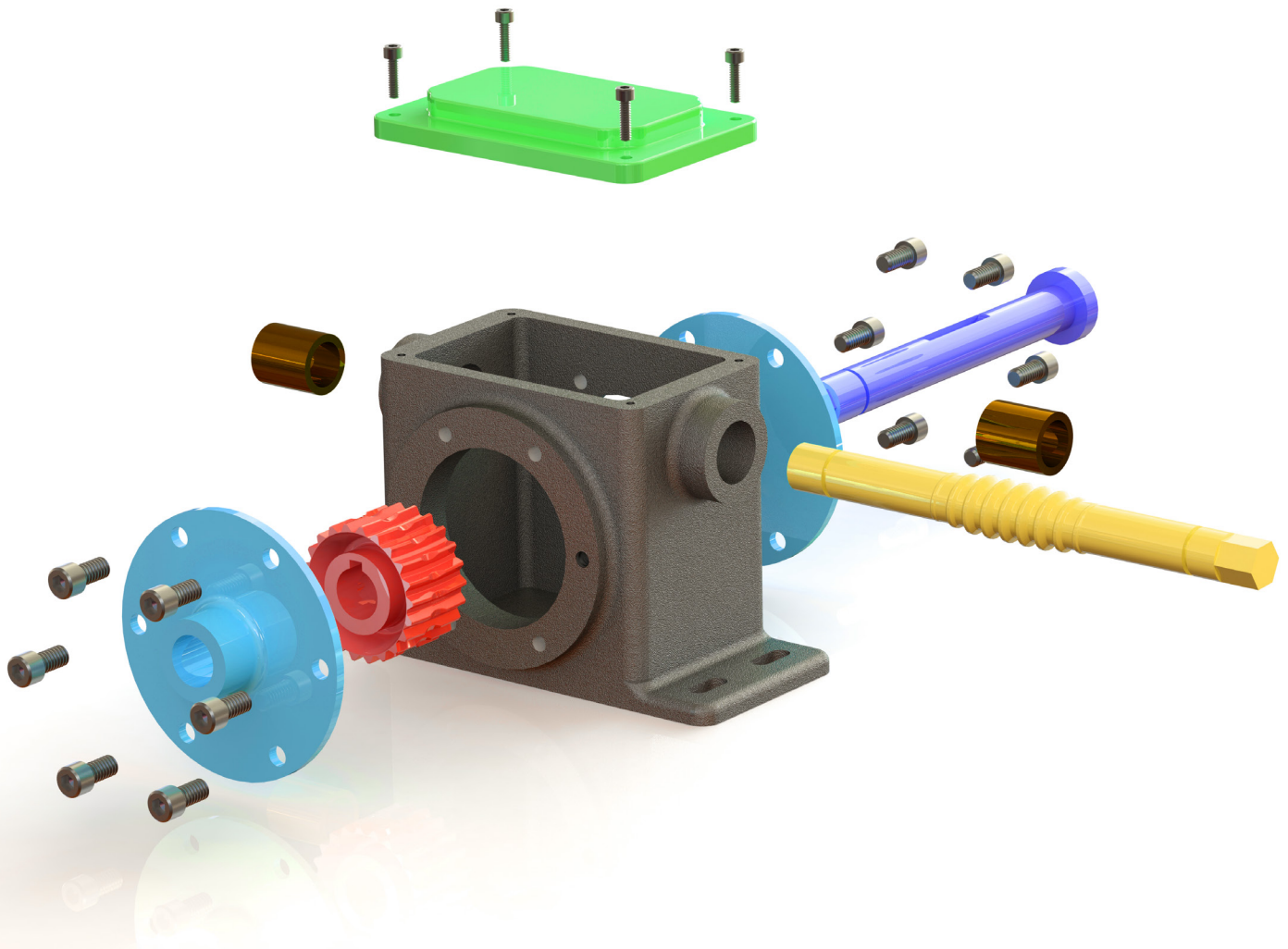# Beginner's Guide to SOLIDWORKS® 2021

Level I

Parts, Assemblies, Drawings, PhotoView 360, SimulationXpress and CSWA Preparation

Alejandro Reyes MSME, CSWE, CSWI

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Chapter 3: 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 visually attractive and pleasant to see and touch, making extensive use of curvature and organic shapes. These products include things like your remote control, a computer mouse, a coffee maker, perfume bottles, appliances, etc., and very often, their commercial success can be directly attributed to their appearance. They have to look nice, 'feel' right, and of course perform the task they were intended for. Sweeps and Lofts (also used to create "organic" shapes like those found in nature) are widely used in the automotive, consumer products and aerospace industry, where cosmetics, aerodynamics, and ergonomics are a very important aspect of the design.

Sweeps and Lofts have many different options that allow us to create anything from relatively simple to extremely complex shapes. Considering 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 commands 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 and options of the new features.

