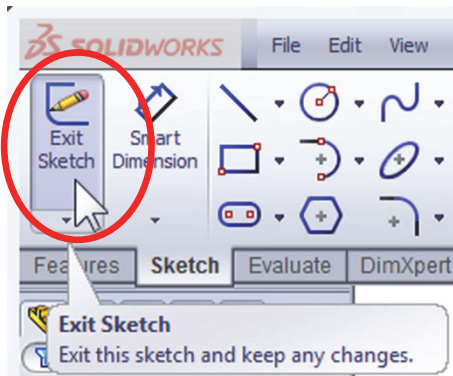
**151.** - Select the "*Front Plane*" and create the following sketch using an ellipse. Be sure to add a "Vertical" geometric relation between the top and bottom points of the ellipse to fully define it. Select the "**Ellipse**" command from the Sketch tab in the CommandManager or from the menu "**Tools,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



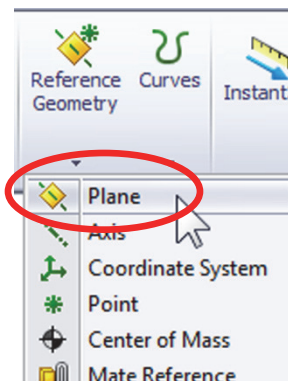


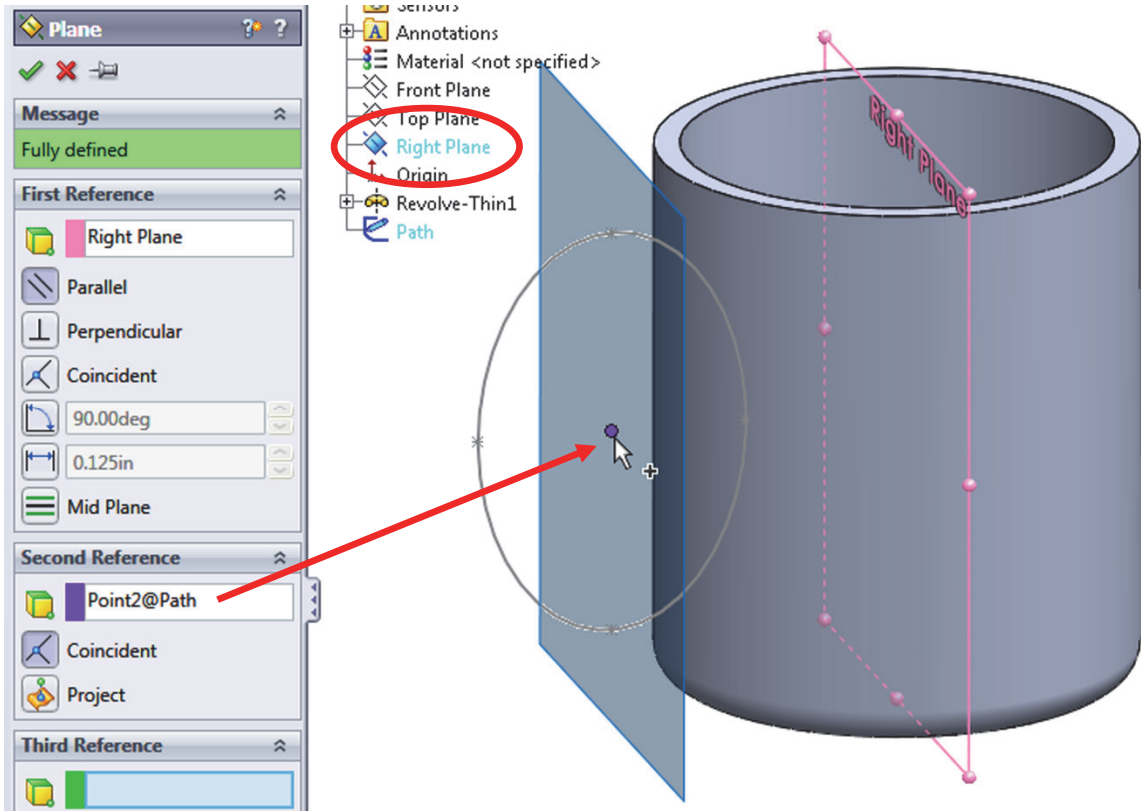


Exit the sketch and rename it "Path." We will not use the sketch for a feature just yet.

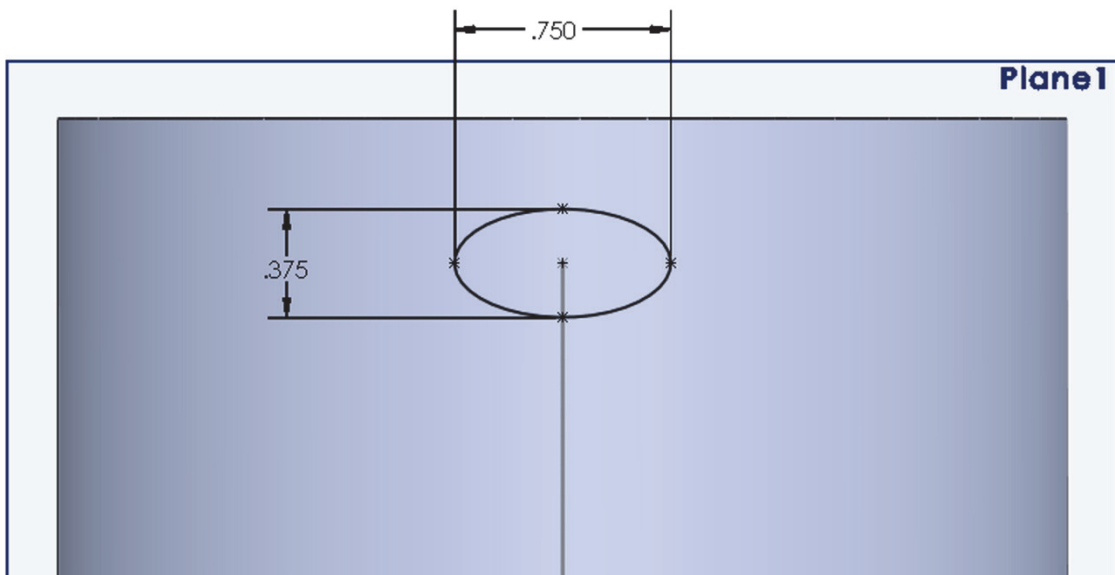


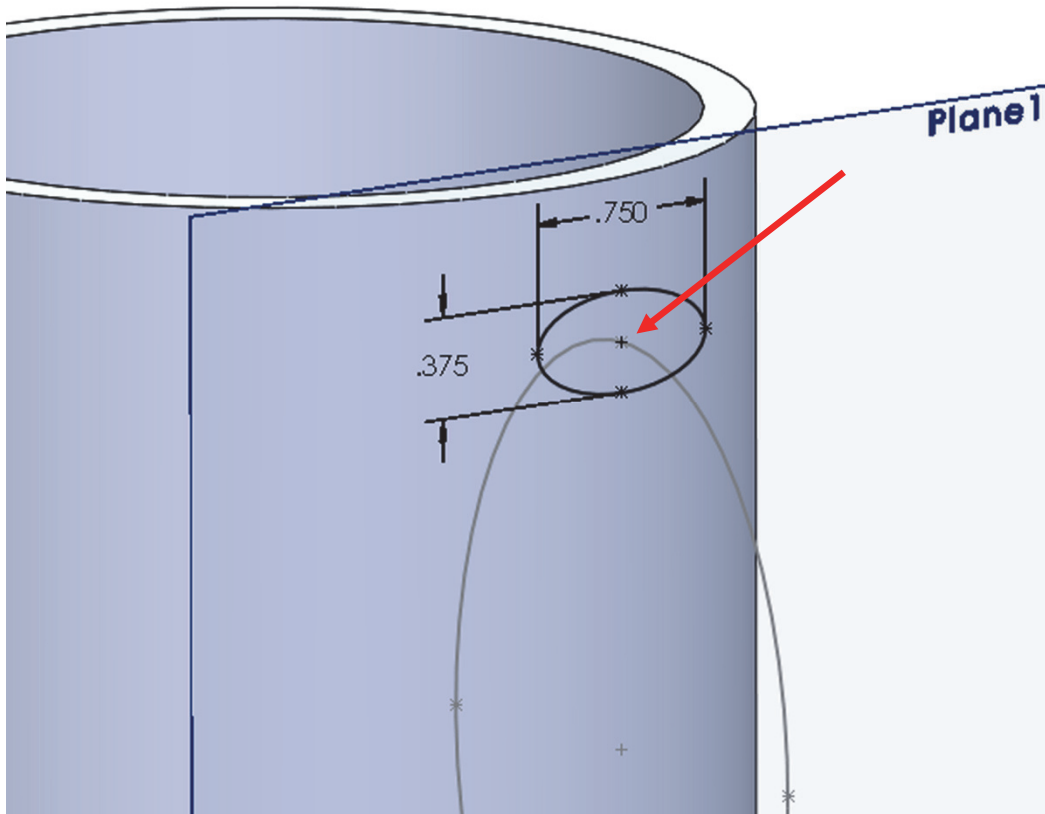
152. - Create an Auxiliary plane parallel to the "Right Plane" using the center of the ellipse as a second reference as shown.





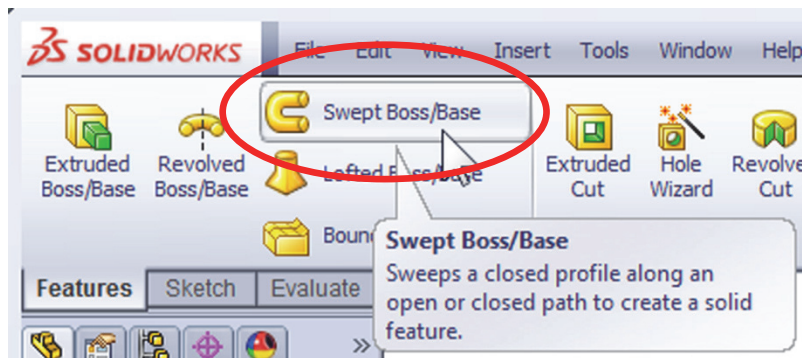
**153.** - Select the plane just created and draw the next sketch. Start the center of the new ellipse at the top point of the previous sketch's ellipse. Add a coincident relation if needed. Remember to add a horizontal relation between the points of the major (or vertical to the minor) axis. Exit the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

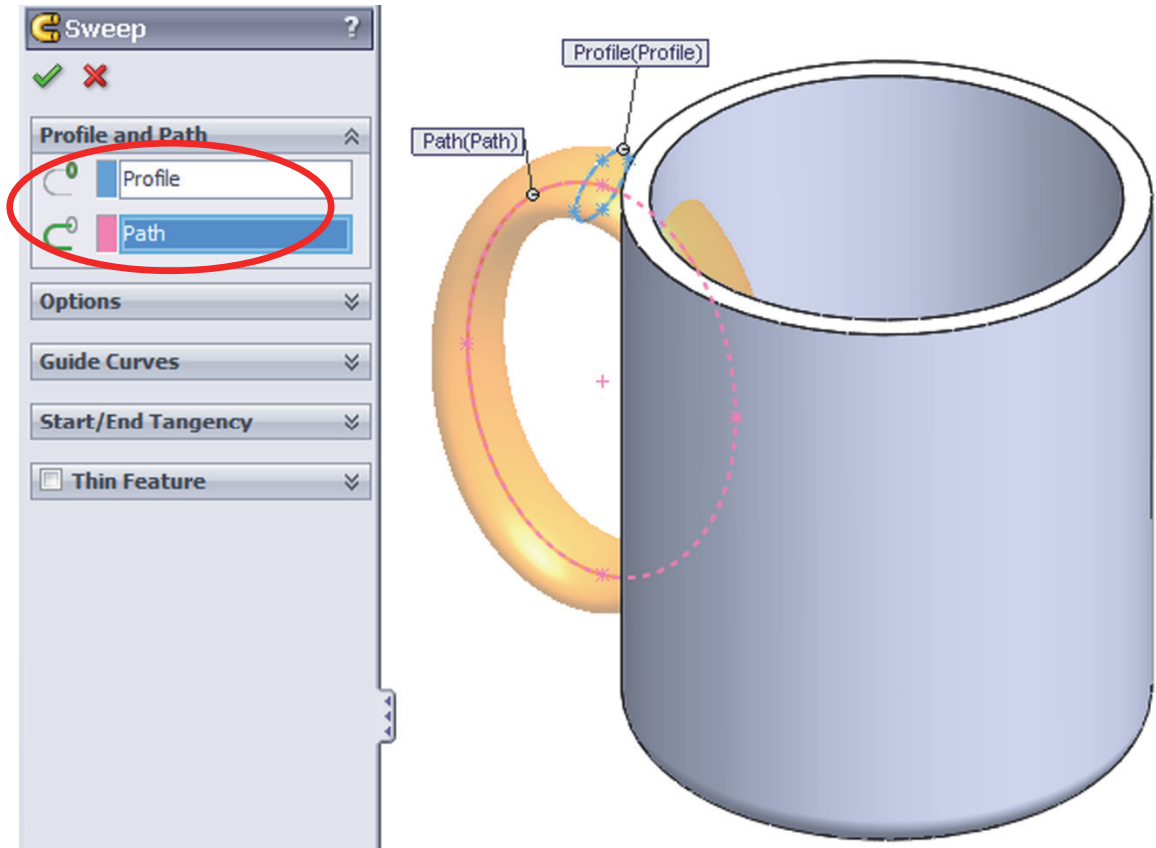




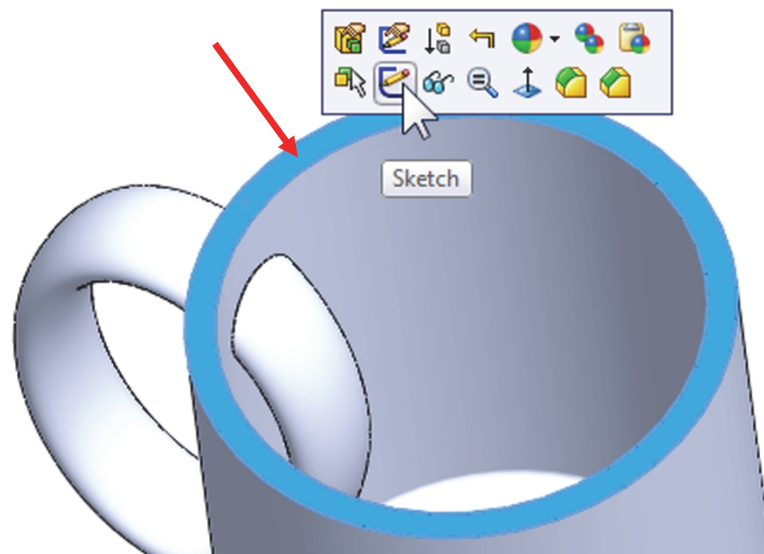
**154.** - Select the “**Sweep**” icon from the Features tab in the CommandManager. The sweep is a feature that requires a minimum of two sketches: one for the sweep profile and one for the path (the path can also be a model edge or a user defined curve.)

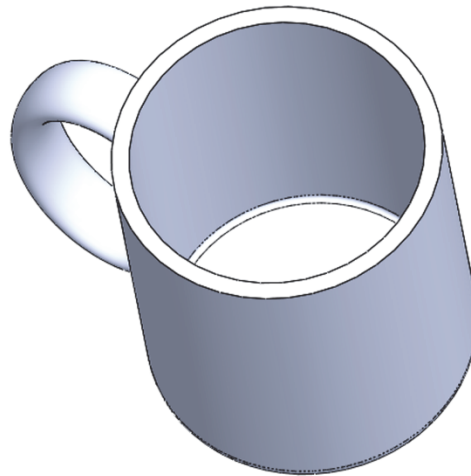
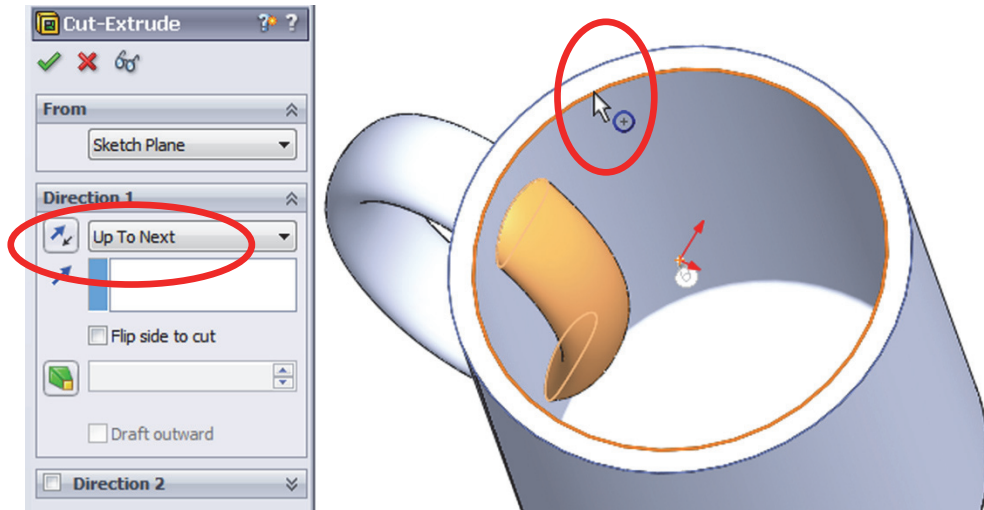
In the “**Sweep**” properties, select the “*Profile*” sketch in the Profile selection box, and the “*Path*” sketch in the Path selection box. Optionally, a Sweep can have guide curves and other parameters to better control the resulting shape, but in this case we are using only the basic options. Notice the preview and click OK when done. Hide the auxiliary planes using the “**Hide/Show Items**” toolbar if so desired.



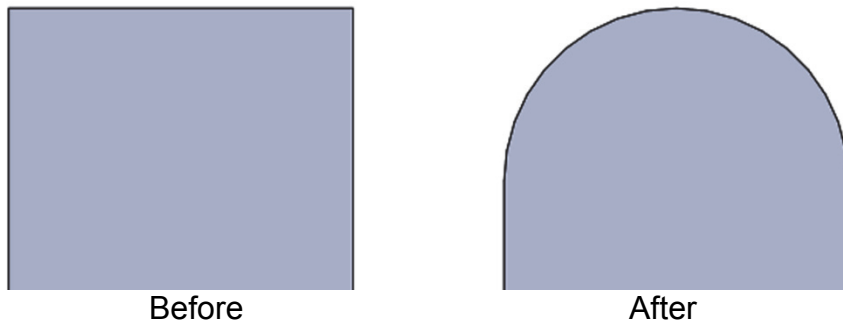


**155.** - Notice the sweep is also inside the cup. We will make a cut to fix this. Create a sketch in the flat face at the top of the cup. Select the inside edge at the top and use “**Convert Entities**” from the Sketch tab to convert the edge to sketch geometry. Select the “**Extruded Cut**” icon and use the “Up to next” end condition from the drop down selection list. This end condition will make the cut until it finds the next face (the bottom) effectively cutting the part of the handle inside the cup.





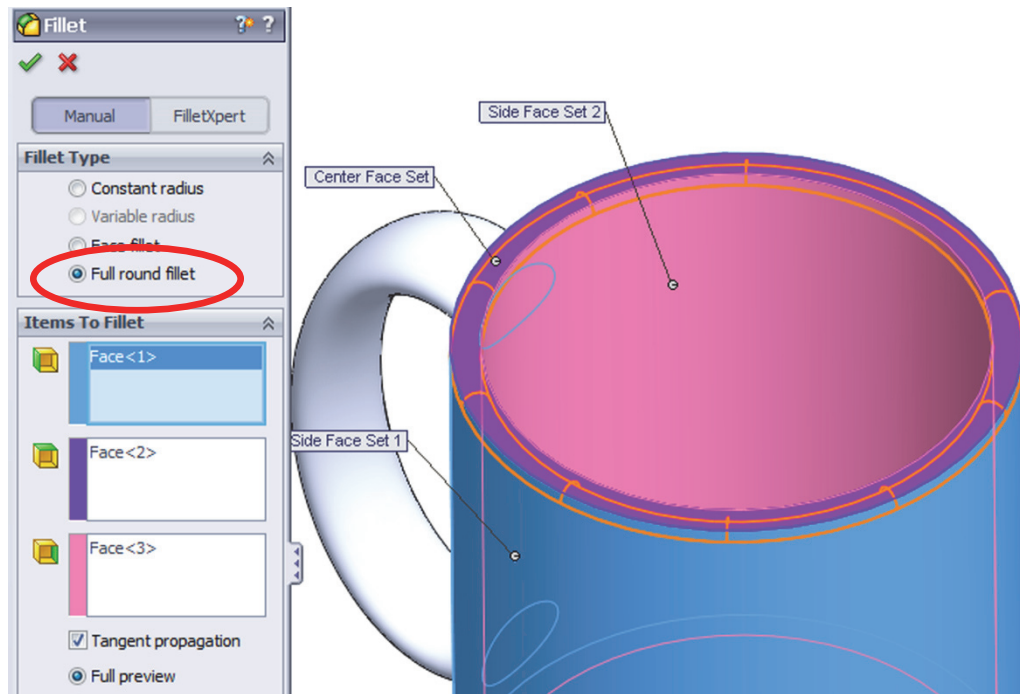
156. - Now we need to round the flat face at the top lip of the cup. To do this we'll add a **Fillet** using the **Full Round Fillet** option. The full round will essentially remove the flat face at the top, and replace it with a rounded face.