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

These are some examples of designs made using advanced modeling techniques; some of them are covered in the Level II book.
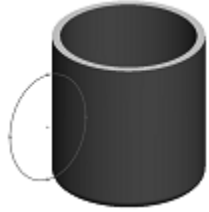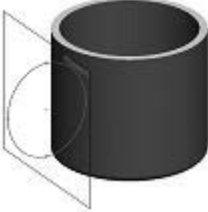
Sweep: Cup and Springs

Notes:

The Cup

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 basics of the Sweep command, a new Fillet option to create a Full Round fillet and a review of auxiliary Planes. This is the sequence of features to complete the cup:

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

8.1. - 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. Make a new part with a template based in inches. 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.)

Thin Features can be made using either an open or closed sketch but using an open sketch will always make a thin feature.

3.500

4.000

R.250

243
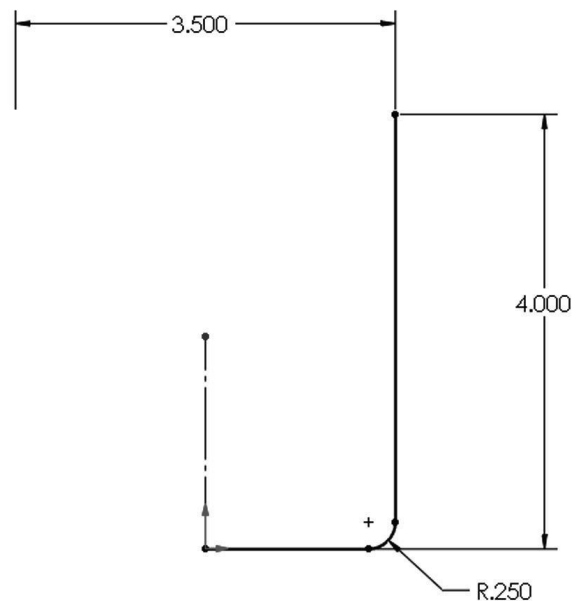
8.2. - After selecting the "**Revolve Boss/Base**" command we get a warning telling us about the sketch being open. Since we want a thin revolved feature, select "No."
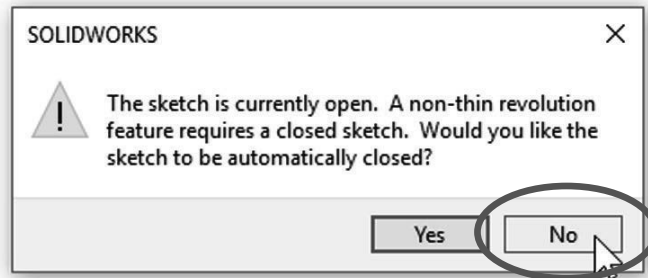
In the "Revolve" options, "**Thin Feature**" is automatically activated. Since we want the dimensions we added to be external model dimensions, select the Thin Feature's "**Reverse Direction**" option to add the material <u>inside</u> the cup. Notice the preview showing the change. Click OK to finish the first feature.
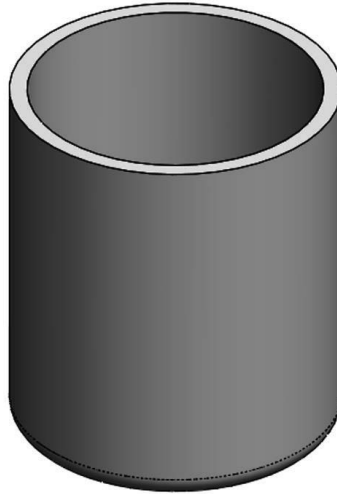
In the value box we typed 3/16; we can enter a fraction and SOLIDWORKS changes it to the corresponding decimal value when we click OK. We can also type simple mathematic expressions including addition, subtraction, multiplication, and division in any value box where we can type a value.
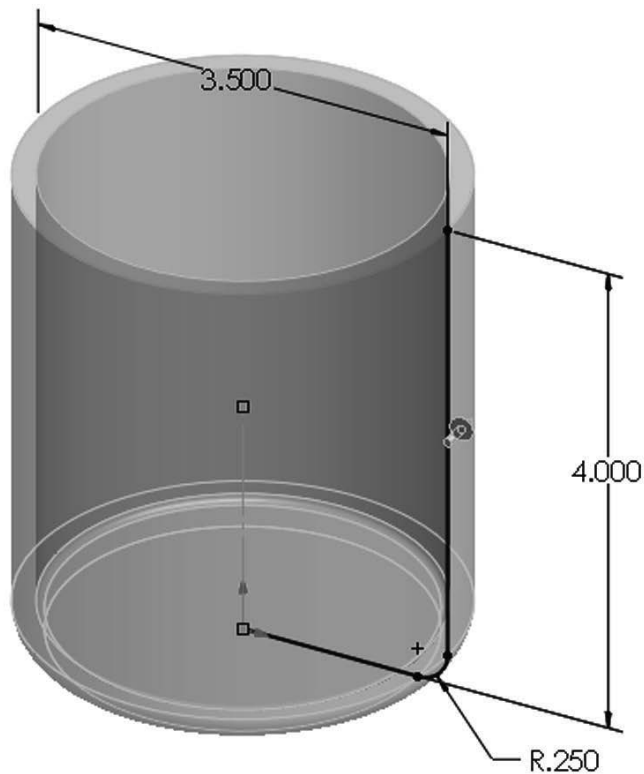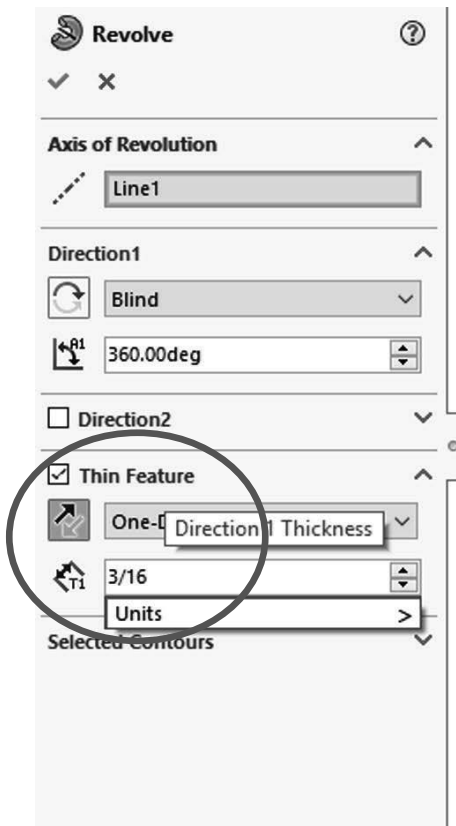
Our part looks like this and "*Revolve-Thin1*" is added to the Feature Manager.