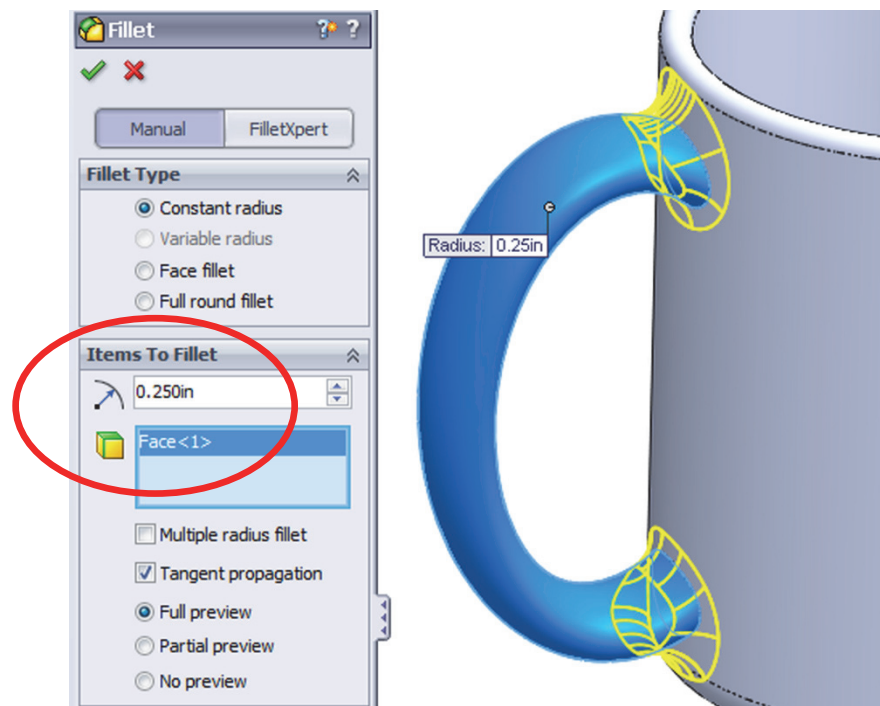
We need to select three faces, the middle face will be the one replaced by the fillet. After selecting the Fillet command, select the "Full round fillet" option to reveal the selection boxes. With the first selection box active select the outside face of the cup. Click inside the second selection box and select the top flat face



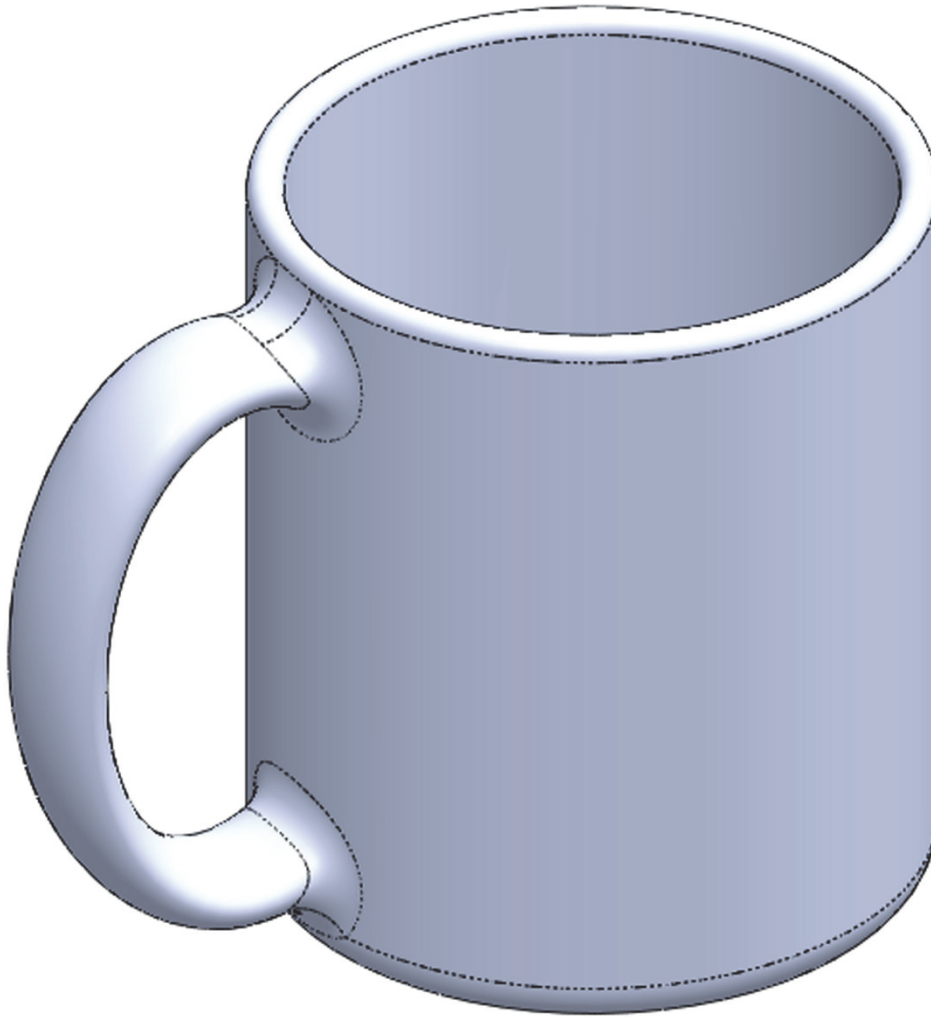
of the cup. Click inside the third selection box and select the inside face of the cup. Note the selected faces are color coded. When done selecting faces click OK to apply the fillet.



**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.

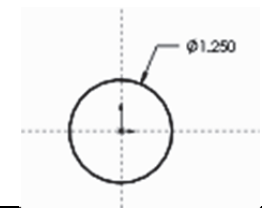
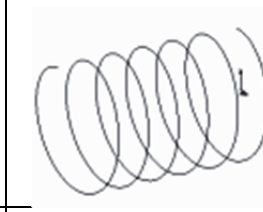
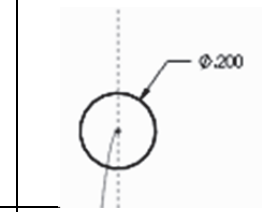
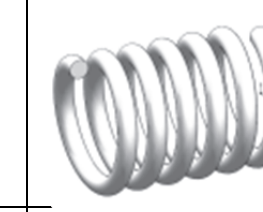


**158.** - Save the finished part as '*Cup*' and close.

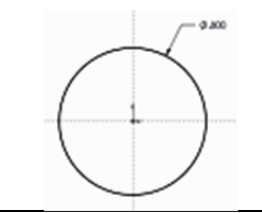
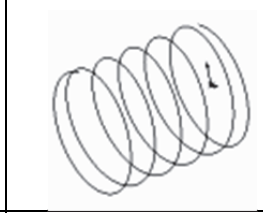
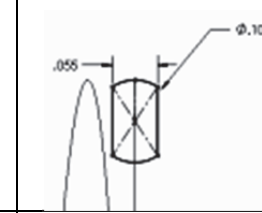
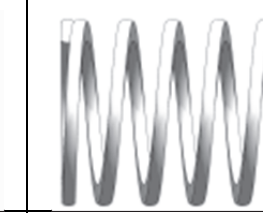


In the next exercise we are going to show how to make a simple and a variable pitch spring. In order to make these springs we'll have to learn how to make simple and variable pitch helixes to be used for the sweep paths. The sequence of features to complete the springs are:

### Simple Spring

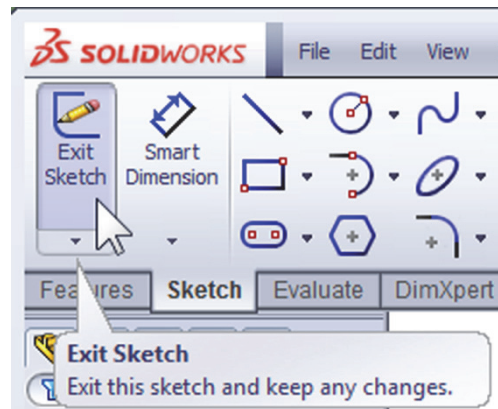
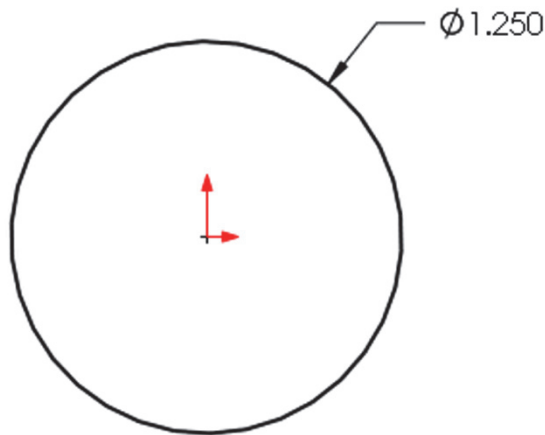
			
Draw circle	Make Helix	Draw Spring Profile Sketch	Make Sweep

### Variable Pitch Spring

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

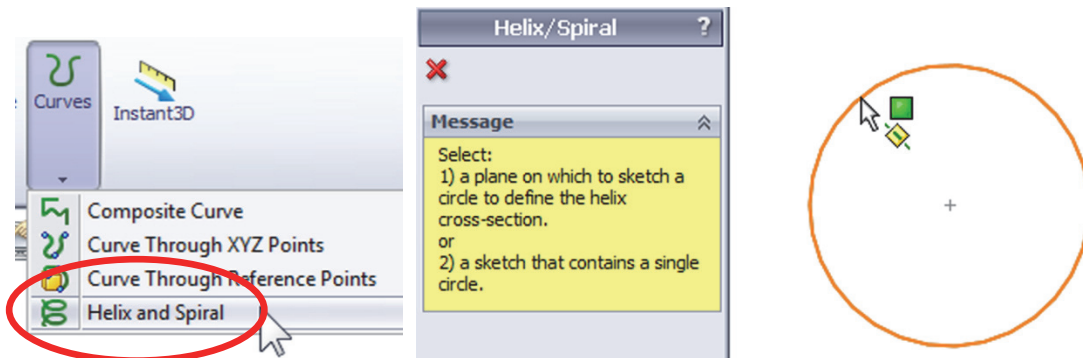
### Simple Spring


**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.

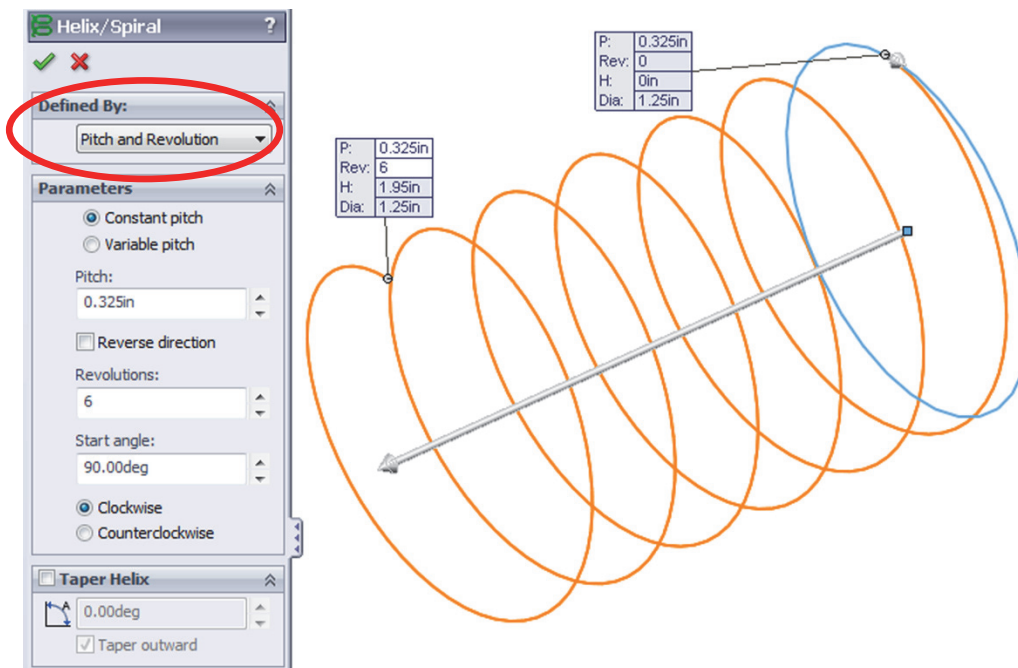
**160.** - In the Features tab select “**Curves, Helix and Spiral**” from the drop-down icon or the menu “**Insert, Curve, Helix/Spiral.**” If asked to select a plane or a sketch, select the sketch we just drew.




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

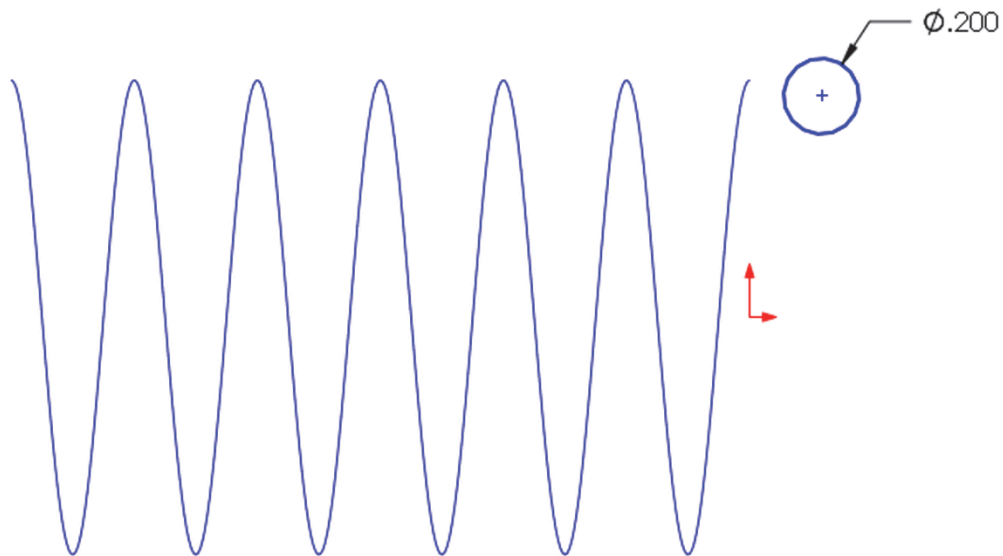
**161.** - The helix can be defined by the combination of any two parameters between Pitch, Revolutions and Height and the third parameter is calculated.

For this example we'll select “Pitch and Revolution” from the “Defined By:” drop down menu, and make the pitch 0.325" and 6 Revolutions. The “Start Angle” value defines where the helix will start. By making it 90 degrees it will start at the top, coincident with the “*Front Plane.*” If we had made it 0 degrees, it would be coincident to the “*Top Plane.*” instead (Feel free to explore the options.)

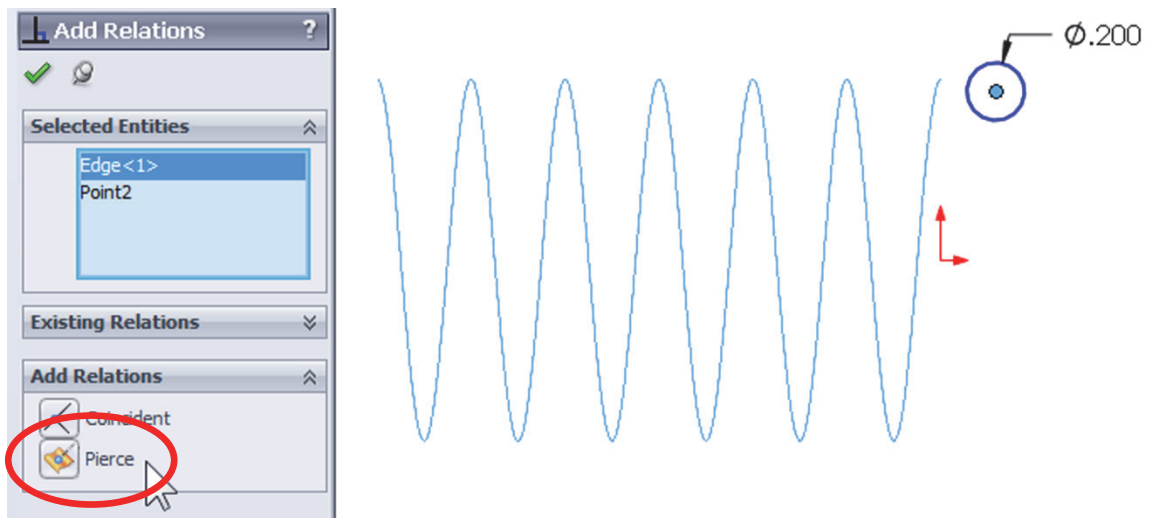


 Note the Helix command has options to make it Counterclockwise, Clockwise, Tapered, Variable pitch and reversed (going right or left.)

**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...



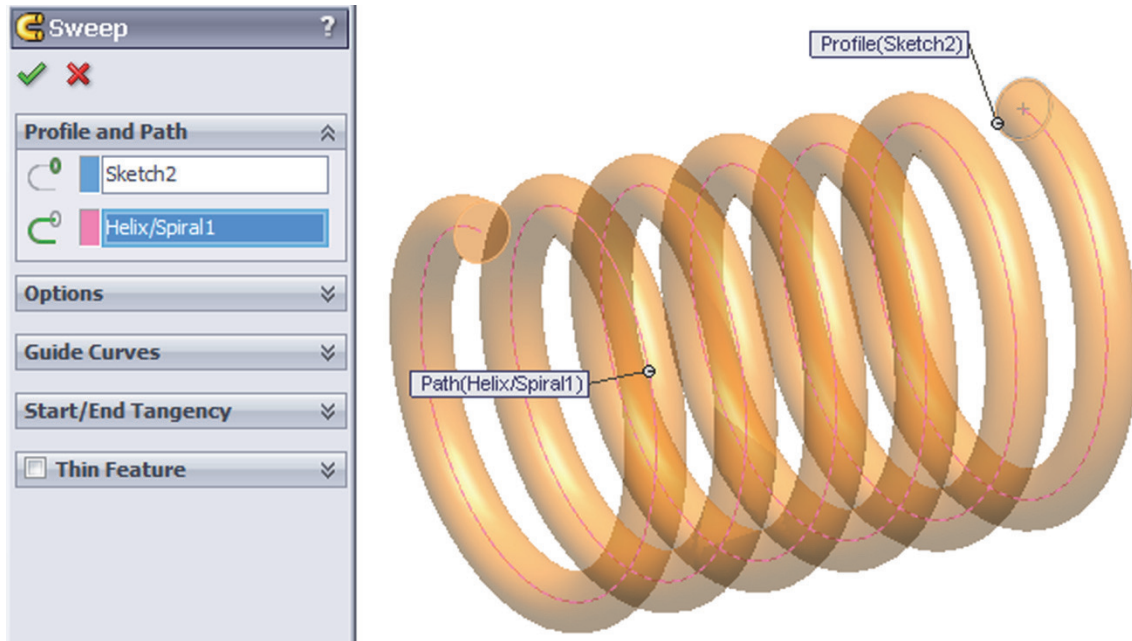
... and add a **"Pierce"** geometric relation between the center of the circle and the helix. This way the path will start at the beginning of the helix. This relation will make the sketch fully defined. Exit the Sketch when done.



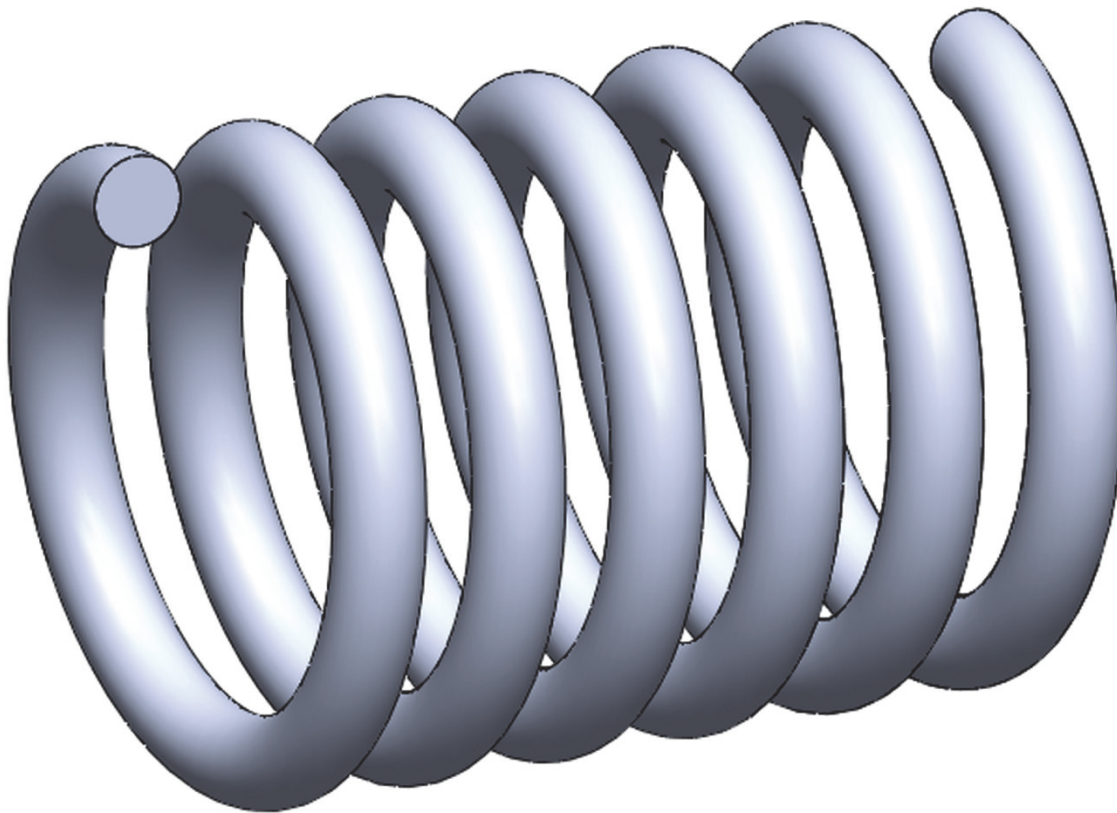
A pierce relation is done between an element that is oblique or perpendicular to the sketch plane and a point in the sketch. Think of it as a needle piercing through a fabric, the sketch being the fabric and the helix (or curve, model edge or another sketch) the needle.



**163.** - Select the Sweep command and make the sweep using the last sketch as a profile and the Helix as a Path. Note the Preview and click OK to finish.

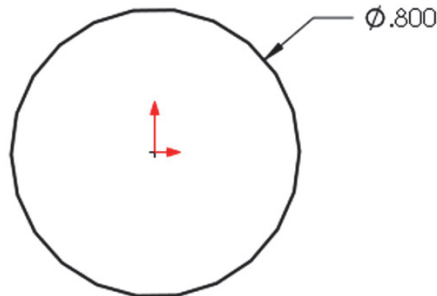


Save as 'Spring' and close the finished spring.

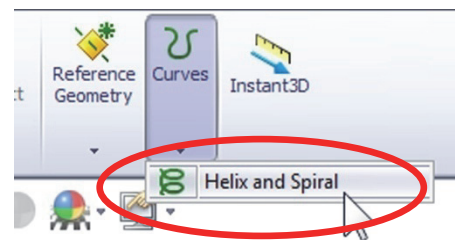


## Variable Pitch Spring

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, in the Features tab from the "Curves" drop down icon select "Helix and Spiral"; 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, fill in the values for the helix with the following values. Click OK when finished to build the helix.

**Helix/Spiral**

Defined By: Pitch and Revolution

Parameters:  Variable pitch

	P	Rev	H	Dia
1	0.01in	0	0in	0.8in
2	0.01in	0.5	0.005	0.8in
3	0.2in	1	0.057	0.8in
4	0.2in	5	0.857	0.8in
5	0.01in	5.5	0.91in	0.8in
6	0.01in	6	0.915	0.8in
7				0.8in

Reverse direction:

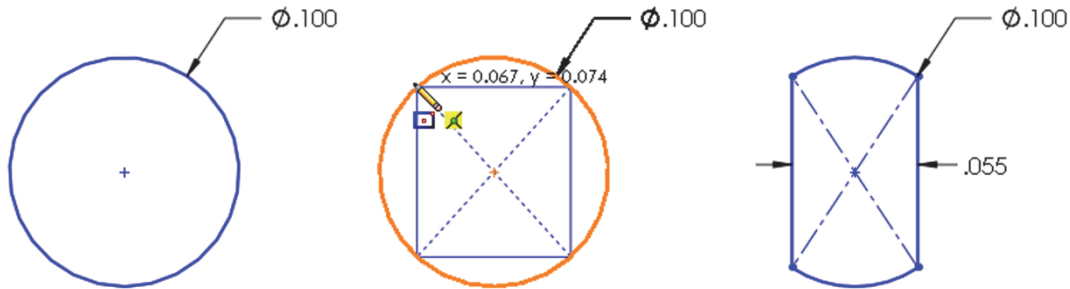
Start angle: 90.00deg

Clockwise  
 Counterclockwise

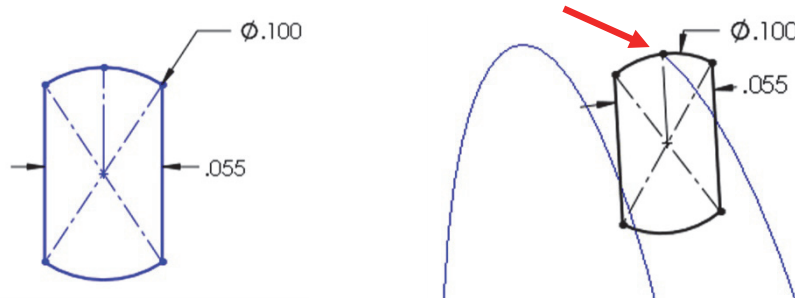
3D Model Parameters:

- Region 1: P: 0.01in, Rev: 6, H: 0.915in, Dia: 0.8in
- Region 2: P: 0.2in, Rev: 5, H: 0.8575in, Dia: 0.8in
- Region 3: P: 0.2in, Rev: 1, H: 0.0575in, Dia: 0.8in
- Region 4: P: 0.01in, Rev: 0, H: 0in, Dia: 0.8in
- Region 5: P: 0.01in, Rev: 5.5, H: 0.91in, Dia: 0.8in
- Region 6: P: 0.01in, Rev: 0.5, H: 0.005in, Dia: 0.8in
- Region 7: P: 0.01in, Rev: 6, H: 0.915in, Dia: 0.8in

**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.

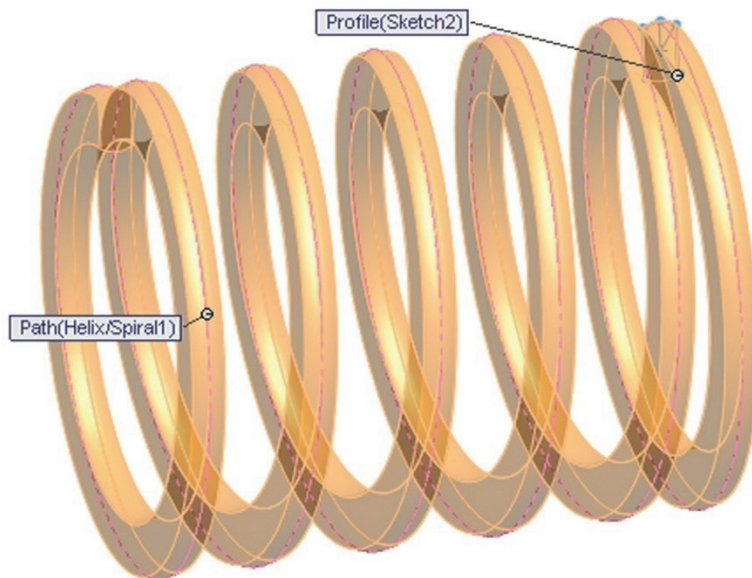


**168.** - Add a centerline from the center to the top line (make sure it is coincident.) Select the endpoint of the centerline and the helix to add a "Pierce" geometric relation to make the sketch fully defined. Exit the sketch and optionally rename it 'Profile'.

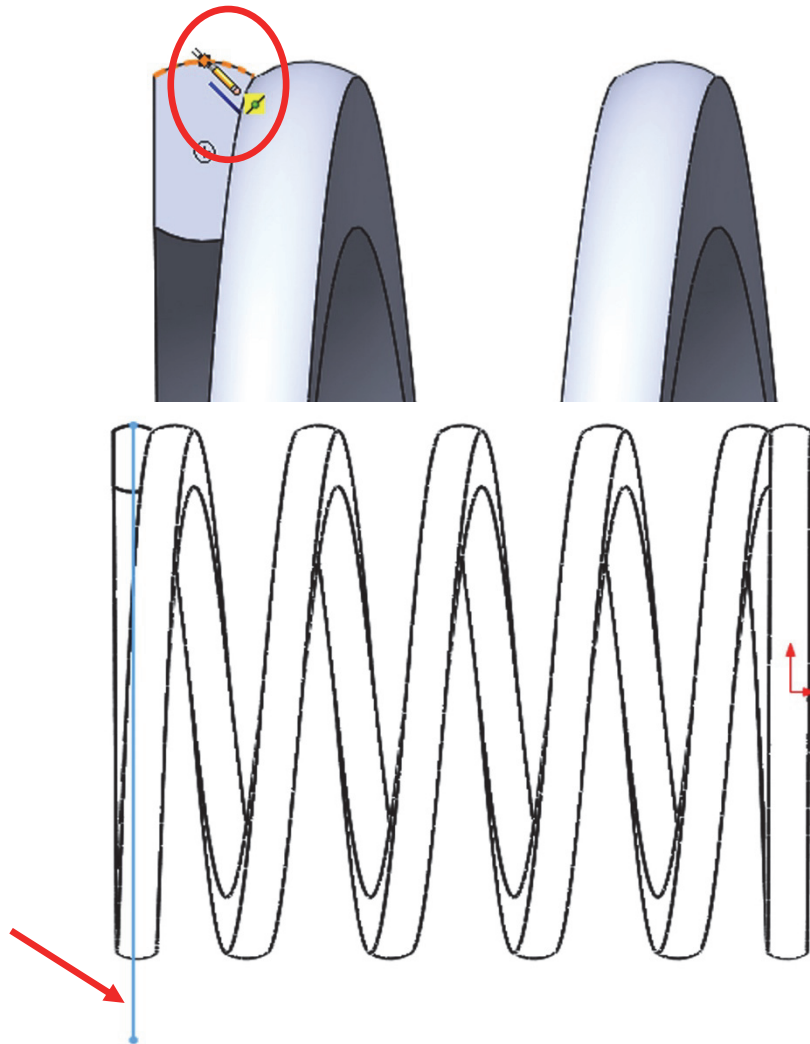


The Pierce relation allows us to fix the profile to the path, and can be added to any part of the sketch. We chose to add it to the top because the original sketch used for the helix is the spring's outside diameter.

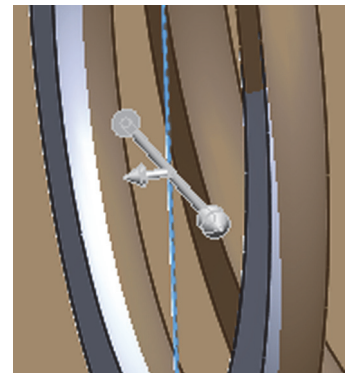
**169.** - Just as we did before, select the "Sweep" command, add the path and profile as shown and click OK to finish.



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

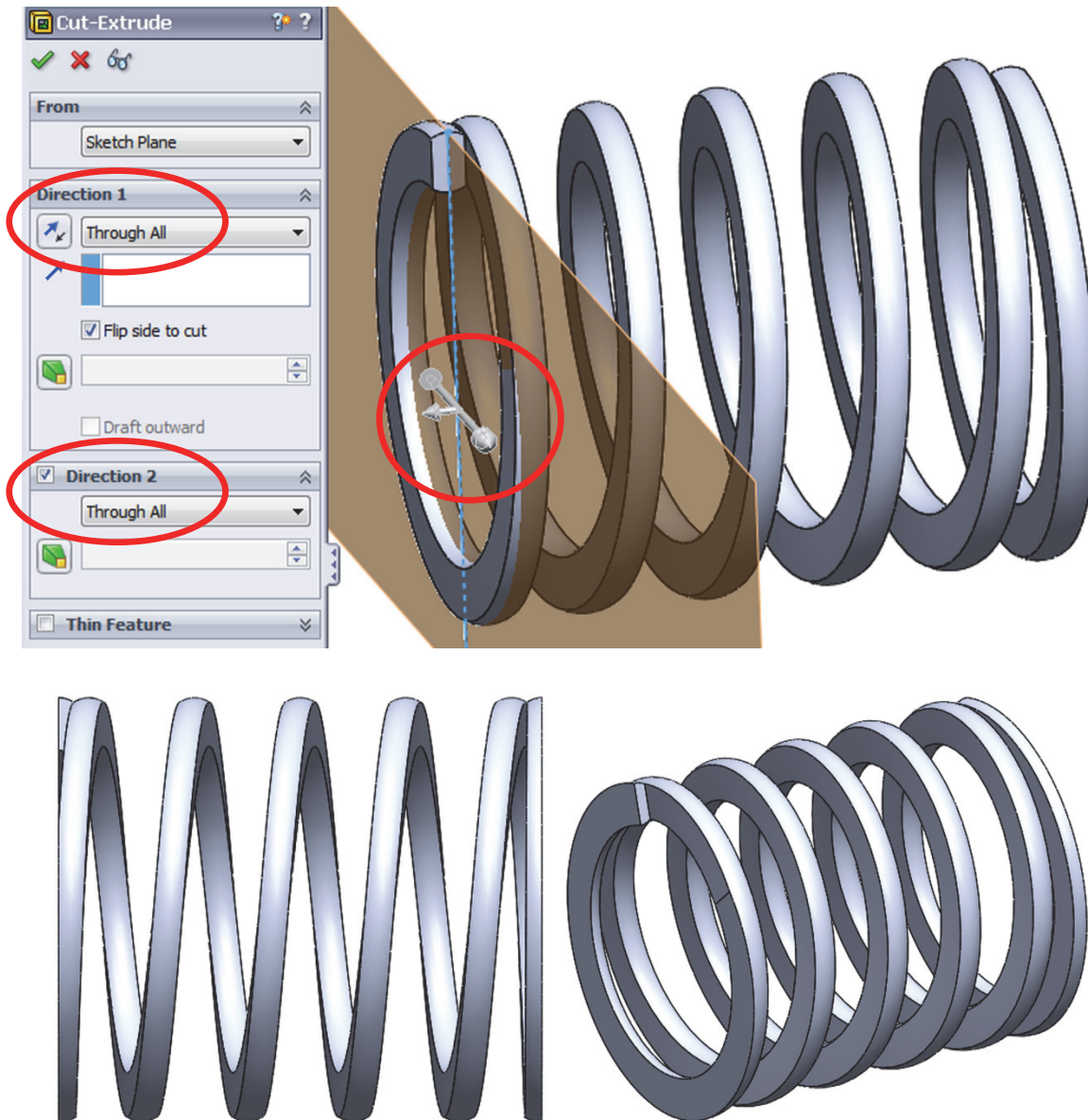


**171.** - Select the "**Extruded Cut**" command. When we use an open sketch to make a cut, the "Through All" option is automatically selected in both directions. One side of the model is cut using the open sketch. The small arrow located at the center of the line indicates which side of the model will be cut. 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. Note the cutting plane is shown in the graphics area.



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





Save the model as '*Variable Pitch Spring*'.

### A note about threads



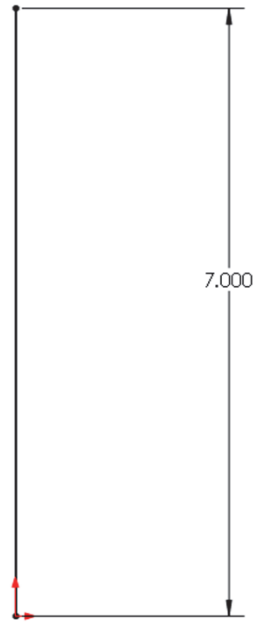
Threads can be added to a model either by adding a Sweep or a Cut Sweep with a helix. When modeling screws and fasteners in general, it is almost always unnecessary to add a helical thread, as it consumes a large amount of computing resources, and a simple representation of it usually suffices (as a Revolved Boss/Cut or Cosmetic Thread.) It is strongly advised to only add threads when required by the model, as in the following bottle exercise.

For the next exercise we'll build the following bottle using a sweep feature with guide curves.

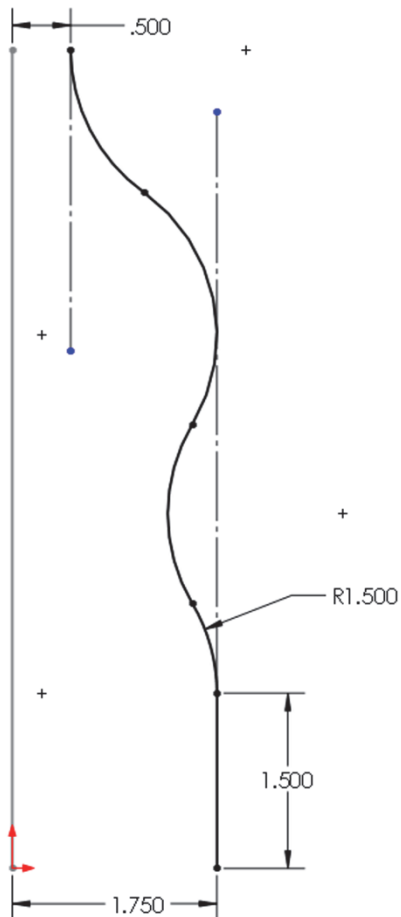


This bottle will be made using a Sweep feature for the body and one for the thread. For the body we'll make a sweep with two guide curves, so we need to make four sketches: Path, Guide Curve 1, Guide Curve 2 and Profile, in that order. The reason to make the Profile last is that it has to pierce the Path and both Guide Curves.

**172.** - Open a new part, set dimensions to inches and three decimal places. Make the Path sketch in the "*Front Plane*" and Exit the sketch. Rename '*Path*'.

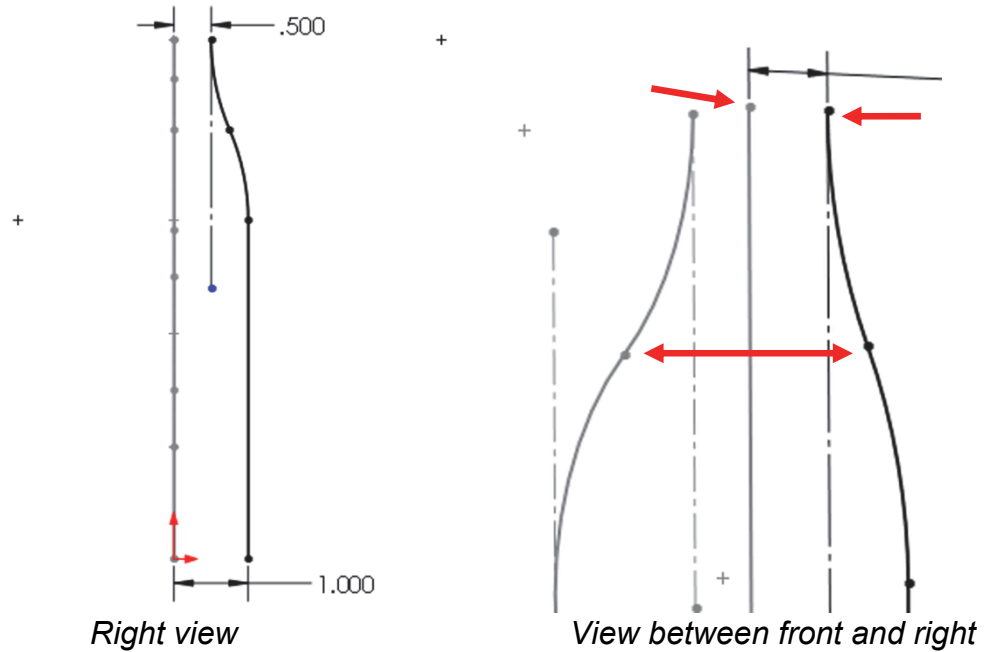


**173.** - Add a second sketch in the "*Front Plane*." Make arcs and lines tangent to each other, add centerlines as reference for tangency. All arcs are equal. Exit sketch and rename '*Guide 1*'.

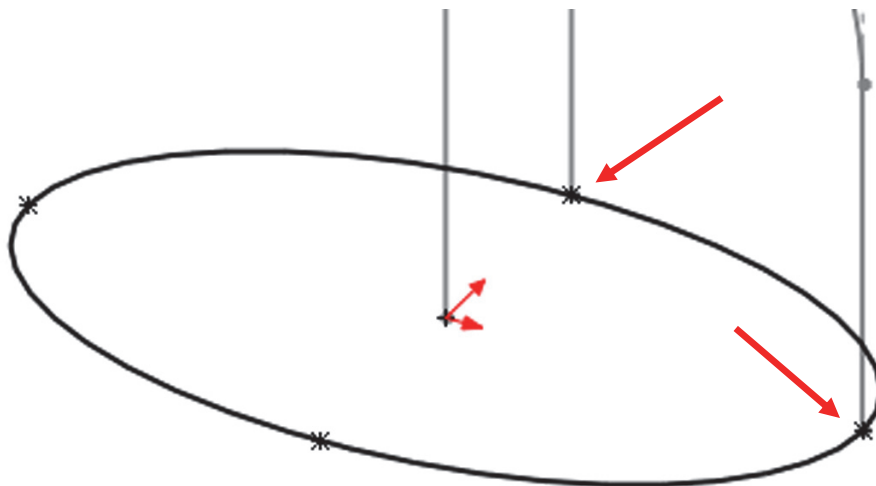




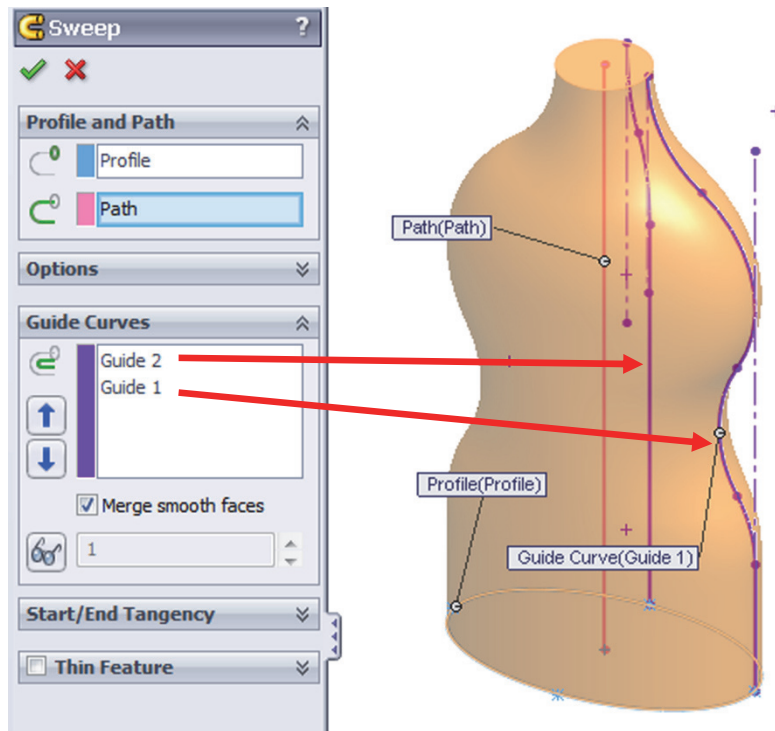
**174.** - Add third sketch to "*Right Plane*." Add geometric relations to previous sketches to maintain design intent. Make the topmost endpoint Horizontal to the topmost endpoint of the '*Path*' sketch. Arcs are equal size and tangent. Make the indicated endpoints horizontal. Only geometric relations and two dimensions are needed to fully define the sketch. Exit the sketch and rename it '*Guide 2*'.



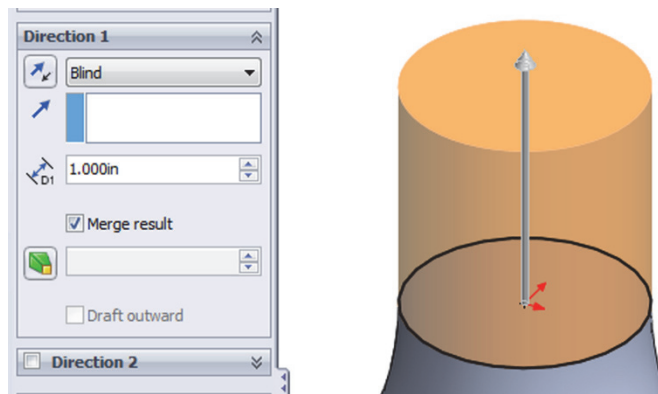
**175.** - Add a new sketch in the "*Top Plane*", make an ellipse adding Pierce relations between the ellipse's major and minor axes to '*Guide 1*' and '*Guide 2*'. Exit the sketch and rename it '*Profile*'.



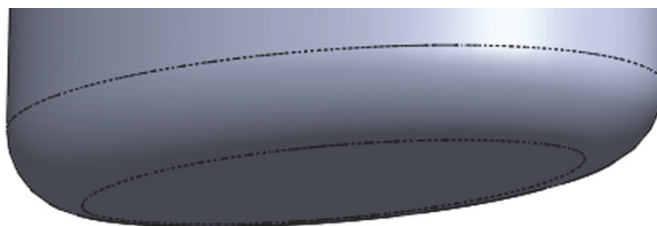
**176.** - Select the "**Sweep Boss/Base**" icon, add the '*Path*' sketch to the "Path" selection box and the '*Profile*' sketch to the "Profile" selection box. Expand the "Guide Curves" selection box and add both guides to it. Click OK to finish.



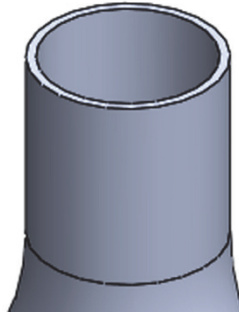
**177.** - Add a sketch on the top face using "Convert Entities" and extrude it 1".



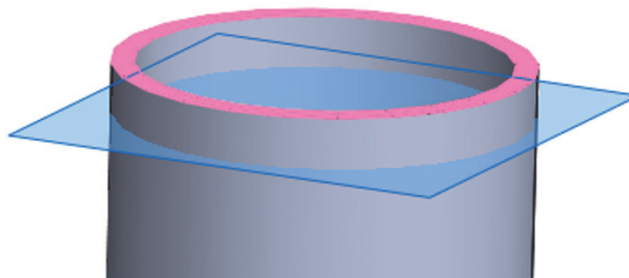
**178.** - Add a 0.375" fillet at the bottom of the bottle.



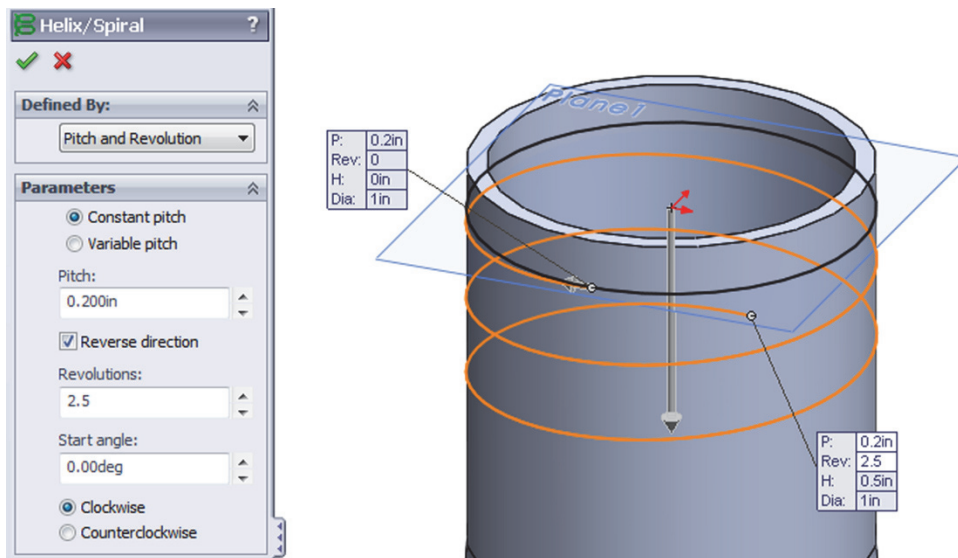
179. - Add a 0.050" shell to the part, remove the top face.