More About Thin Features:

When we create an Extruded Boss using an open sketch, and elect to Not Close the sketch, the Thin Feature option is automatically activated. Additional options include adding the material on one side, both, or midplane, as well as automatically rounding the corners with a specified radius to the inside edges.
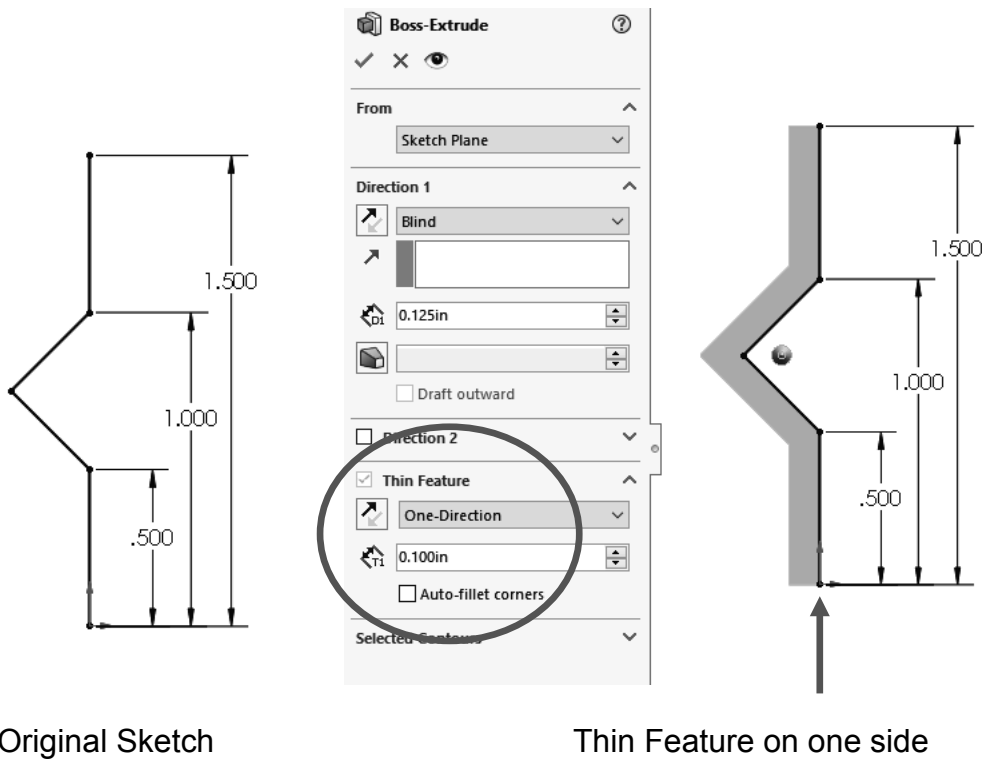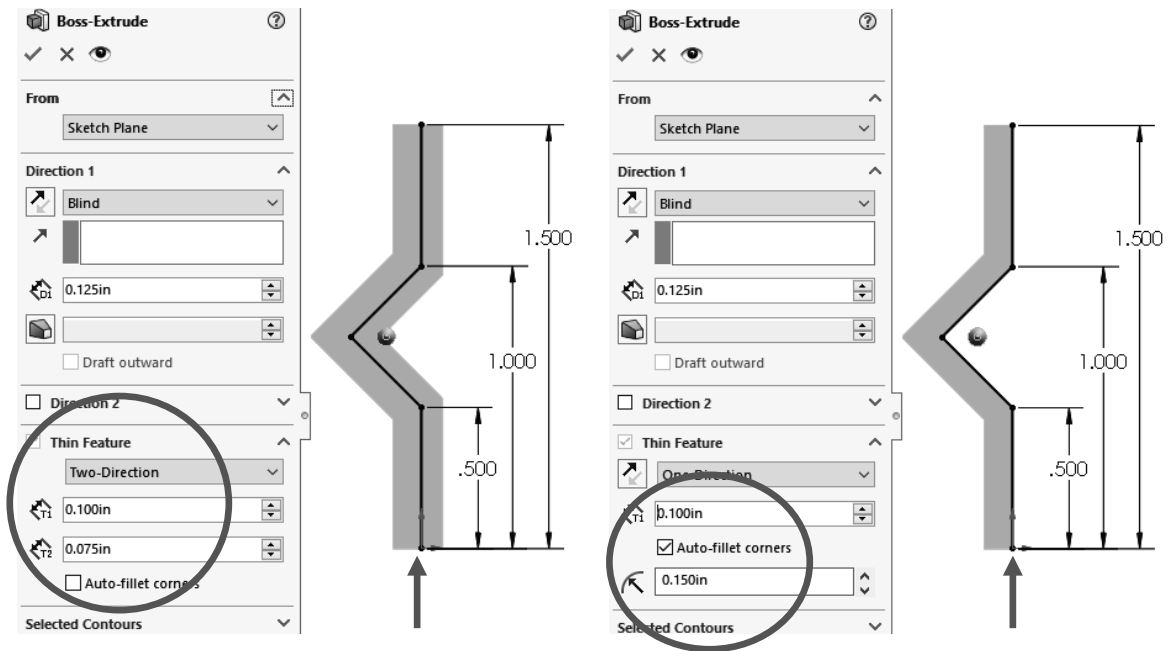