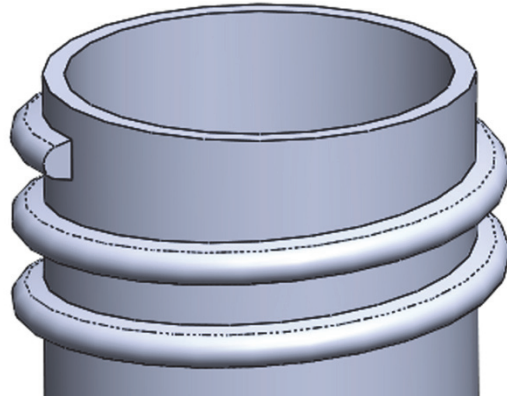
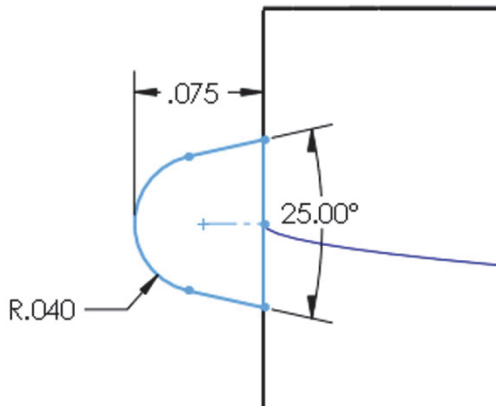
180. - Add auxiliary plane parallel to the topmost face 0.125" below it.



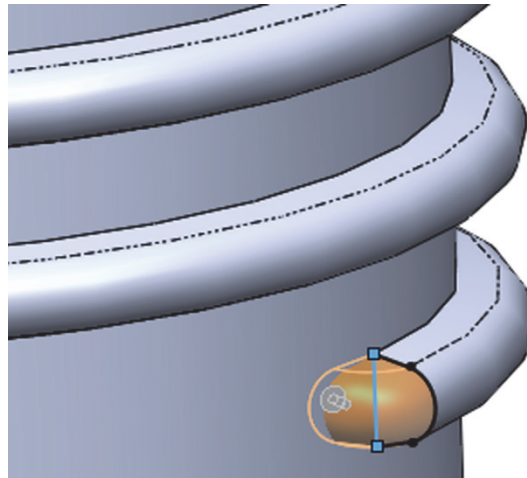
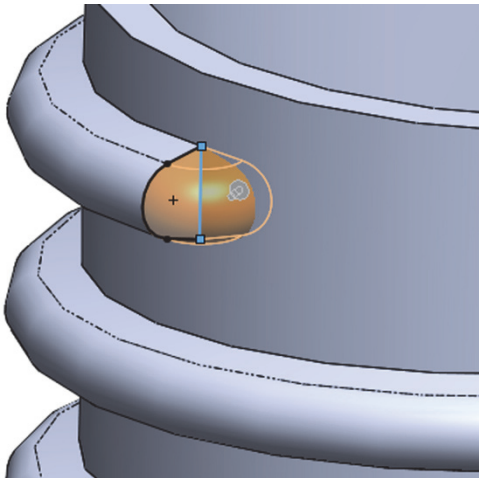
181. - Add sketch to this plane, use Convert Entities to project top outside edge, and make a Helix with a 0.2" pitch, 2.5 turns. Start helix at 0 degrees.



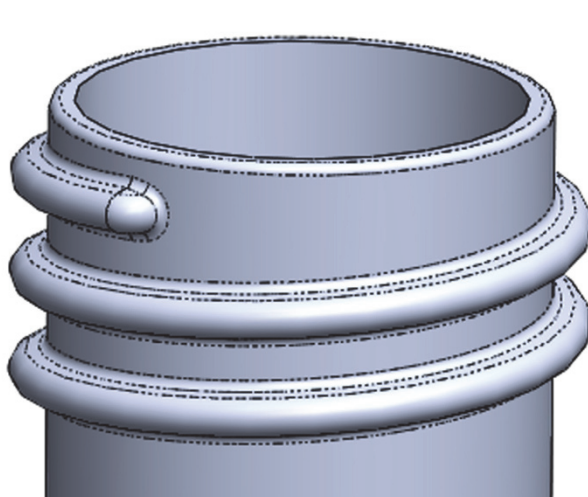
182. - Add the following sketch to the "Right Plane." Be sure to add a Pierce relation between the profile and the helix. Exit the sketch and make a sweep feature using the helix and this profile.



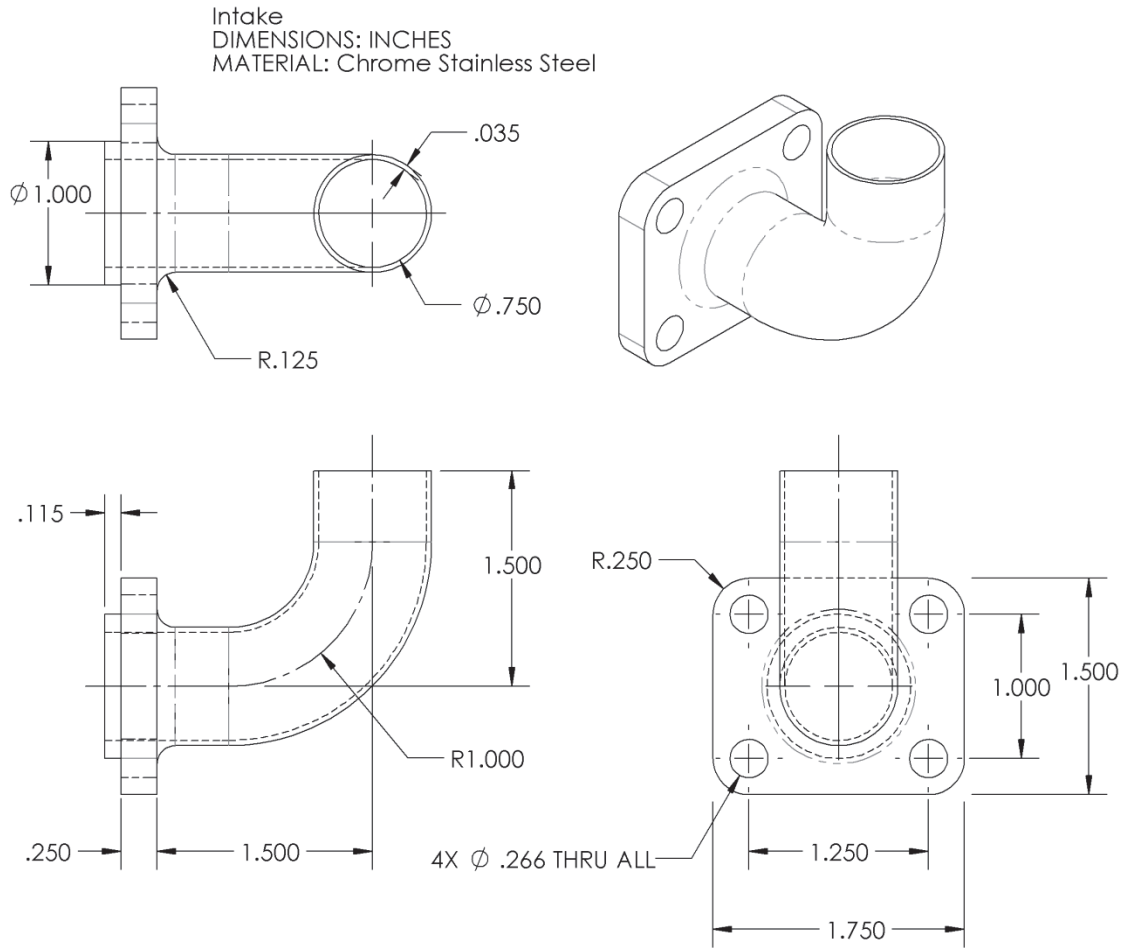
**183.** - Add a sketch on the front face of the thread, use Convert Entities and make a 120 degrees "Revolved Boss." Add a second one at the other end of the thread.

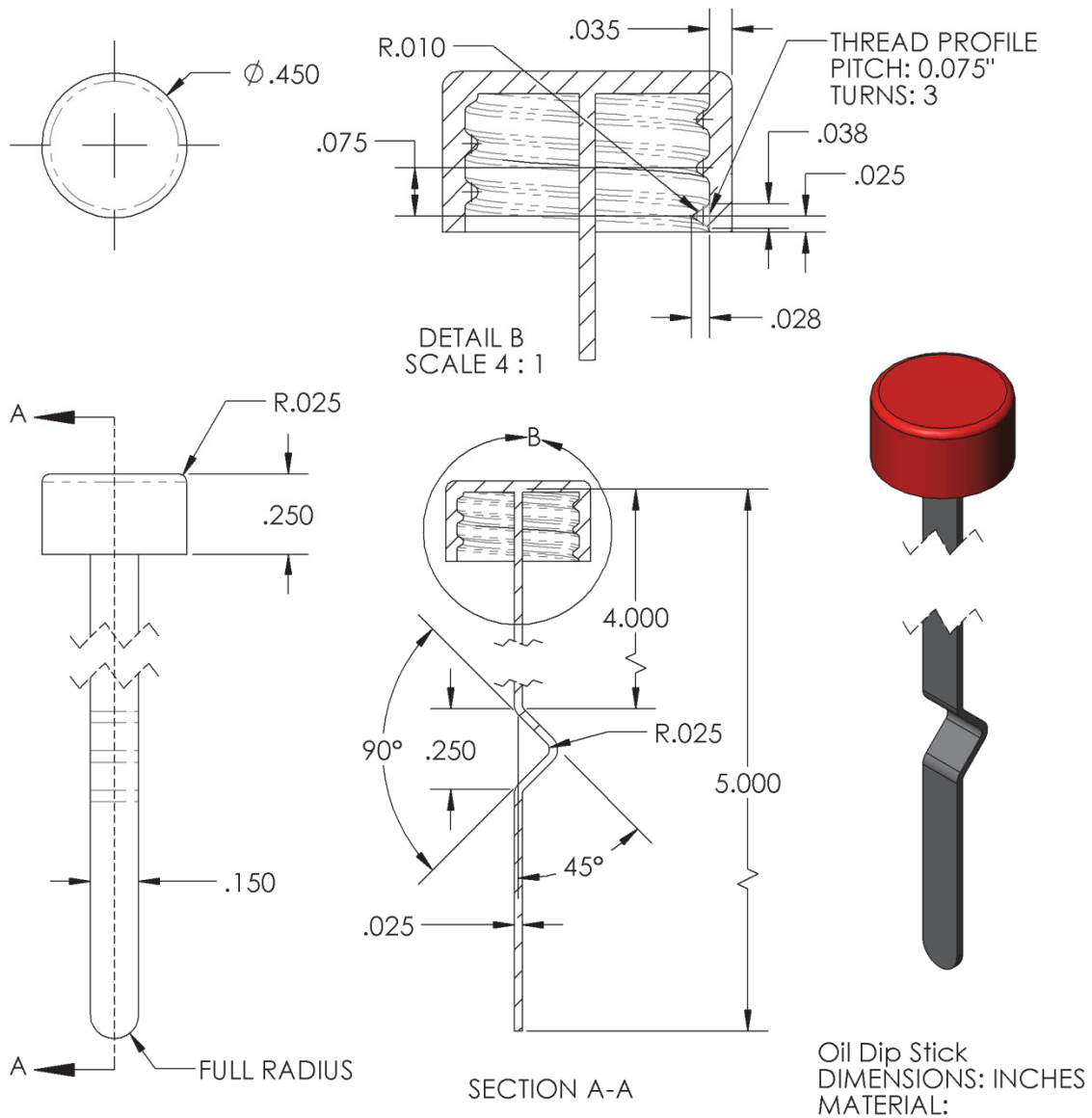


**184.** - Add a 0.02" fillet to the thread as a finishing touch. You may need to select multiple edges.



**Engine Project Parts:** Make the following components to build the engine. Save the parts using the name provided. High resolution images at [mechanicad.com](http://mechanicad.com).





***Loft: Bottle***

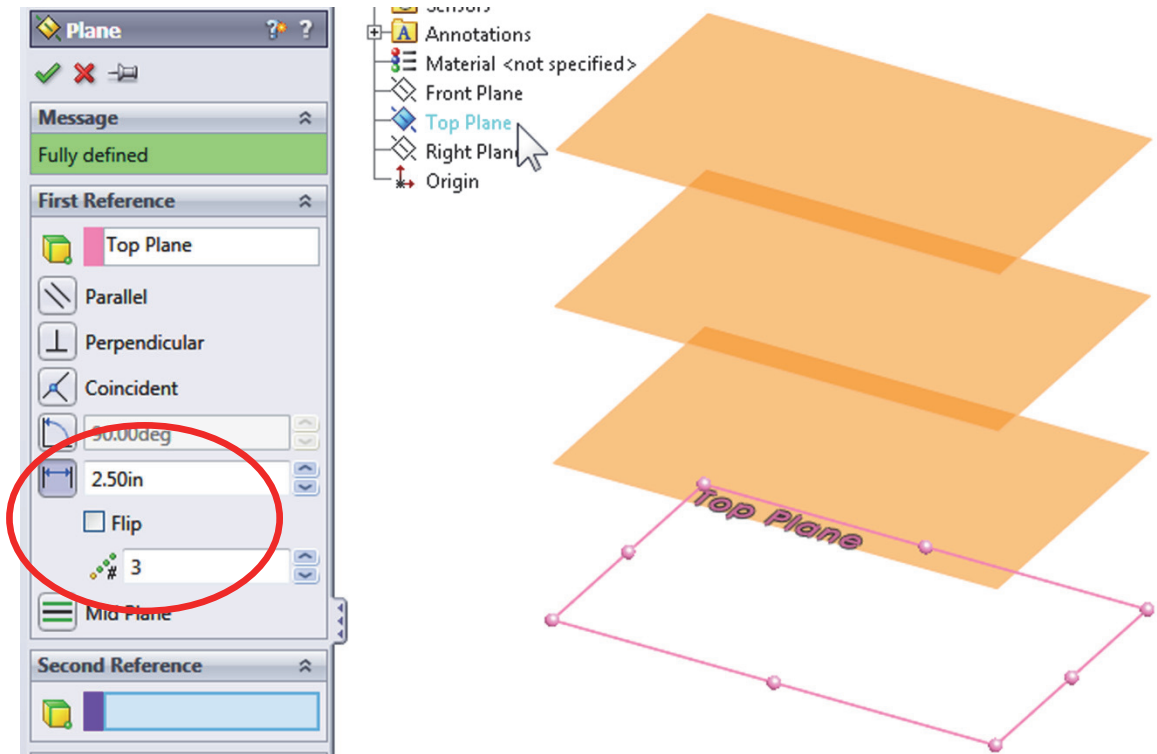




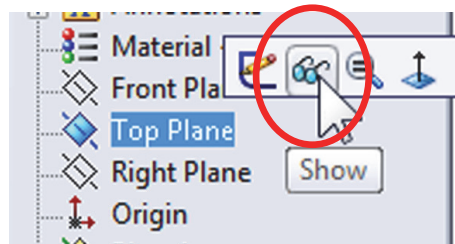
**Notes:**

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 complex shapes with more control over the cross section. In this exercise we will make a bottle using a loft with four different sketches.

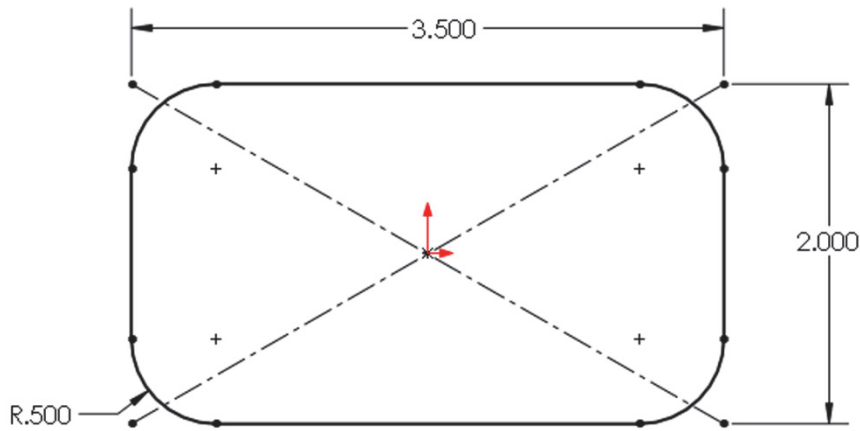
**185.** - Make a new part and create three auxiliary planes using the **“Plane”** command. Select the **“Top Plane”** as reference, change the number of planes to 3 and space them 2.5” as shown. Click OK when done.



**186.** - For clarity, select the **“Top Plane”** in the **FeatureManager** and show it using the **“Hide/Show”** command from the pop-up toolbar.



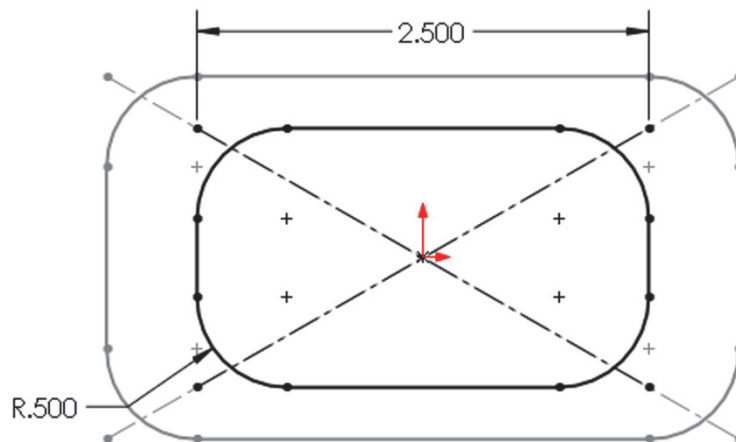
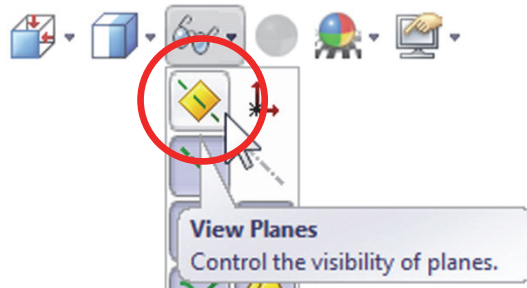
**187.** - Switch to a Top view, select the "Top Plane" and draw the following sketch using the "Center Rectangle" and "Sketch Fillet" tools. Exit the sketch when finished.



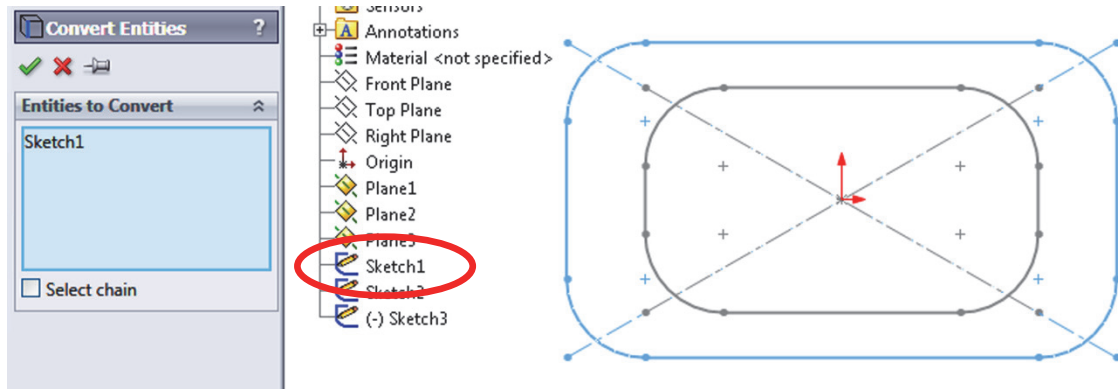
**188.** - Still in the Top view, select 'Plane1' from the FeatureManager and create the second sketch. Use the "Center Rectangle" tool; be sure to start in the origin and make the rectangle's corner coincident to the previous sketch's diagonal line. Add a 2.5" width dimension and the 0.5" "Sketch Fillet"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



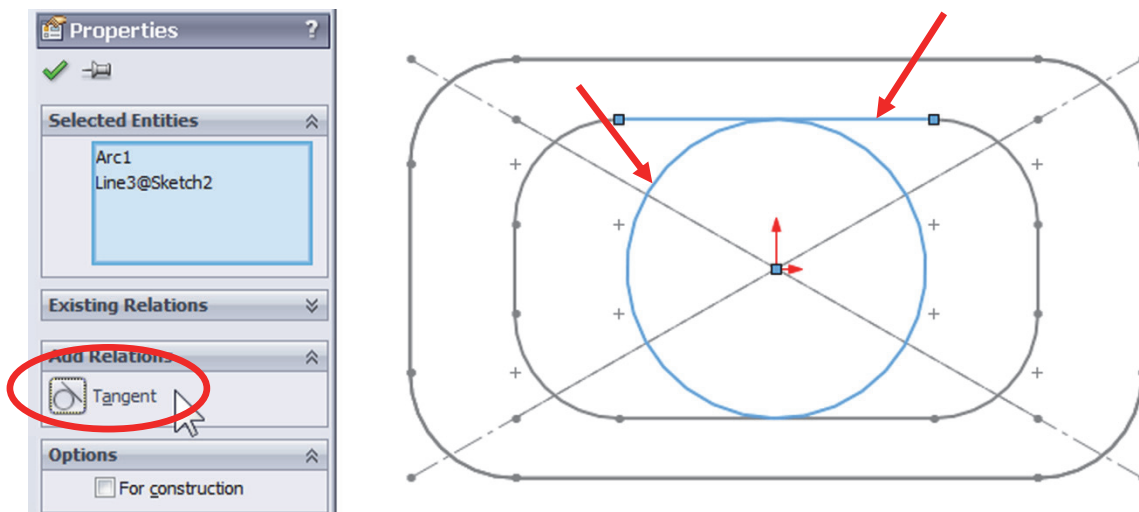
Turn off the display of Planes for clarity with "Hide/Show Items."



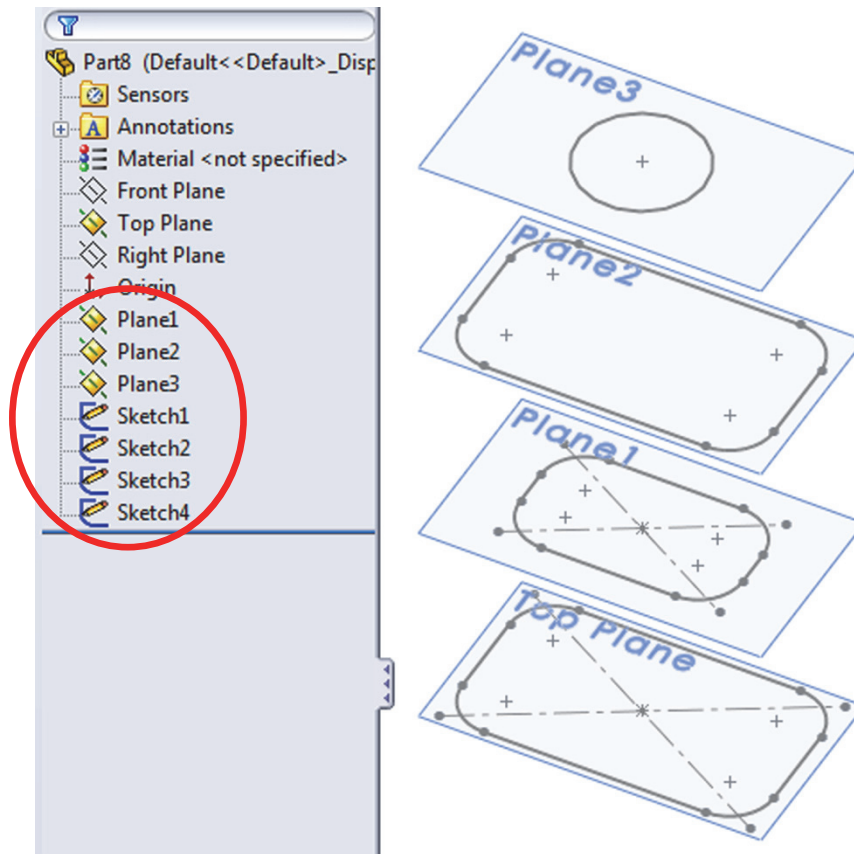
**189.** - For the third profile, select '*Plane2*' from the FeatureManager and create a new sketch in it. This sketch will be exactly the same as the first one. To help us save time and maintain design intent, we'll use the “**Convert Entities**” tool. In the “**Convert Entities**” selection box, select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



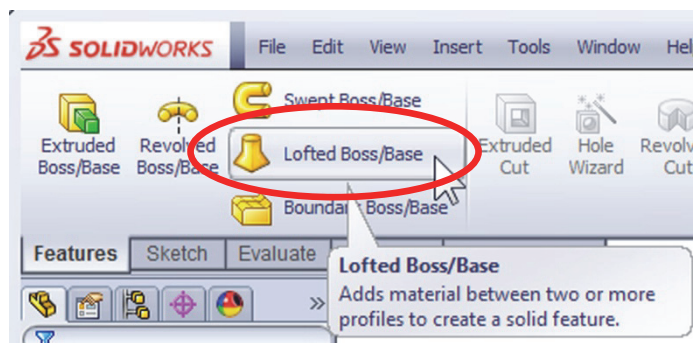
**190.** - For the last profile select '*Plane3*' from the FeatureManager and create a new sketch. Draw a circle and add a geometric relation to make it “Tangent” to the horizontal line in '*Sketch2*' as indicated. This relation will fully define the sketch. Exit the sketch to finish.



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

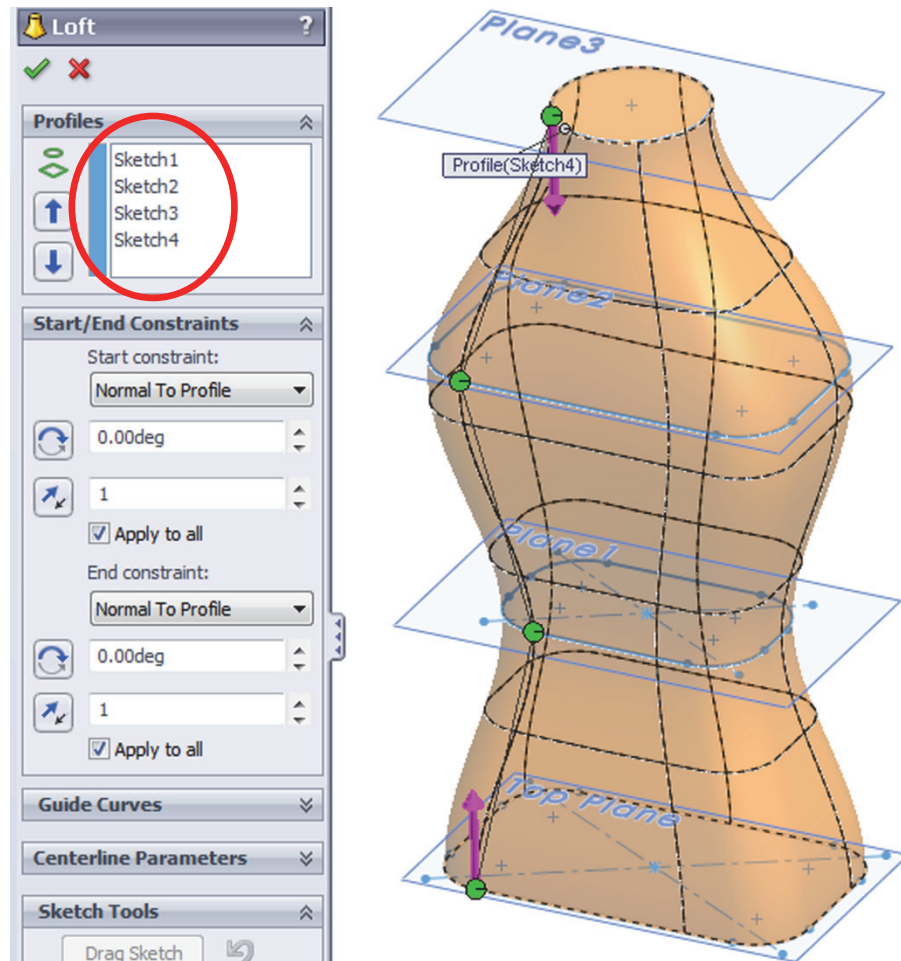


192. - Now we are ready, select the “Loft” icon from the Features tab. The loft feature requires two or more sketches and/or faces, and we'll use the four sketches we just made to build the bottle.

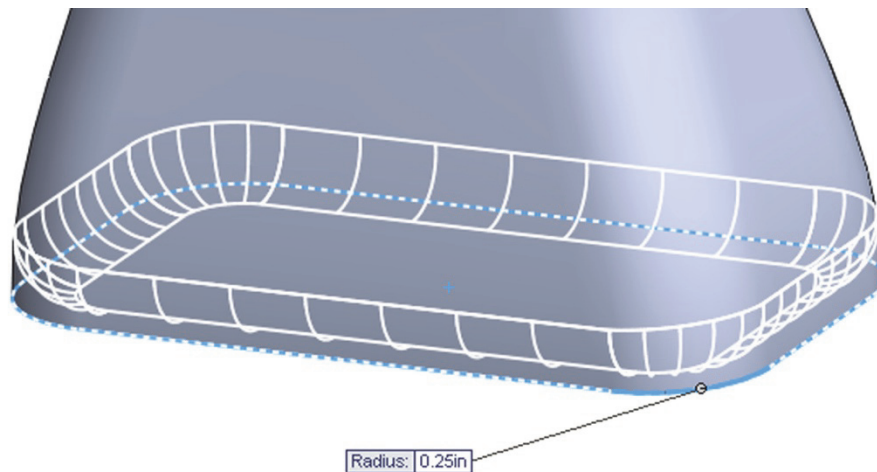


We will select the profiles starting with the first one we made at the bottom and finishing with the last one at the top. It is important to select the sketches thinking that where we select the profile will affect the result. Click in the graphics area near the indicated 'dots', this line indicates the segment of one profile that will be connected to the next profile. If we select points randomly in the profiles, 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.



**193.** - Add a 0.25" radius fillet in the bottom edge of the part to round it off.



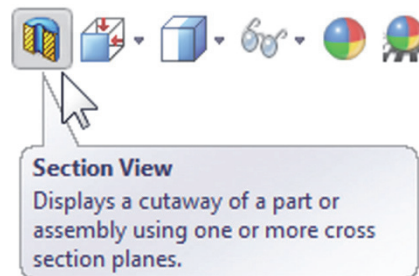
Finish the part using the “Shell” tool; select the top face to remove it and make the wall thickness 0.125”.



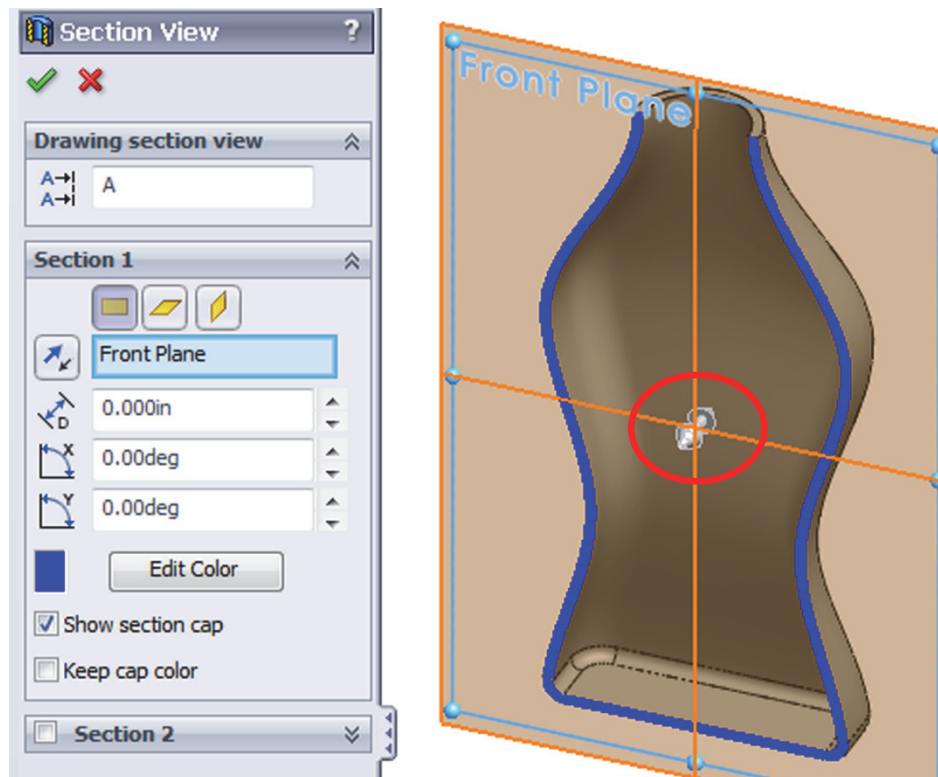
*Image shown using Real View*

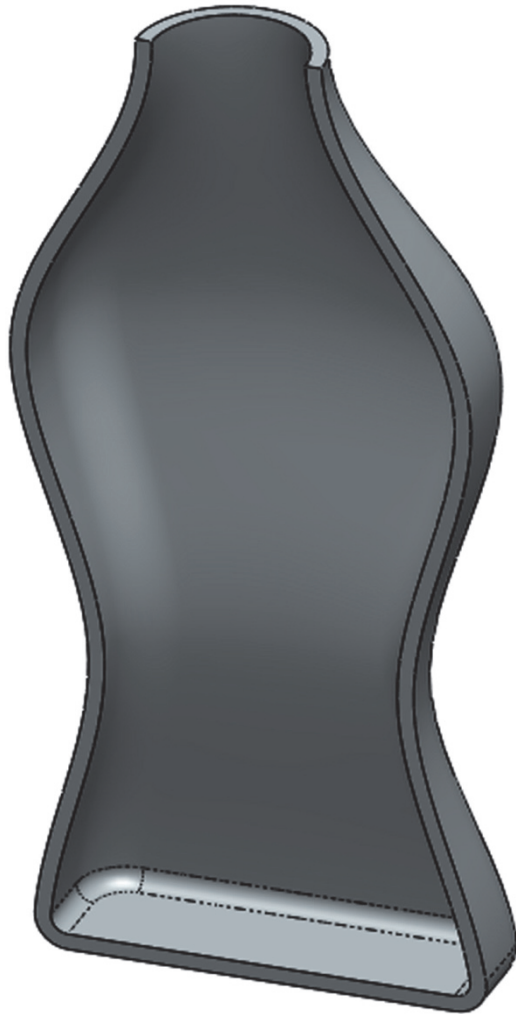


**194.** - To view the inside of the part, select the **"Section View"** icon from the View toolbar or the menu **"View, Display, Section View."** We can define which plane to cut the model with, the depth of the cut and optionally add a second or third section. If we click OK in the Section View, the 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.



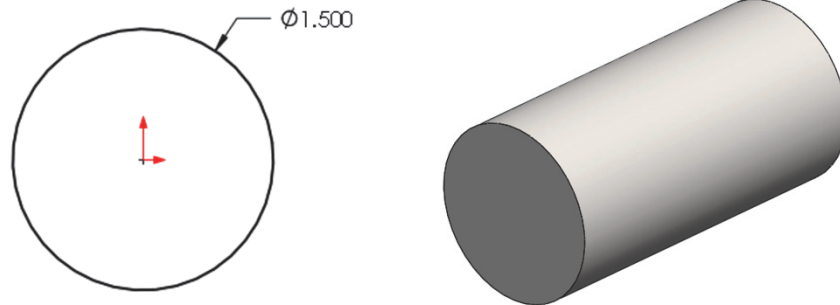


Save the part as '*Bottle Loff*' and close.

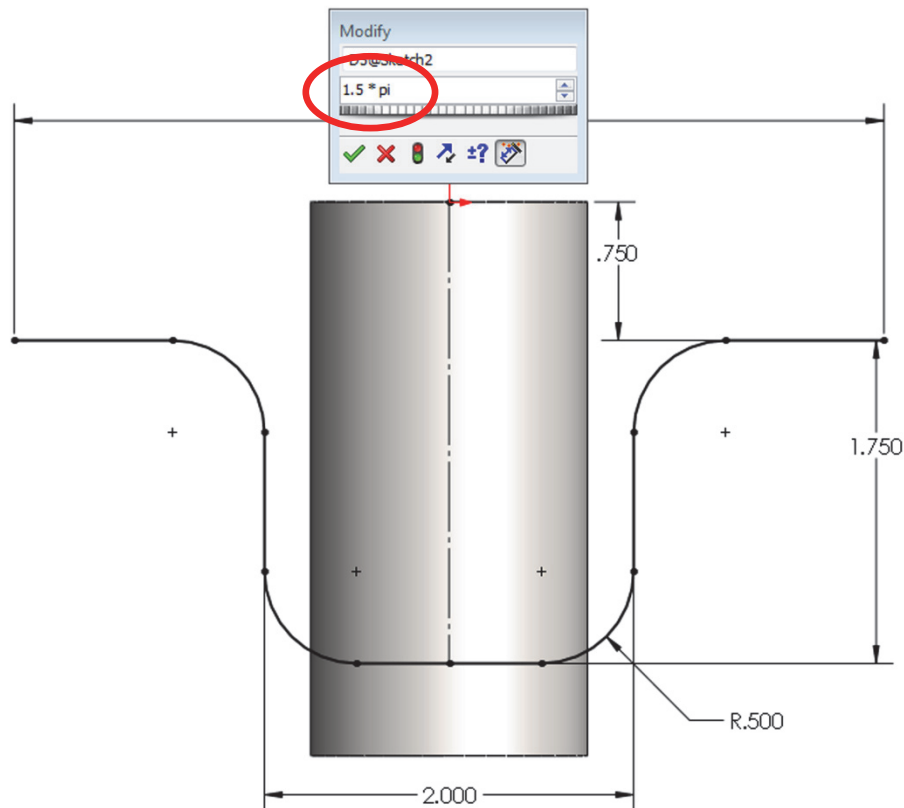
## Wrap Feature

The wrap feature helps us create cylindrical cams or slots on cylindrical surfaces using a sketch on a plane tangent to the surface we want to make the wrap on.

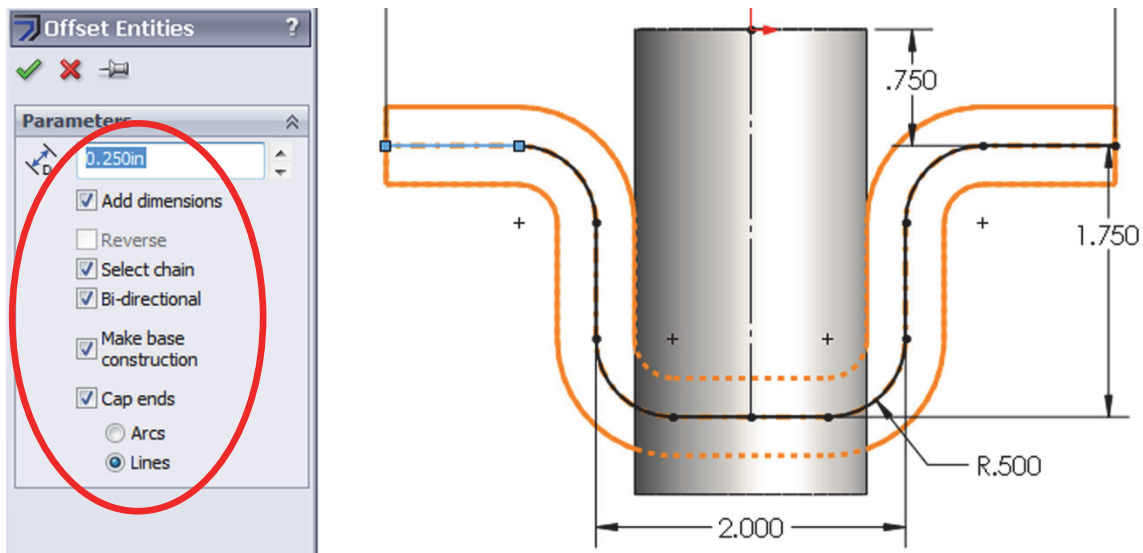
**195.** – Make a new part, set units to inches with three decimal places; add this sketch on the "Front Plane" and extrude it 3".



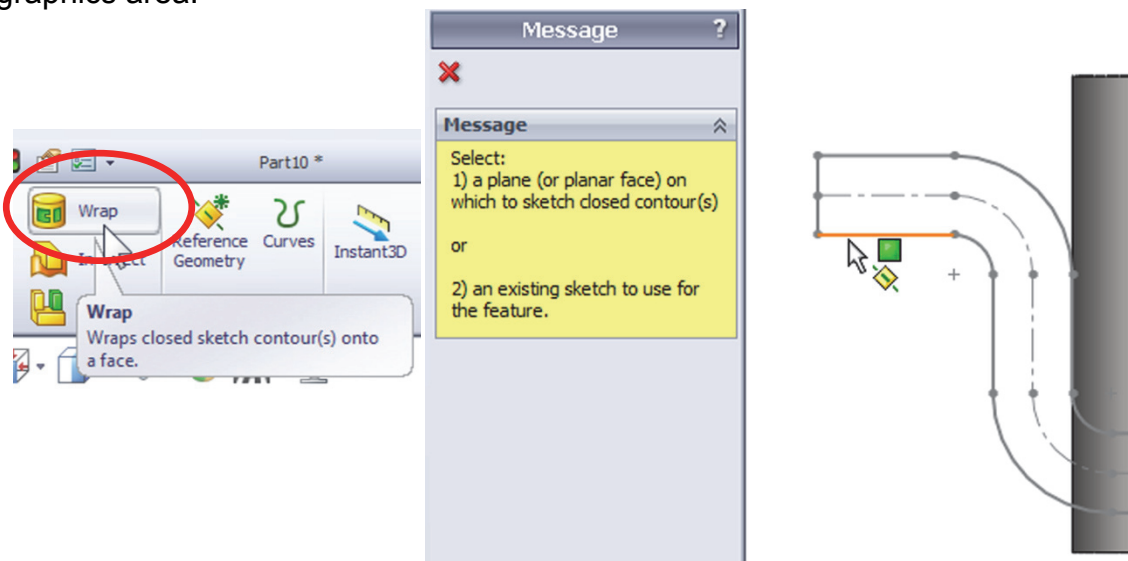
**196.** – The next step is to make the sketch that will be used for the wrap feature. For the sketch plane we have an option of using the "Top Plane", "Right Plane" or create a new plane tangent to the cylinder's surface. For this example we'll use the "Top Plane." Draw the following profile first making sure its symmetrical about the centerline. All arcs are equal radii. The top dimension's value is entered as  $1.5 * \pi$ , since we want the sketch to wrap completely around the cylinder. SolidWorks automatically calculates the resulting value for us.



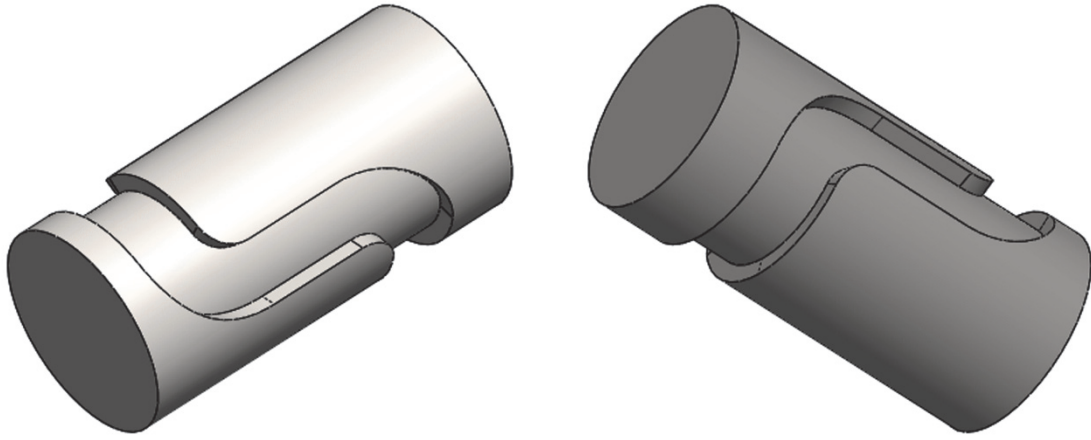
197. - The sketch so far is the centerline of the slot we are going to wrap around the cylinder. Select the "**Offset**" command and turn on the options: "**Select chain**" to automatically select the entire centerline picking only one segment, "**Bi-directional**" to make the offset in both directions, "**Make base construction**" to make the selected line (or chain) construction geometry, and after selecting a segment of the sketch the "**Cap ends**" option is enabled letting us choose to cap the offset with Arcs or Lines. Making the offset 0.25" will give us a 0.5" wide slot. Click OK to complete the offset and exit the sketch. Note how the original sketch is now construction geometry and we have a closed profile. Exit the sketch when done.