Original Sketch   Thin Feature on one side
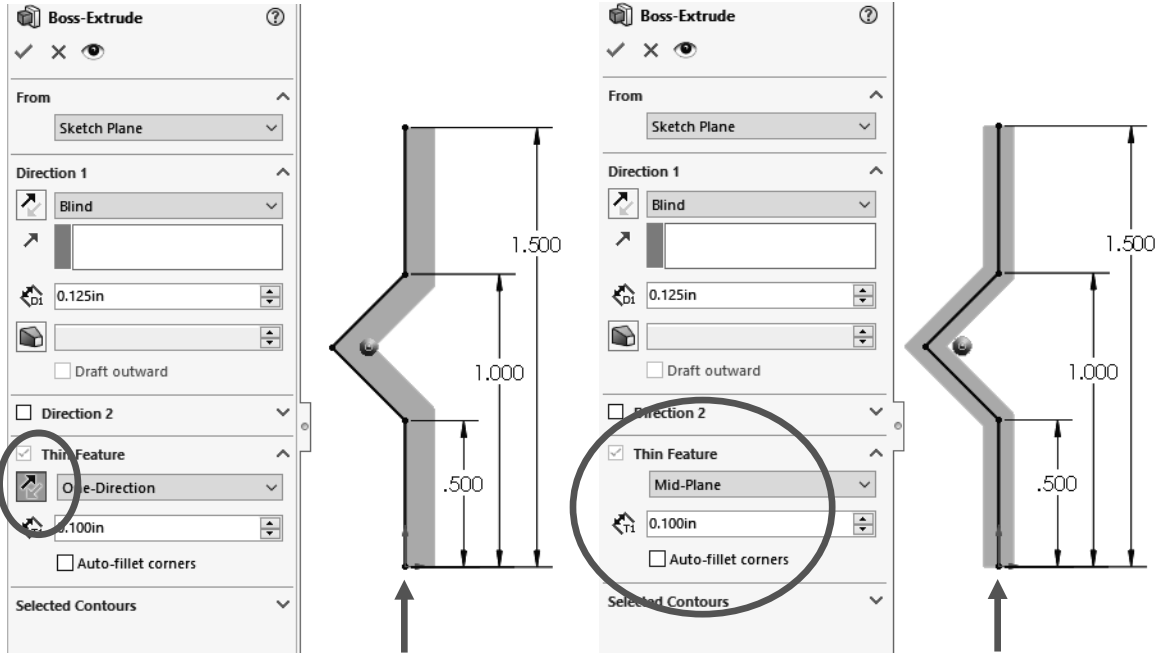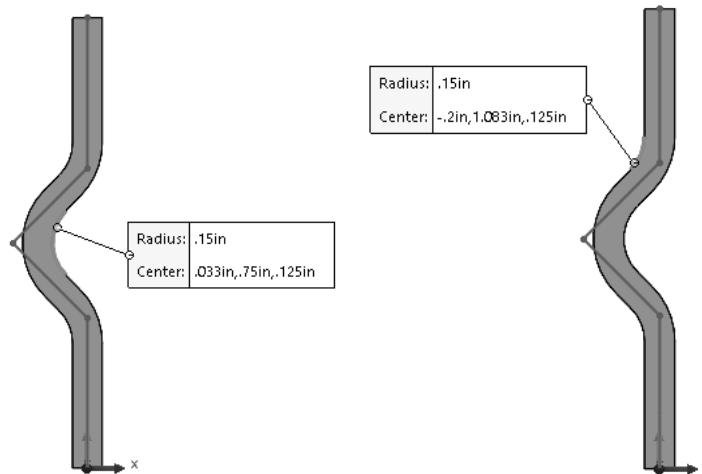
Thin Feature Reversed Direction

Mid-Plane

Two Directions (Different Thickness)

Auto-Fillet Corners (no Automatic preview)

The finished Thin Feature extrusion was made using the thin feature's Mid-Plane end condition and Auto-Fillet option. The sketch was made visible to illustrate the resulting extrusion, and the specified radius is added to the inside edges.