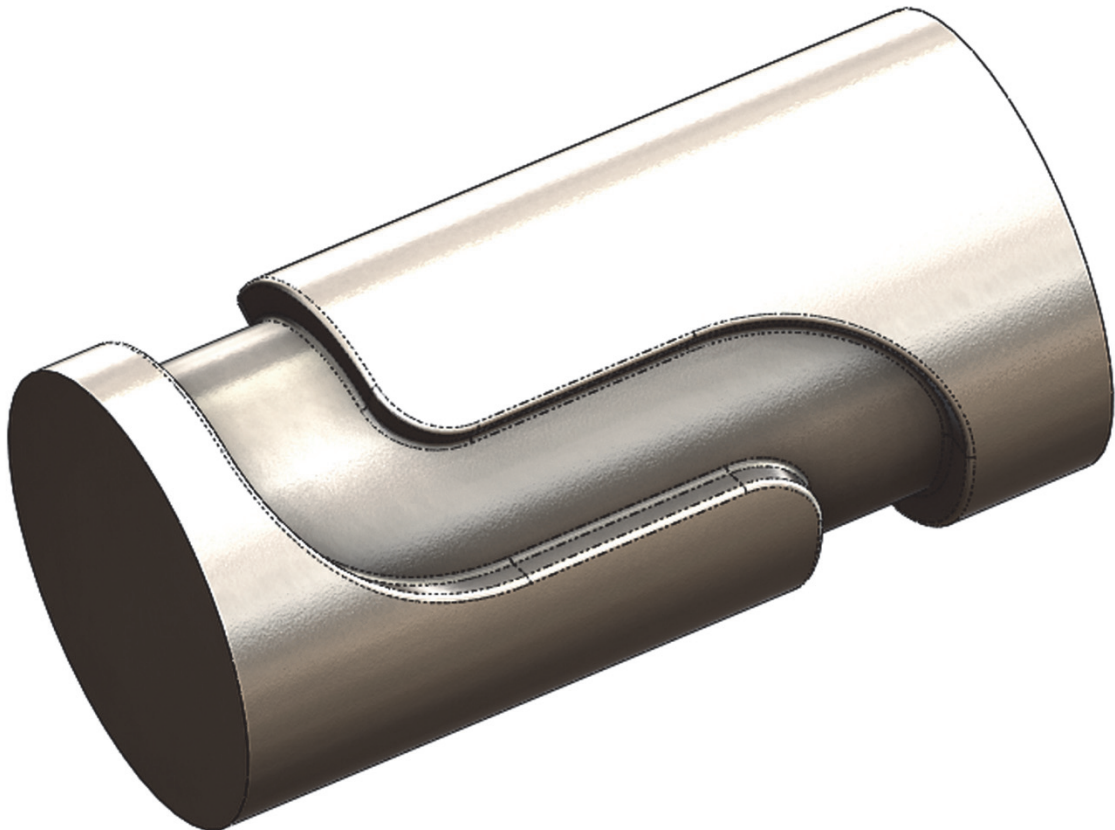
198.- Select the "**Wrap**" feature from the Features tab or the menu "**Insert, Features, Wrap.**" If the sketch is not pre-selected, select the sketch from the graphics area.



**199.-** Select the "Deboss" option to make a cut in the part (The "Emboss" option will add material and the "Scribe" will split the face.) Select the cylinder's face in the "Face for Wrap Feature" selection box and make the depth 0.1". Click OK to finish.



**200. –** Add a 0.02" fillet to finish the part. Save as '*Cylindrical Cam*'.

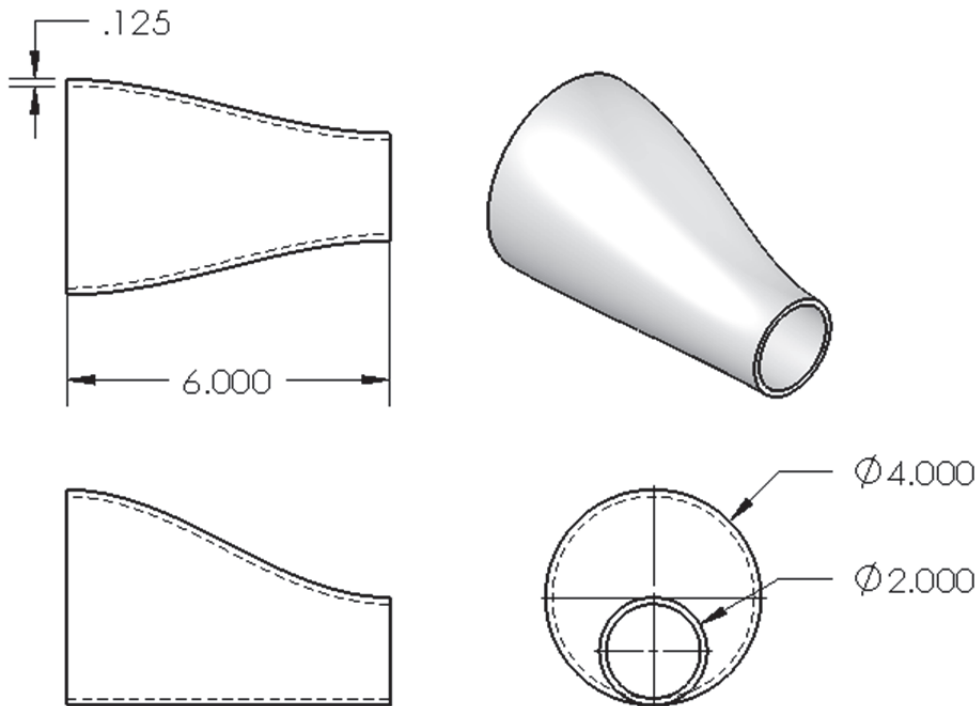


**Exercises:** Build the following parts using the knowledge acquired in this lesson. Try to use the most efficient method to complete the model.

### Eccentric Coupler

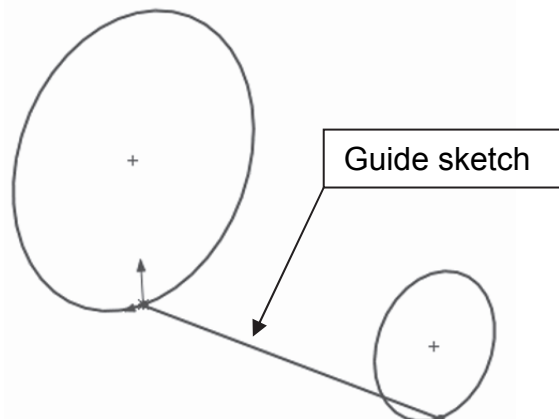
Notes:

- 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.

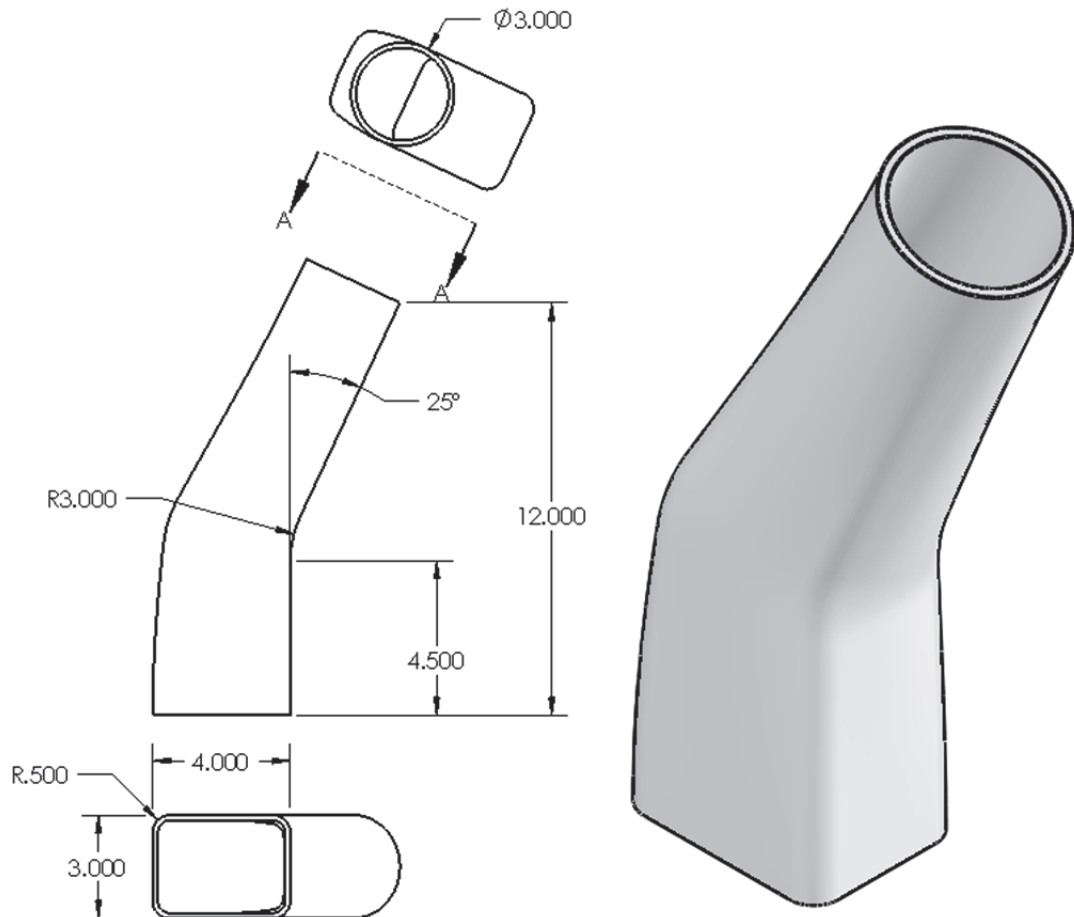


## Bent Coupler

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

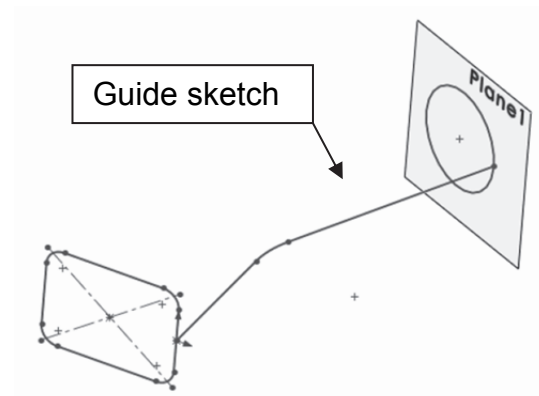
Notes:

- 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:

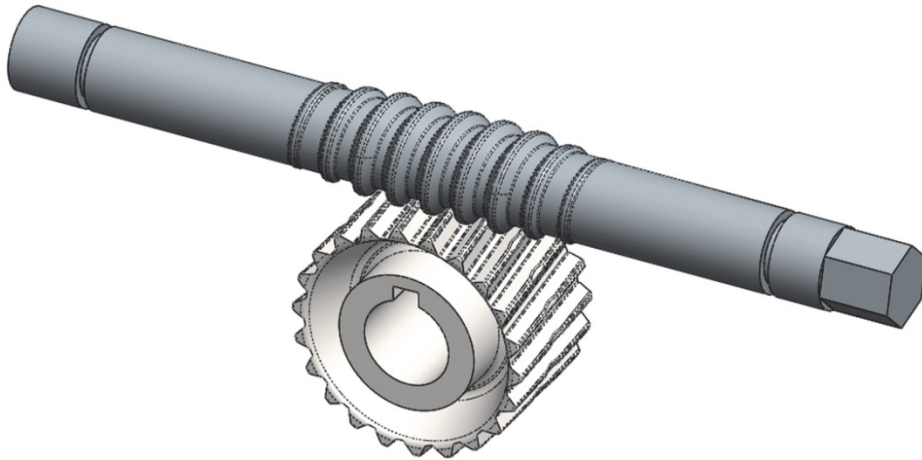
- Make Rectangular sketch first.
- Make Guide sketch.
- Make Auxiliary Plane perpendicular to Guide sketch at the end point.
- Make circular sketch in Auxiliary Plane.
- Make Loft using Guide sketch as a guide curve.
- Shell the part.



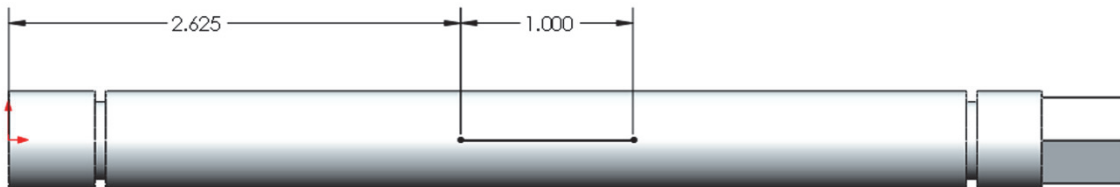


**Challenge Exercises:** Build the 'Worm Gear' and 'Offset Shaft' complete gears using the knowledge learned so far with the information given. High resolution images at [mechanicad.com](http://mechanicad.com).

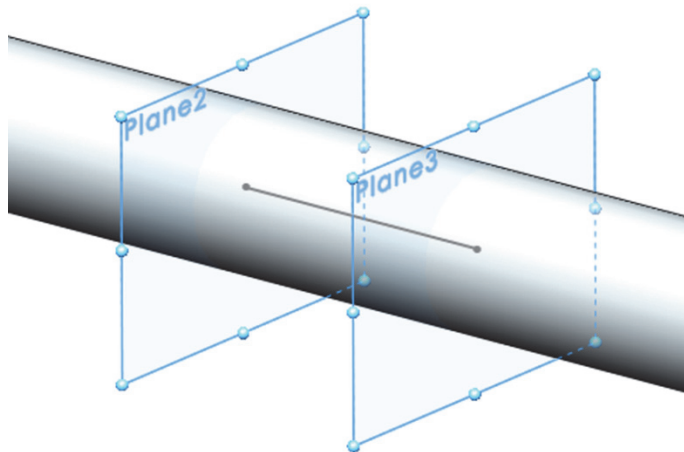
**DISCLOSURE:** The gears modeled in this tutorial are not intended for manufacturing, nor is this tutorial meant to be a gear design guide. Its sole purpose is to show the reader how to apply the learned knowledge using a simplified version of the gears.



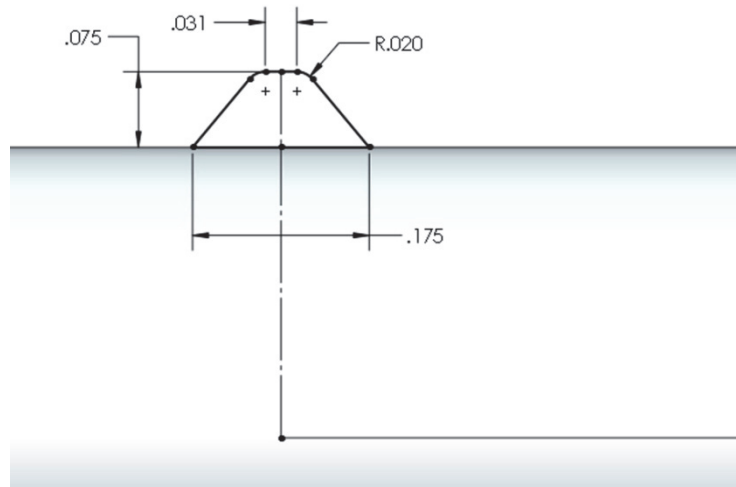
- **Offset Shaft:** Add the following sketch in the "Front Plane", Exit the sketch. Rename it "Gear Width." This is to be the length of the full size helix.



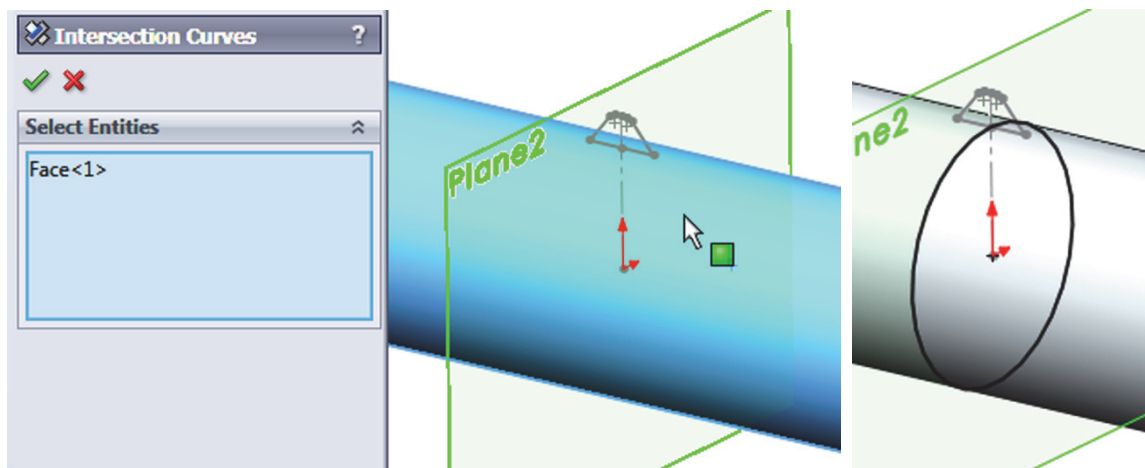
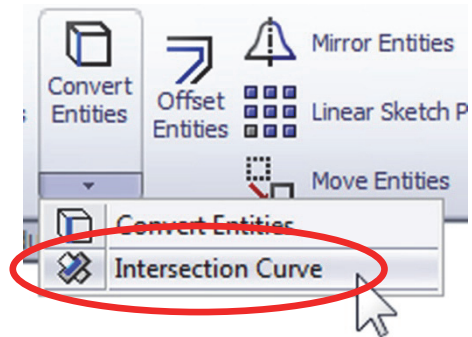
- Build two reference planes parallel to the "Right Plane", one at each end of the "Gear Width" sketch.



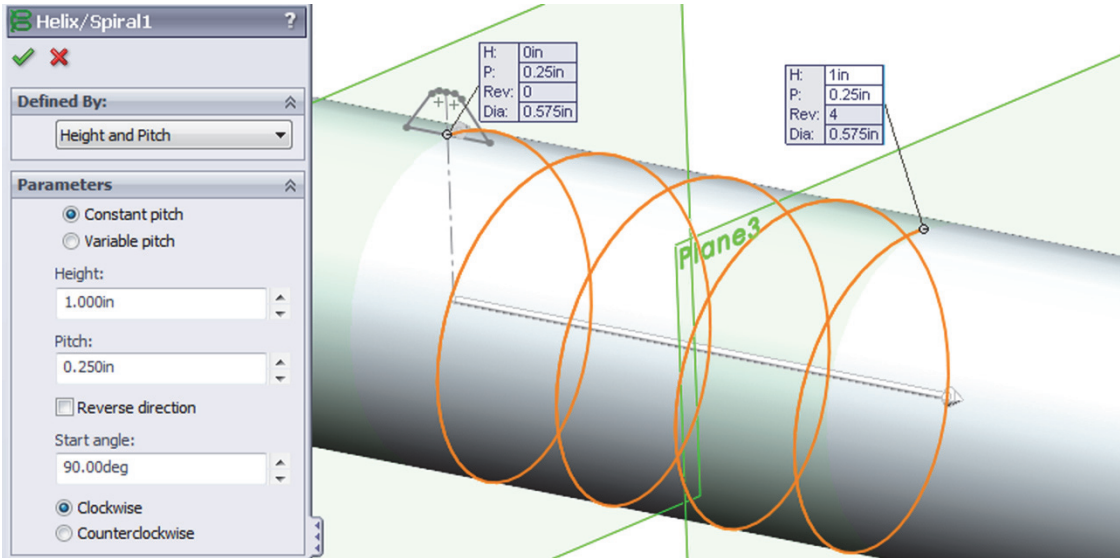
- Add the gear's profile sketch in the "*Front Plane*", at the left side of the "*Gear Width*" sketch. Exit the sketch and rename "*Thread Profile*."



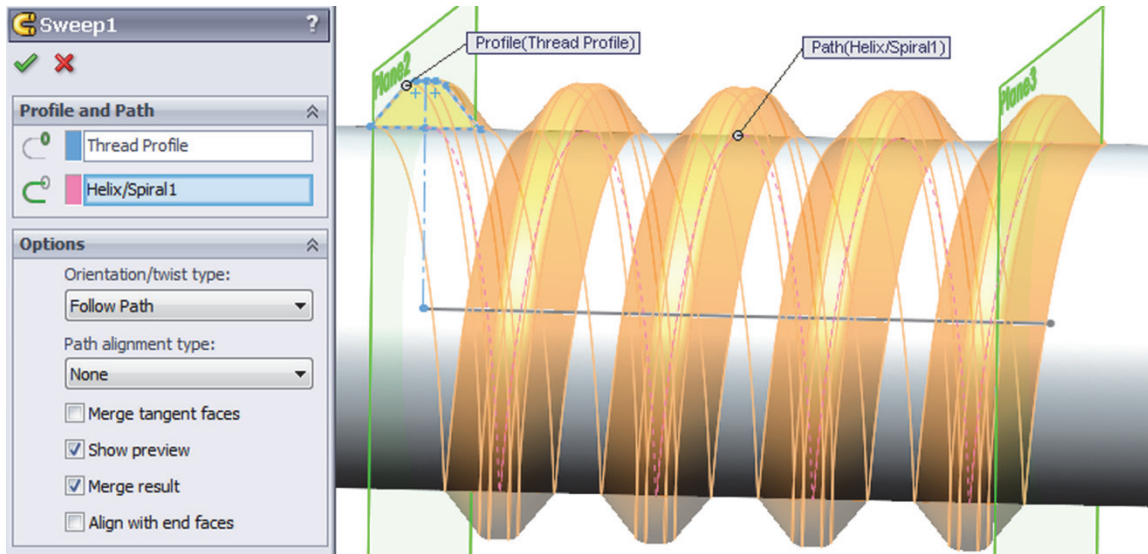
- In the plane located at the left of the "*Gear Width*" sketch add a new sketch, use the "Convert Entities" drop down icon to select "**Intersection Curve**", this command will create sketch entities at the intersection of the selected surface(s) and the current sketch plane. This tool is particularly useful when we have irregular surfaces intersecting the sketch plane. In this case it works the same as "Convert Entities", but for oblique surfaces it's the best option to obtain the intersection of the surface and the sketch plane.