8.3. - Select the "*Front Plane*" and create the following sketch using an ellipse. Switch to a Front view and select the "**Ellipse**" command from the Sketch tab in the CommandManager or from the menu "**Tools, Sketch Entities, Ellipse**." To draw an ellipse, first click to locate its center, click again to locate one axis horizontally, and again to locate the vertical axis. Add the corresponding dimensions between the ellipse's points at the major and minor axes. Be sure to add a "Vertical" geometric relation between the top and bottom points of the ellipse (or "Horizontal" between left and right) to make them vertical (or horizontal) to each other to fully define the sketch.
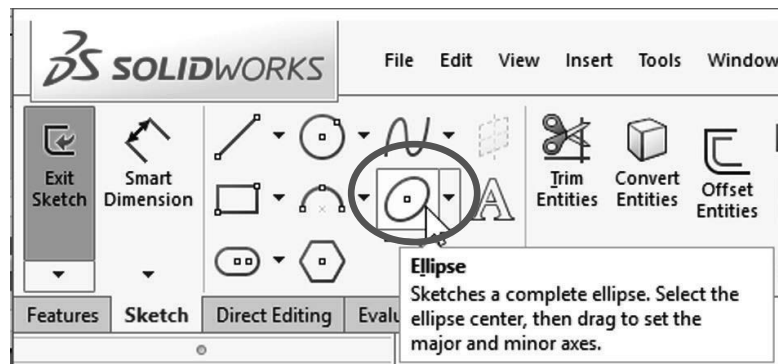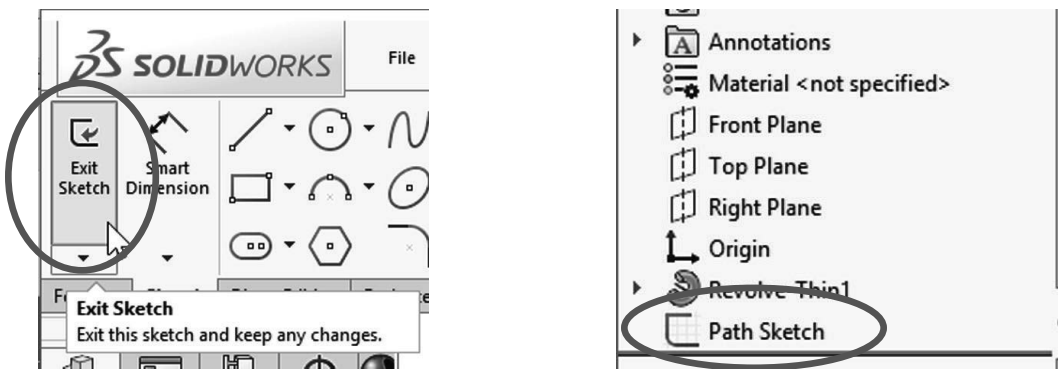

When looking at a round surface's profile from an orthogonal view we can add a dimension to its silhouette. This option will be needed to add the 1.375" dimension.
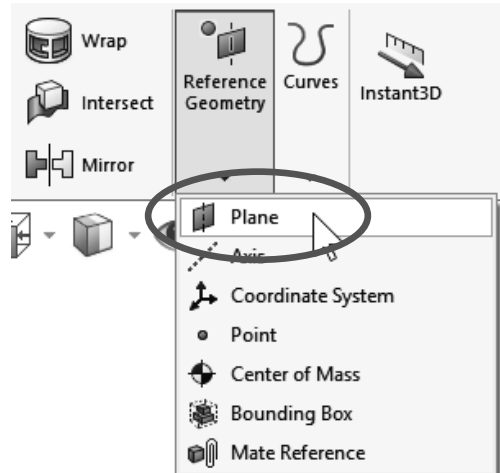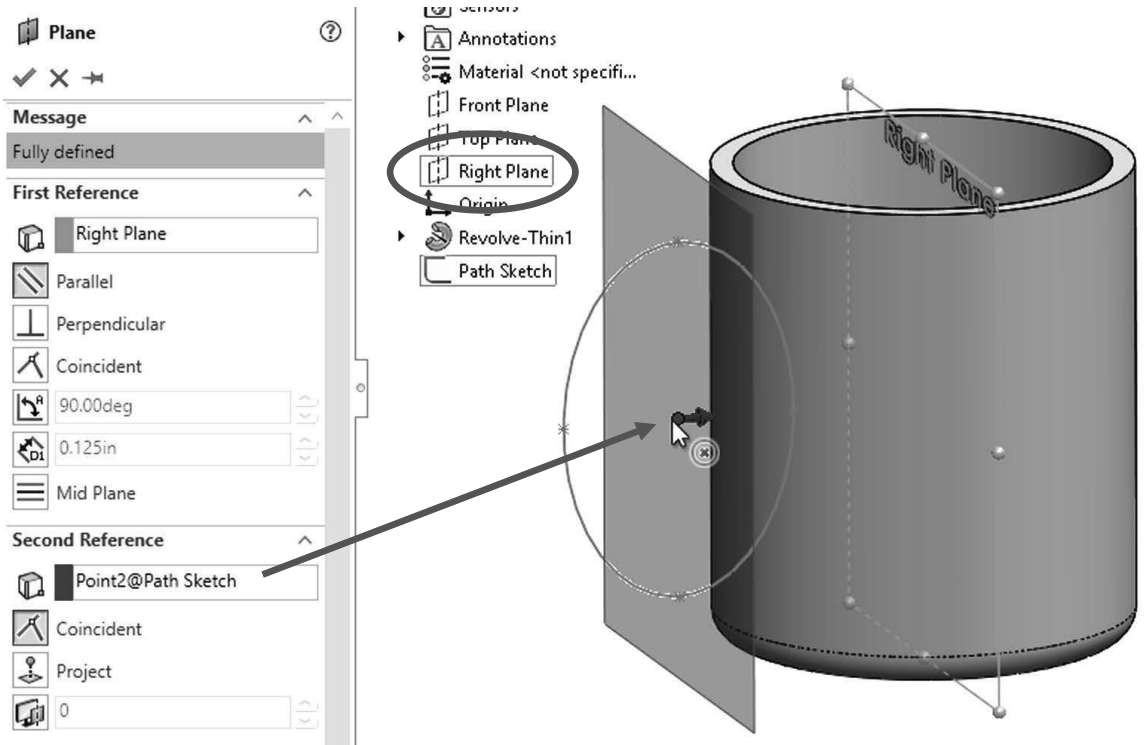

247
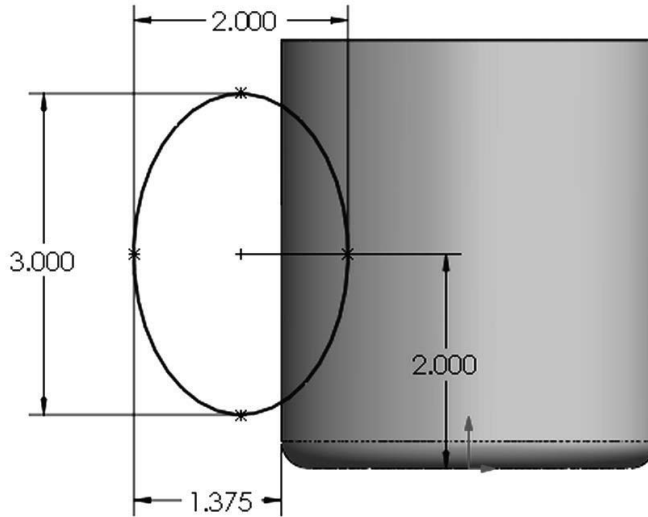
Exit the sketch and rename it *"Path Sketch."* We will not use the sketch for a feature just yet.


8.4. - Create an Auxiliary parallel plane using the "*Right Plane*" as the first reference and the center of the ellipse as the second reference as shown. Click OK to complete the plane.

8.5. - Select the plane just created and draw the next sketch in it. Looking at the plane perpendicular to it helps us better visualize the sketch. Before adding the sketch, select the plane and press the "Normal To" command (shortcut Ctrl+8). Selecting "Normal To" again will rotate the view 180°.
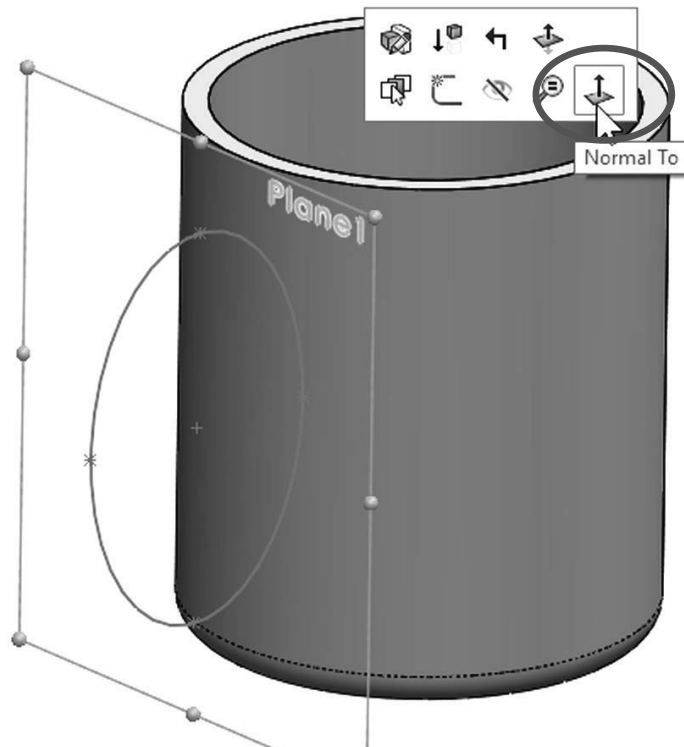
Add a new ellipse and start drawing it at the top point of the previous sketch's ellipse. Add a coincident relation to locate it if needed. Remember to add a horizontal relation between the points of the major (or vertical to the minor) axis.