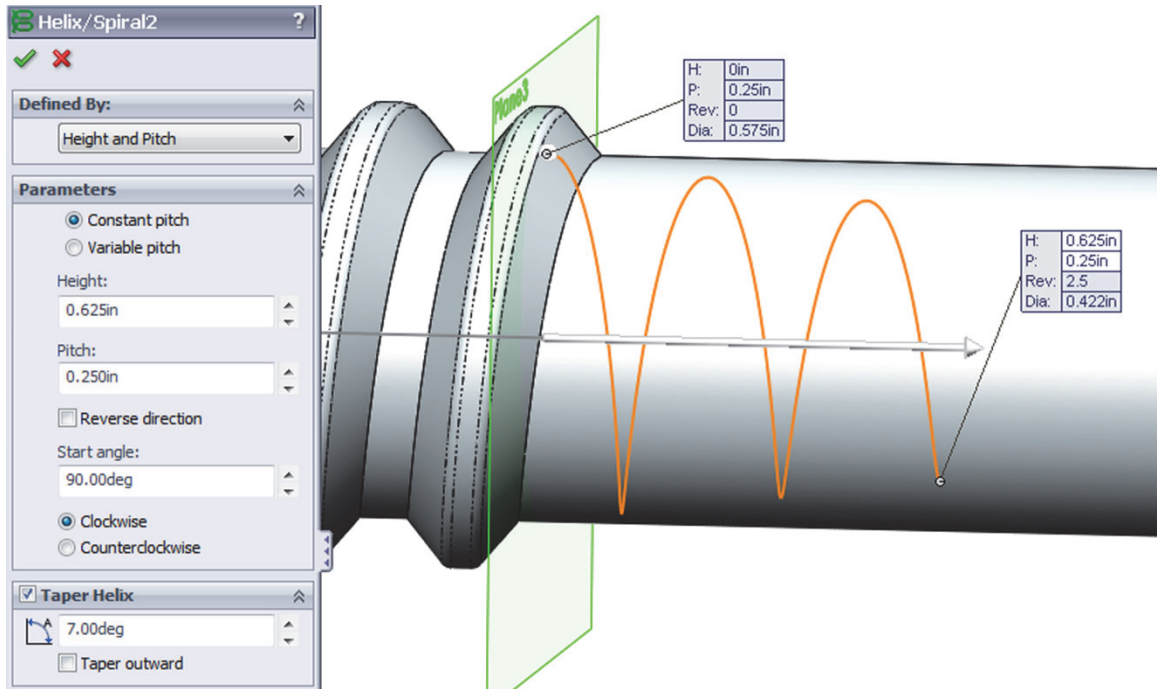
- Make the Helix 1" Height with a 0.25" Pitch



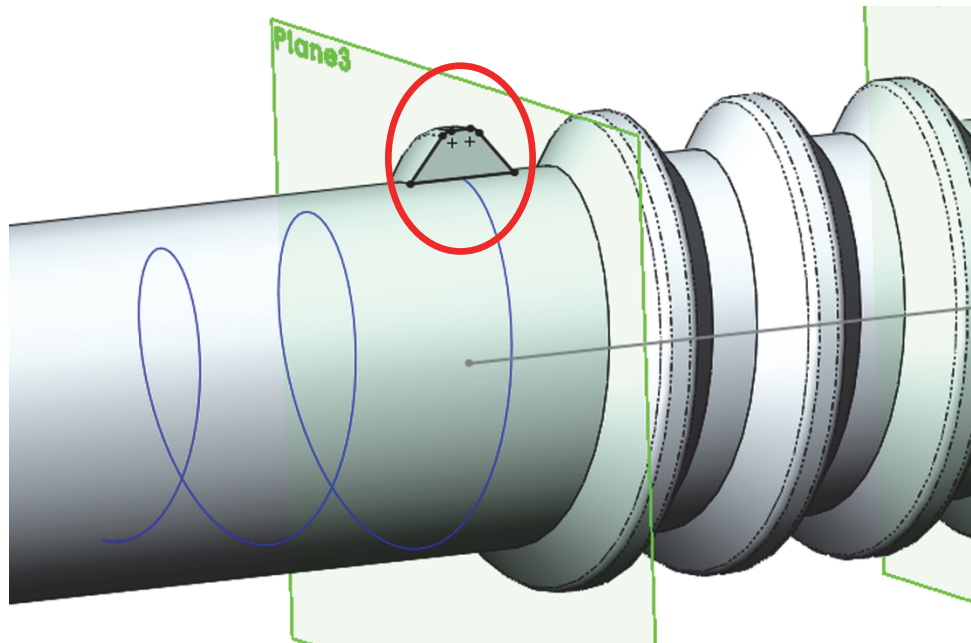
- Make the first sweep using the profile and the first helix.



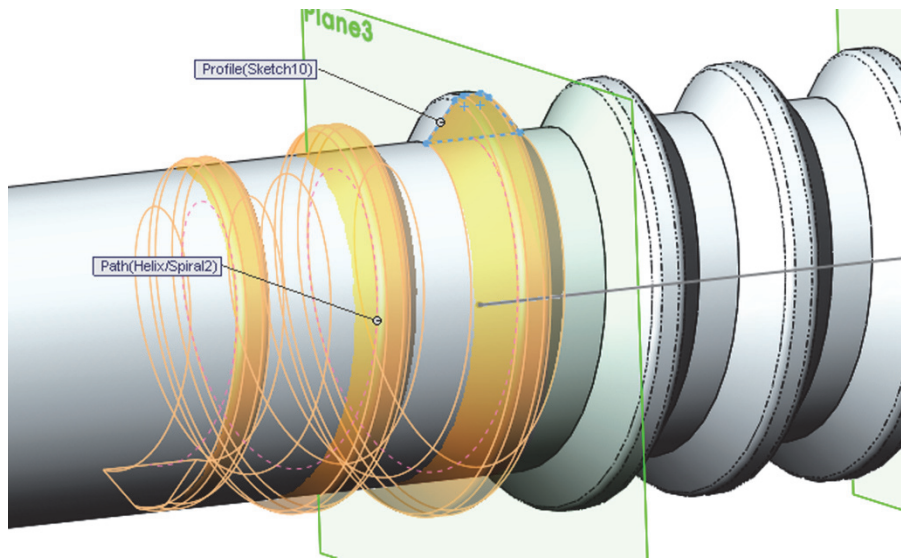
- In the plane at the right of the "Gear Width" sketch add a sketch to start a second Helix using "Convert Entities", and make the helix activating the "Taper Helix" option with a 7deg taper.



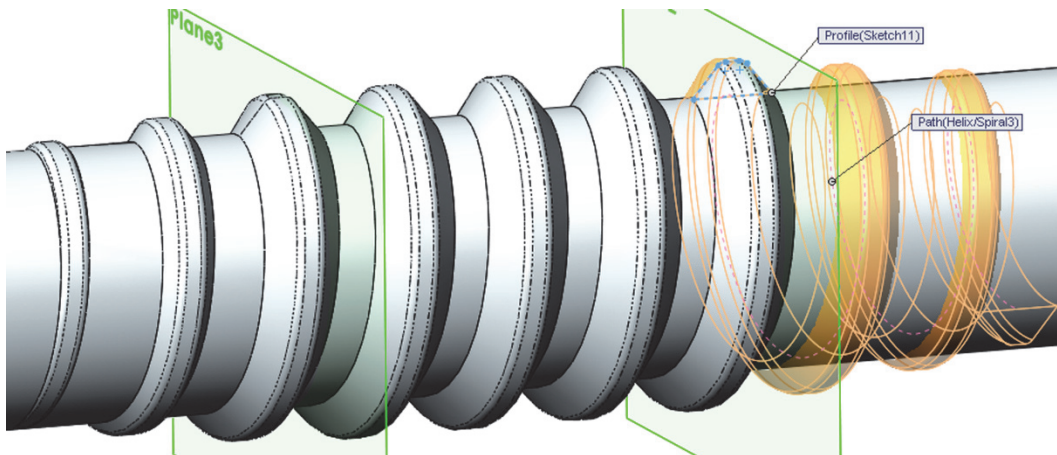
- Add a sketch at the flat end of the thread using "**Convert Entities**" to use as the profile for the tapered helix (Rear view), and make the sweep.



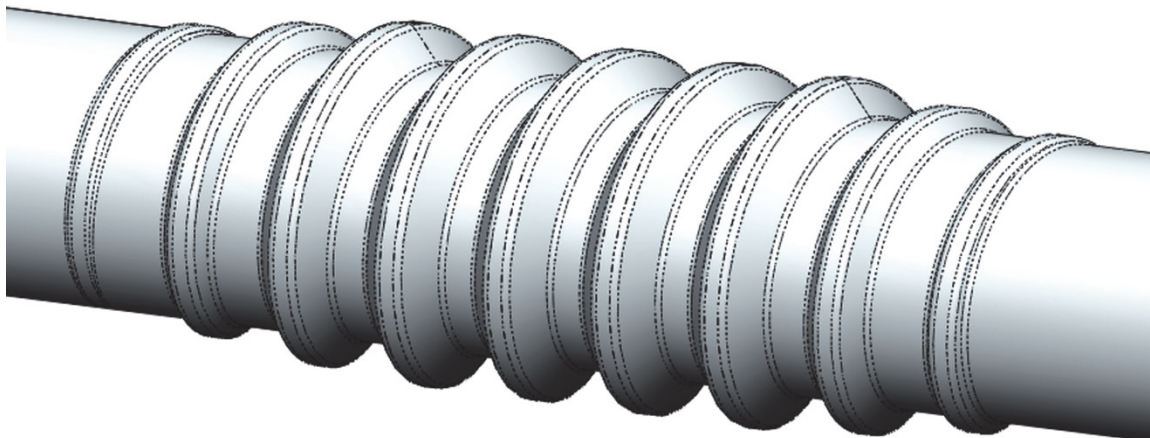




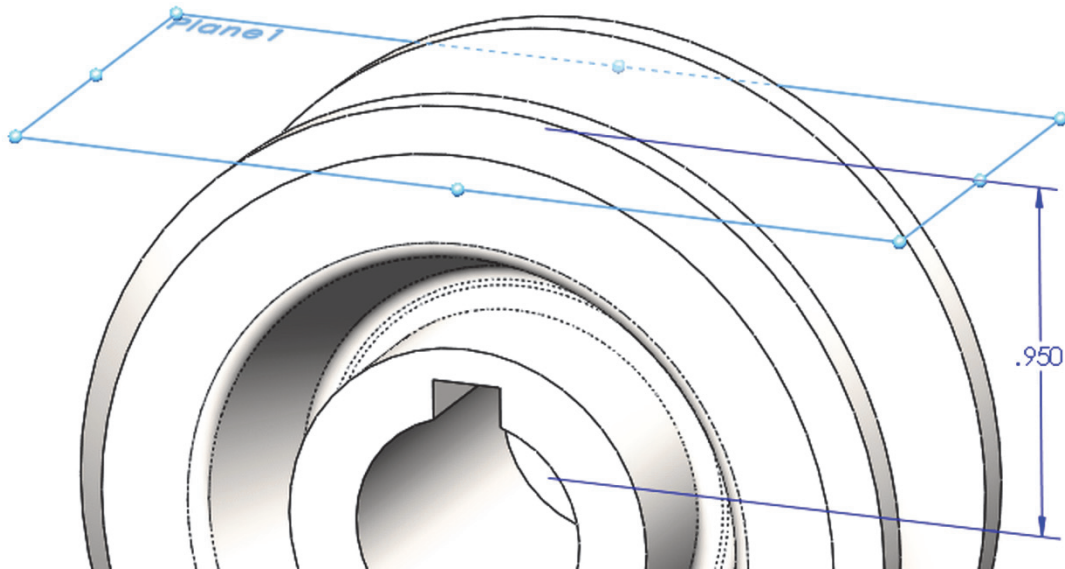
- Repeat a helix and sweep for the other side



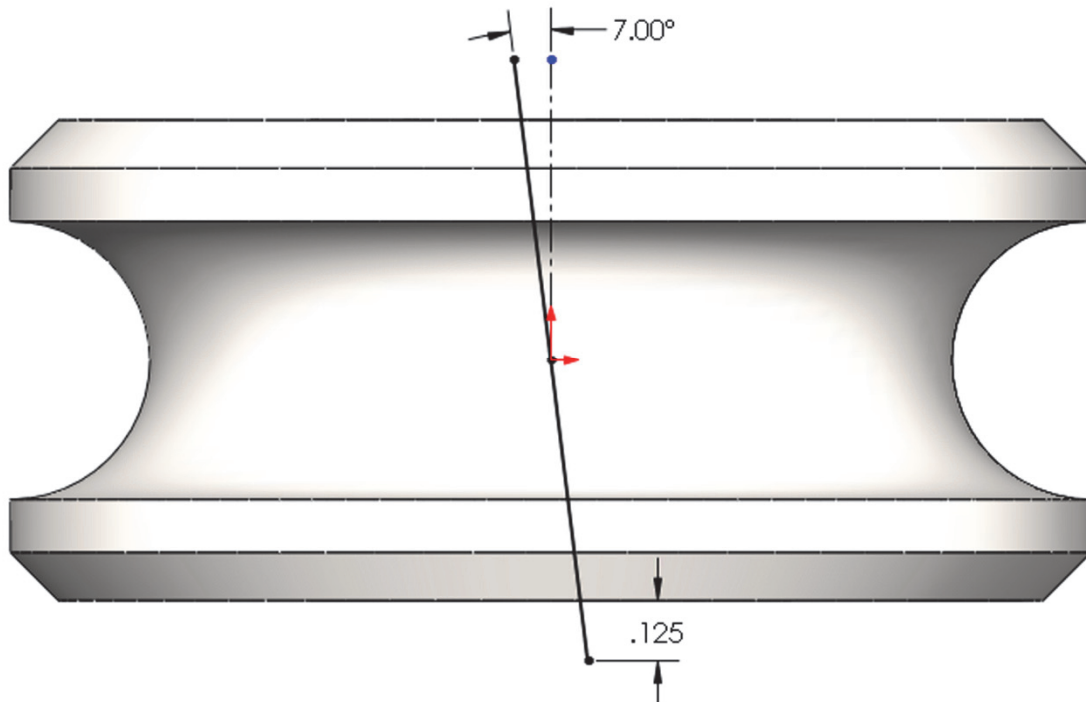
- Add a 0.015" fillet to round the edges of the thread. Save as '*Offset Shaft Gear*' and close.



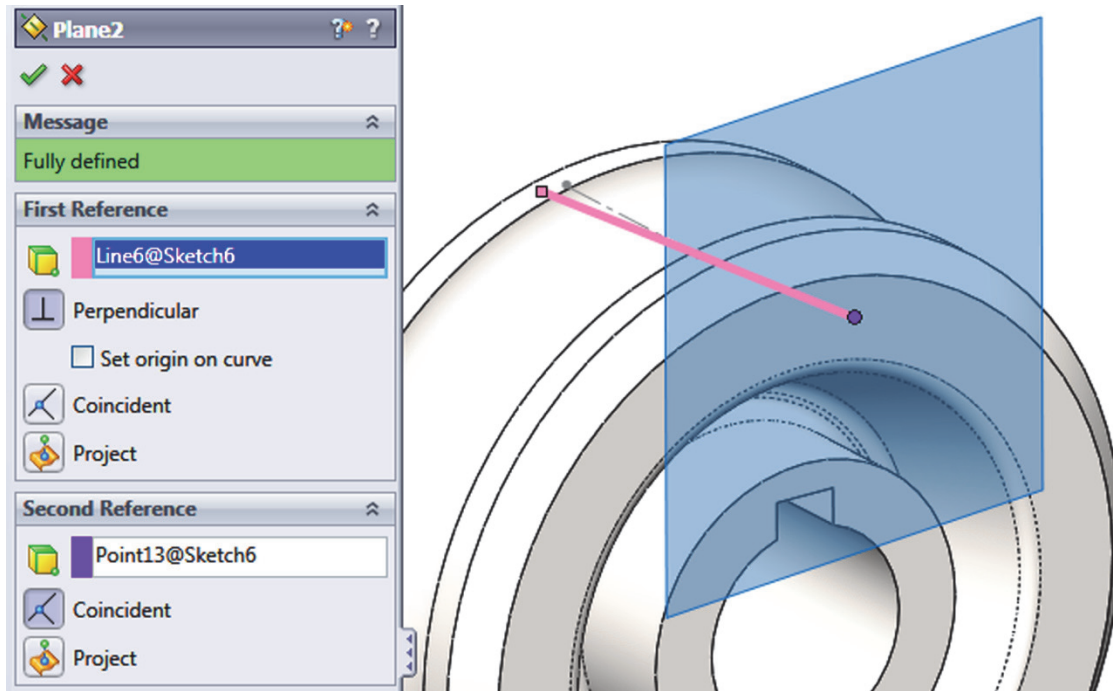
- **Worm Gear:** Create a parallel plane 0.95" above to the "Top Plane."



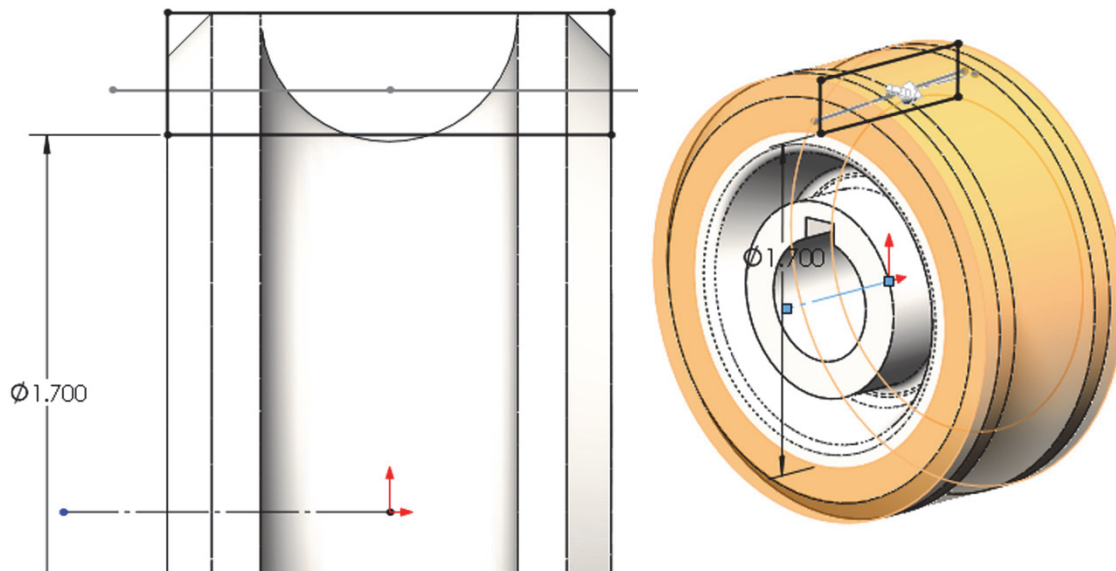
- Add the following sketch in the plane made in the previous step, exit the sketch and rename it "Path."



- Add a new auxiliary plane perpendicular to the "Path" sketch. Rename it "Profile Plane."

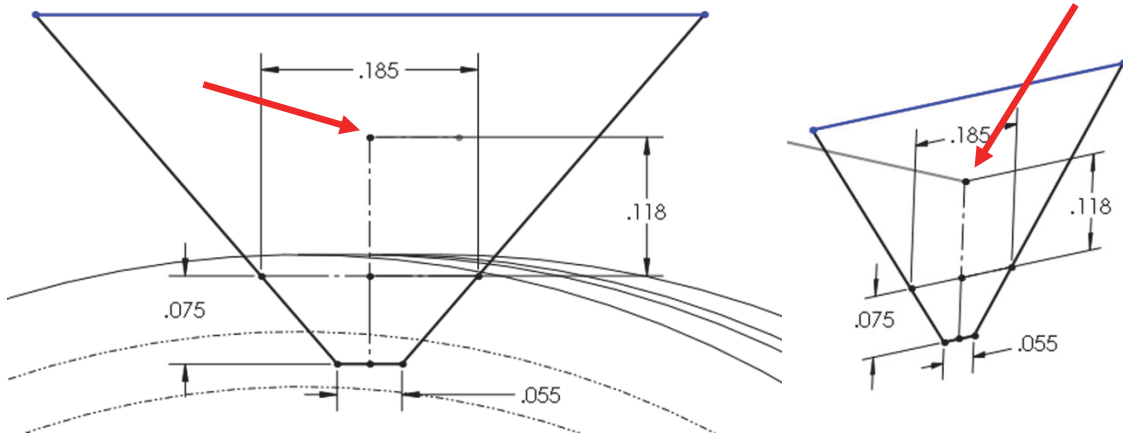


- Add this sketch in the "Right Plane" and make a revolved cut.

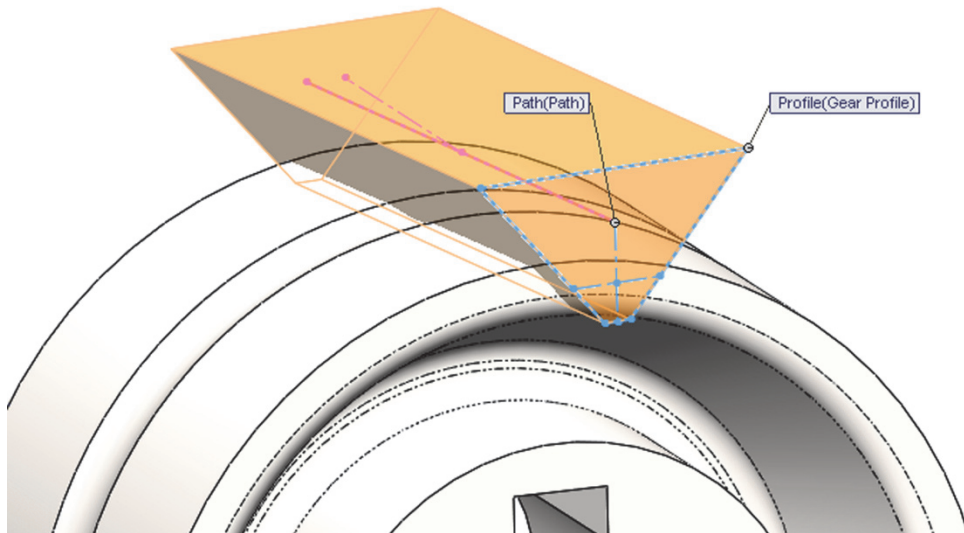




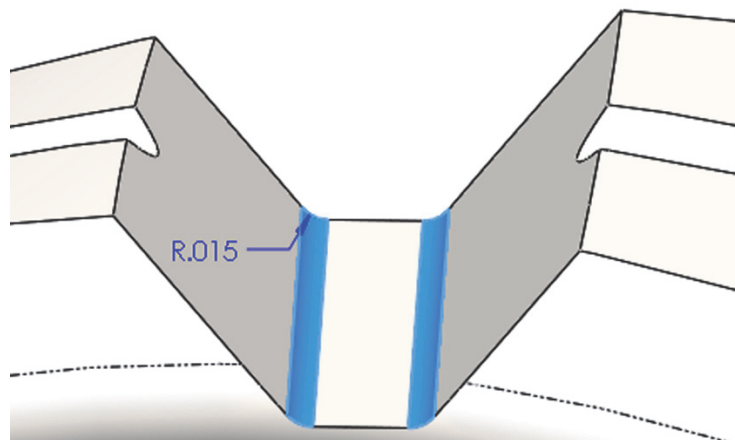
- In the "Profile Plane" add the following sketch. This will be the gear cut profile. Locate the sketch using the "Path" sketch at the indicated point. Exit the sketch and rename "Gear Profile"



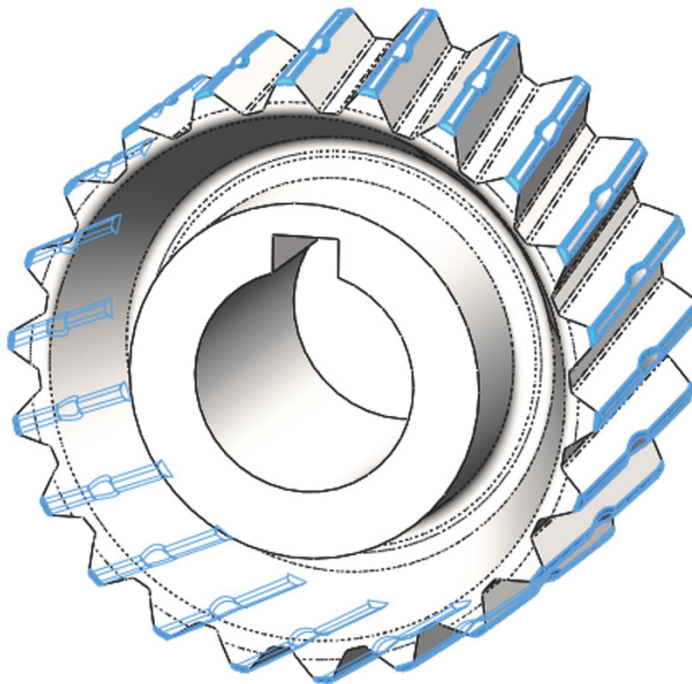
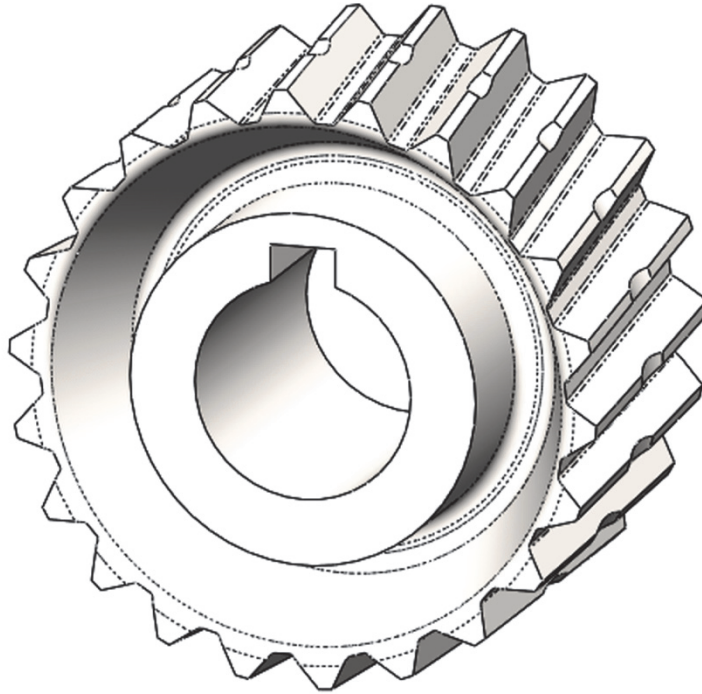
- Make a "Cut Sweep" using the "Path" and "Gear Profile" sketches (We can also make a "Cut Extrude" in this case works the same.)