If necessary, rotate the view to have a better view to select the top point of the previous sketch's ellipse. Exit the sketch and rename it "*Profile Sketch.*"
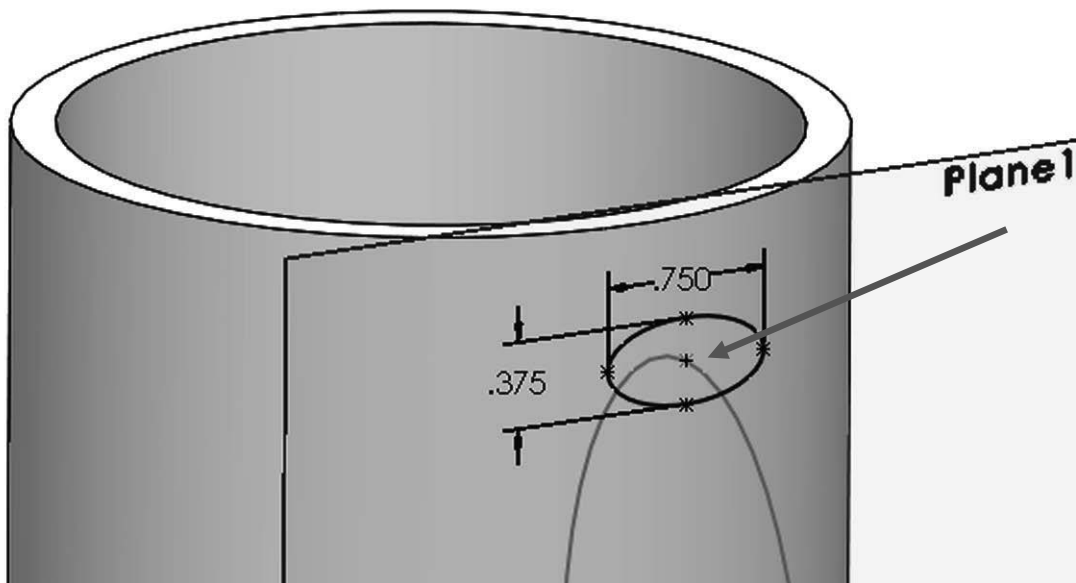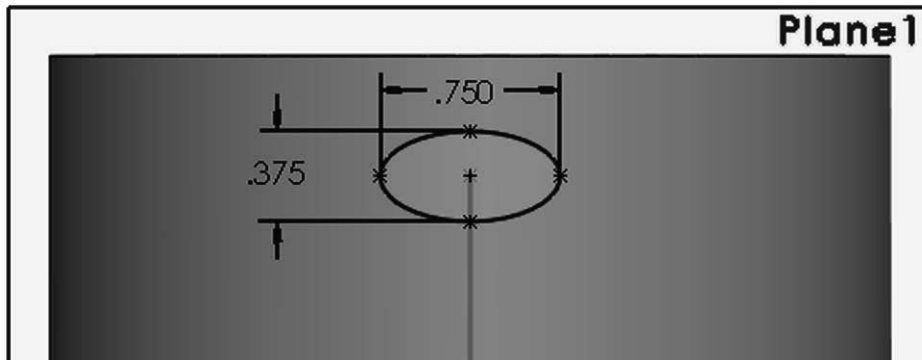
8.6. - Select the "**Sweep**" icon from the Features tab in the CommandManager or from the menu "**Insert, Boss/Base, Sweep**." The sweep is a feature that requires a minimum of two sketches: one for the sweep's profile and one for the path. (For the path of the sweep instead of a sketch we can also use a model edge or a user defined curve that crosses the profile.)
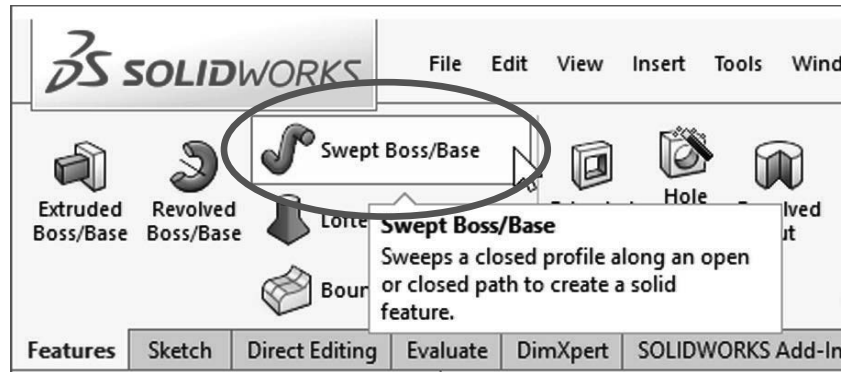
In the "**Sweep**" properties, select the *"Profile Sketch"* in the Profile selection box, and the "*Path Sketch*" in the Path selection box. Optionally, a Sweep can have guide curves and other parameters to better control the resulting shape; in this case we are making a simple sweep feature. Notice the preview and click OK when done to finish the sweep. Hide the auxiliary plane using the "**Hide/Show Items**" toolbar if so desired.
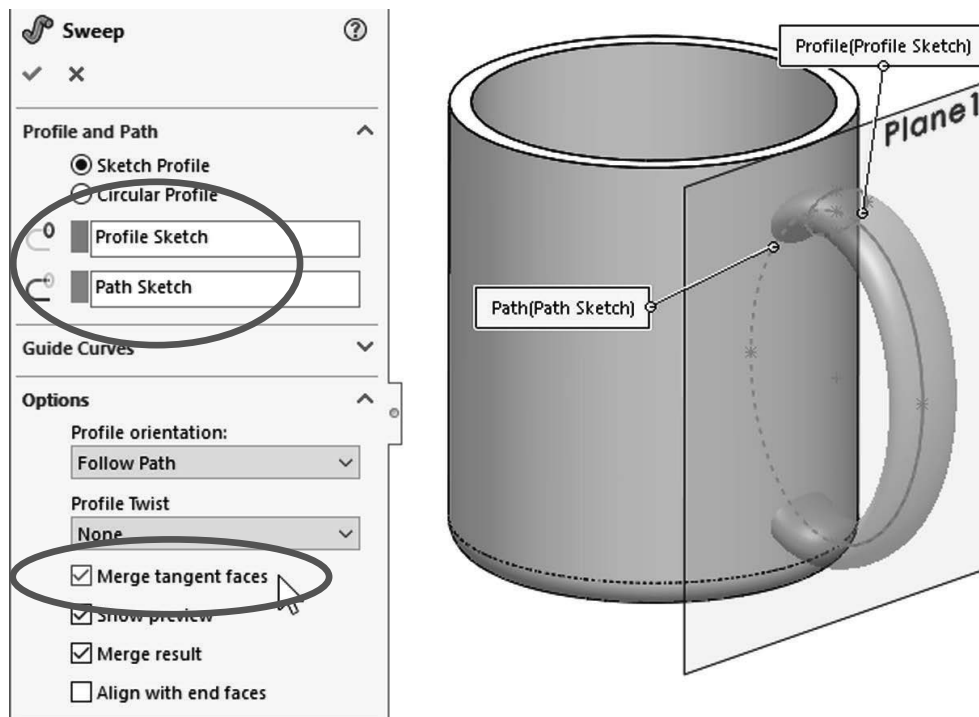
Turn on the "**Merge Tangent Faces**" checkbox under "Options" to have a single, continuous sweep surface by merging tangent faces.