- Add a 0.015" fillet at the bottom of the sweep cut.



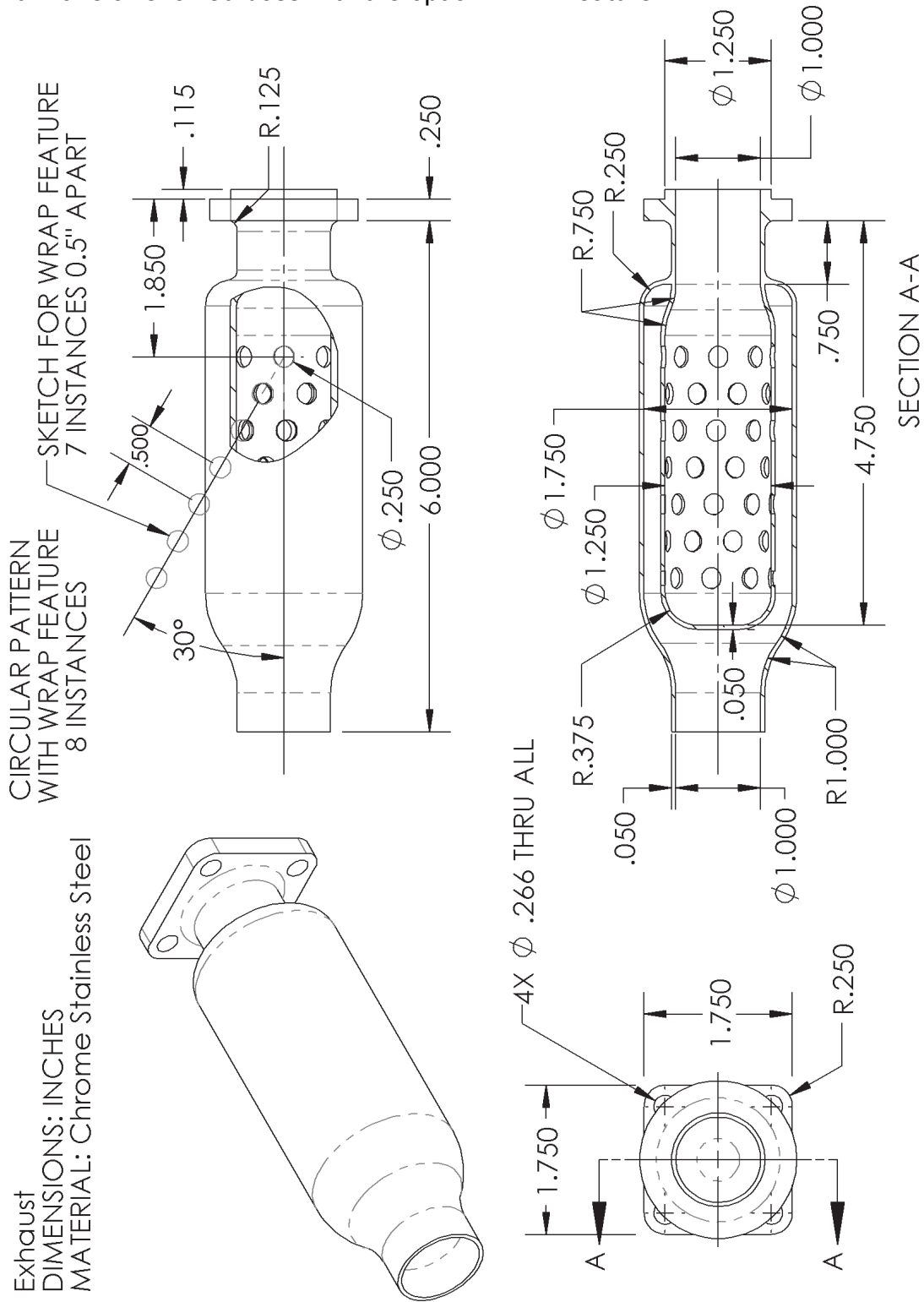
- Make 22 copies of the Cut-Sweep and the previous fillet. Add a 0.015" fillet to the "Cut-Revolve1" and "Cut-Revolve3" to finish the gear.



Save as '*Worm Gear Complete*' and close.

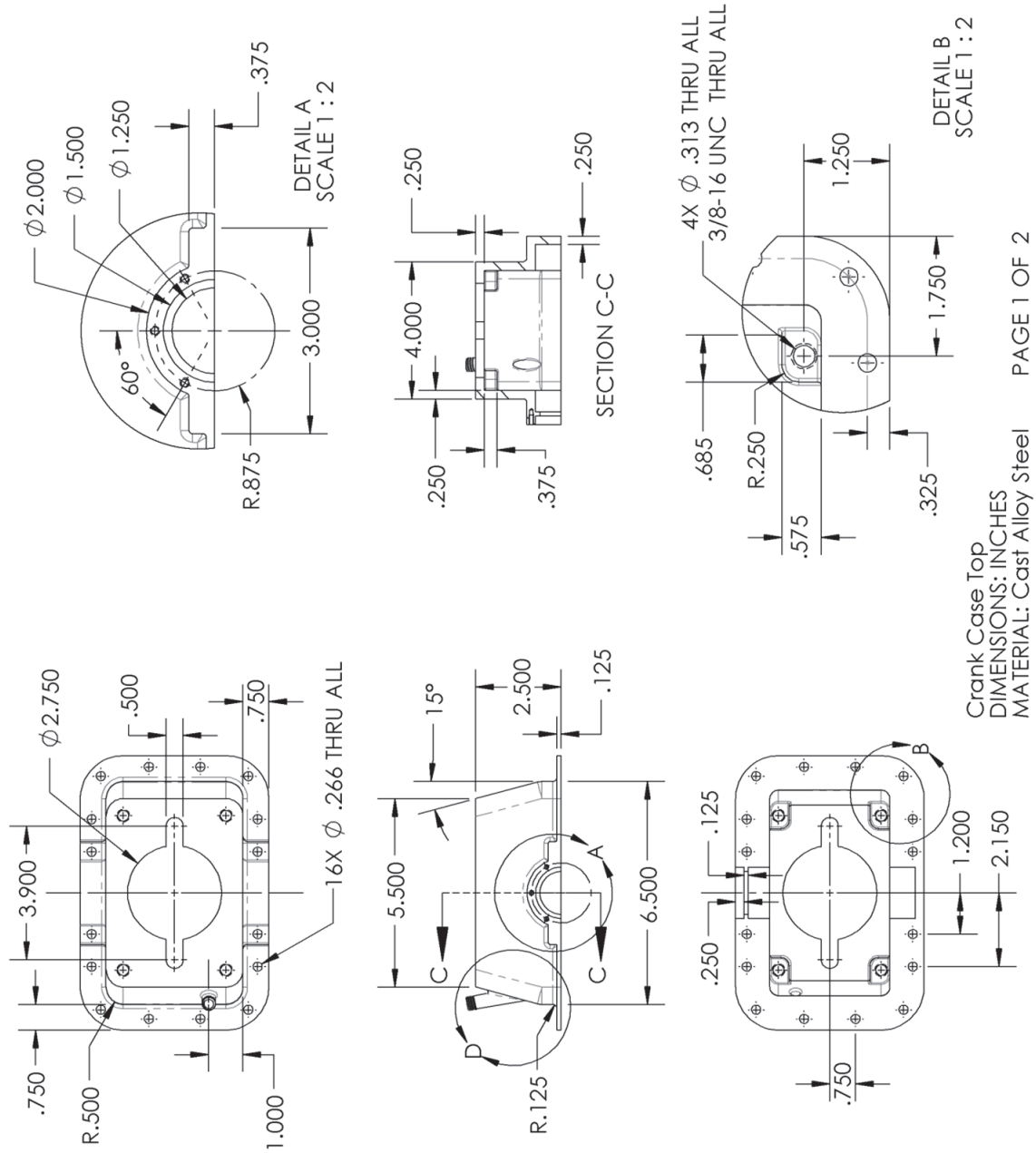
**Engine Project Parts:** Make the following components to build the engine. Save the parts using the name provided. High resolution images at [mechanicad.com](http://mechanicad.com).

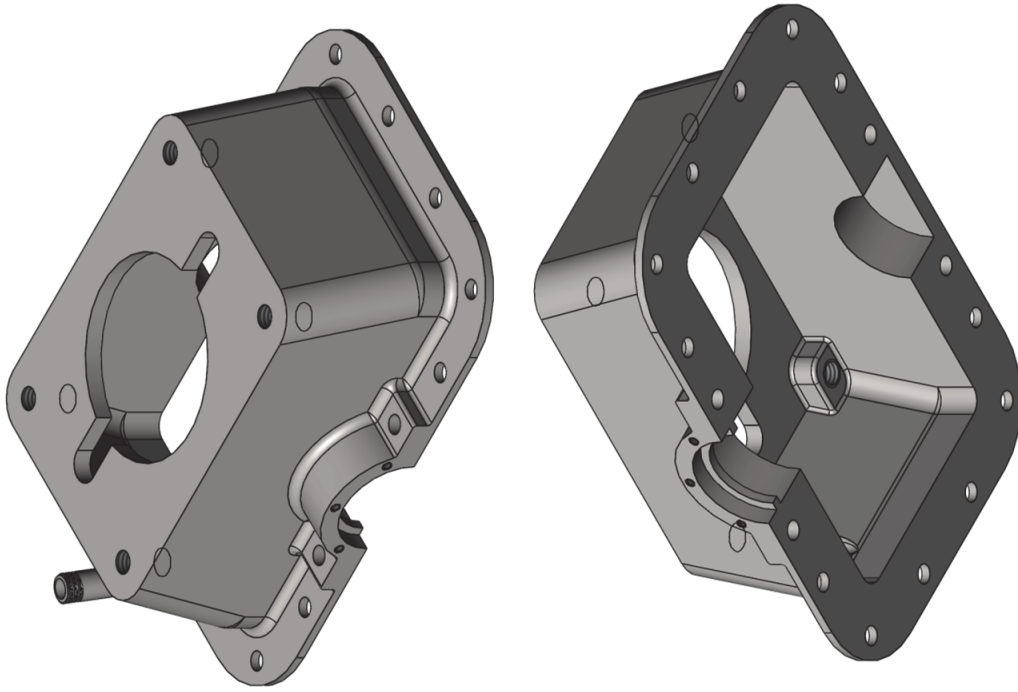
Hint: Make a revolved boss with the option "Thin Feature."



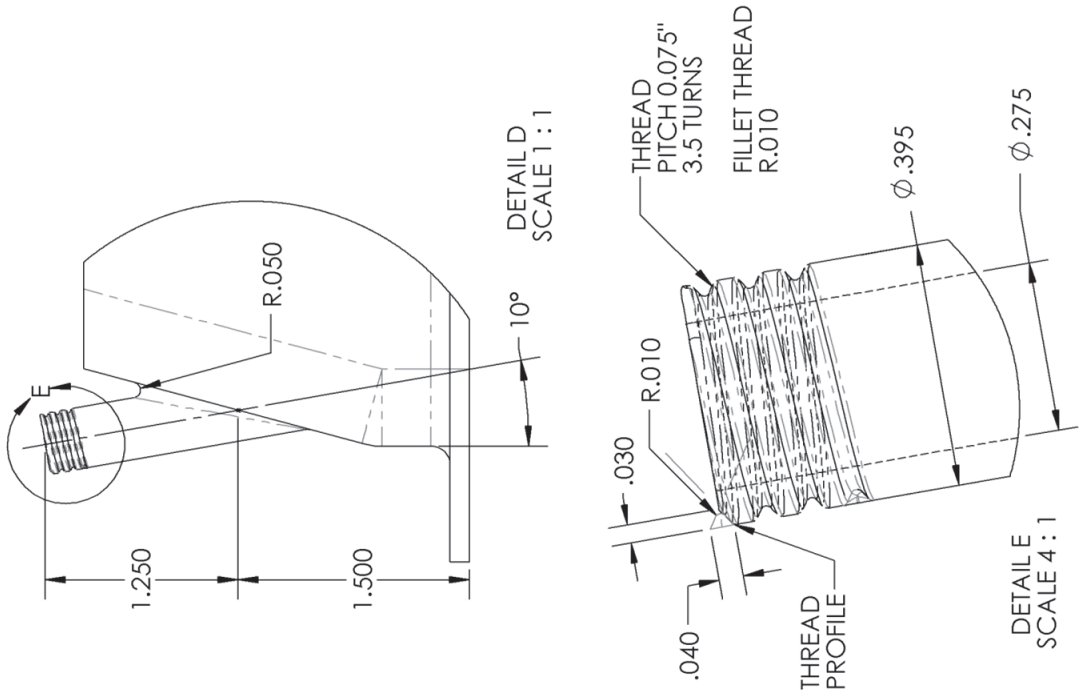
**Final Parts for the Engine Project:** The following components are the last required to complete the engine. Save the parts using the name provided. A help sheet with a sequence of features is located in the Appendix for each of the last three parts. High resolution images at [mechanicad.com](http://mechanicad.com).

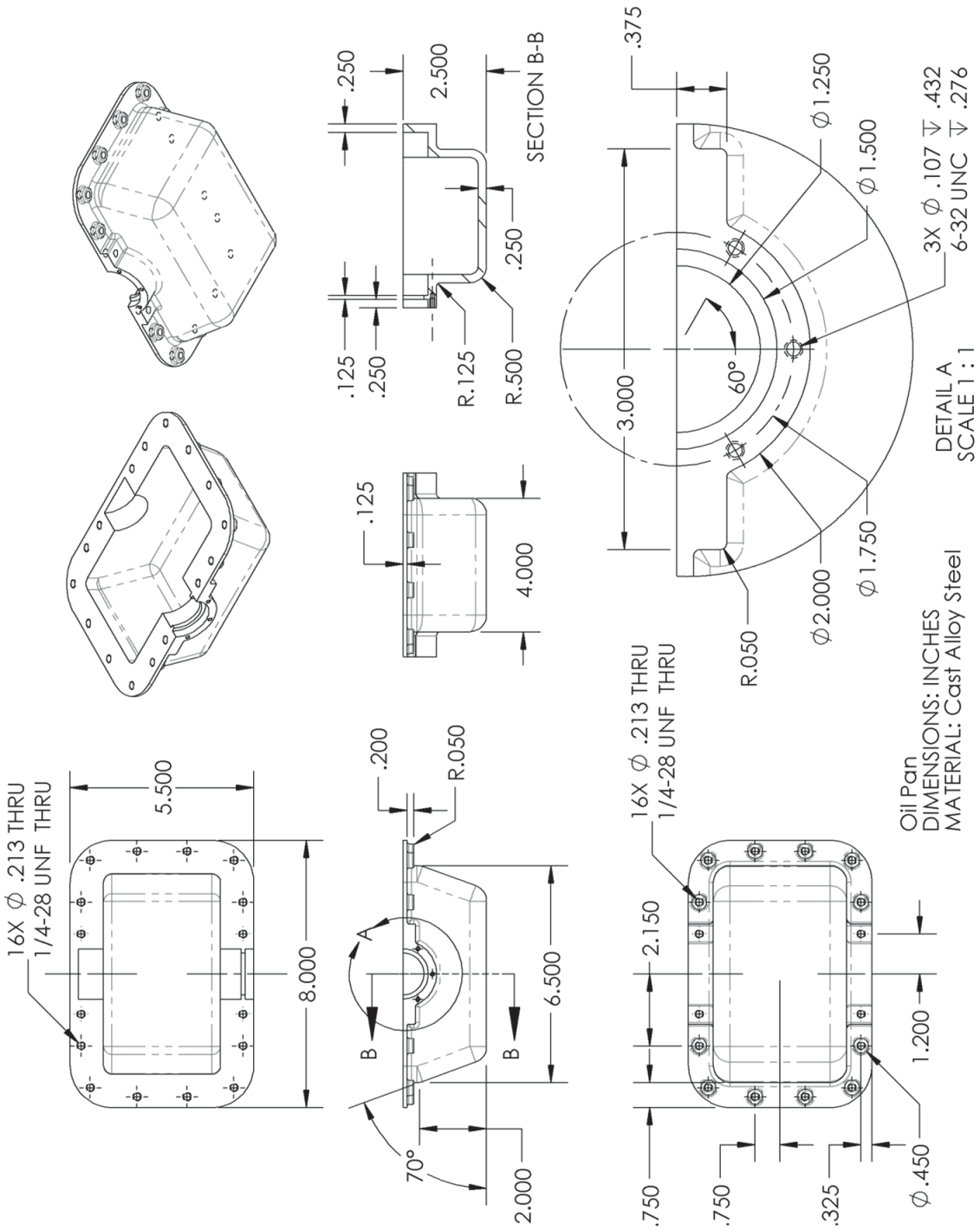
**Crank Case Top (2 Pages)**

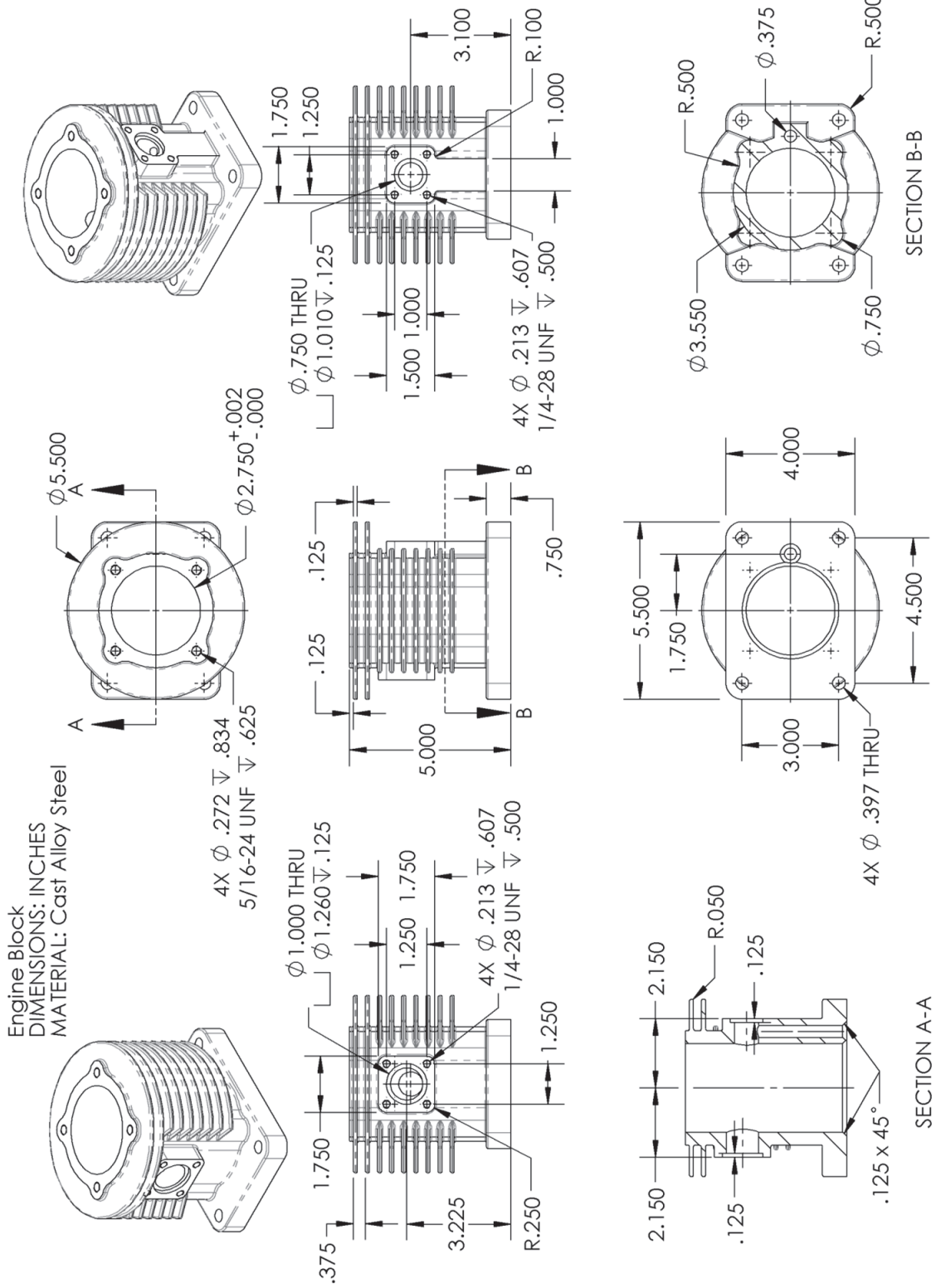




Crank Case Top  
DIMENSIONS: INCHES  
MATERIAL: Cast Alloy Steel PAGE 2 OF 2









**Extra Credit:** Visit our web site to download 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 components and after the Assembly lesson make the Gas Grill assembly.

<http://mechanicad.com/download.html>