251

8.7. - Notice the sweep also goes inside the cup. To fix this we'll make a cut. Create a sketch in the flat face at the top of the cup.
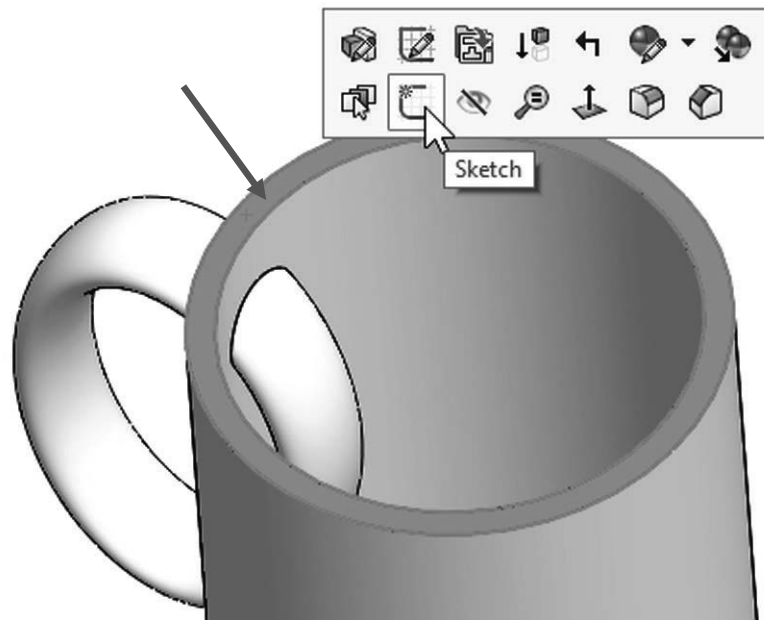

Select the <u>inside</u> edge at the top and use "**Convert Entities**" from the Sketch tab to convert the edge to sketch geometry. Click OK to continue.
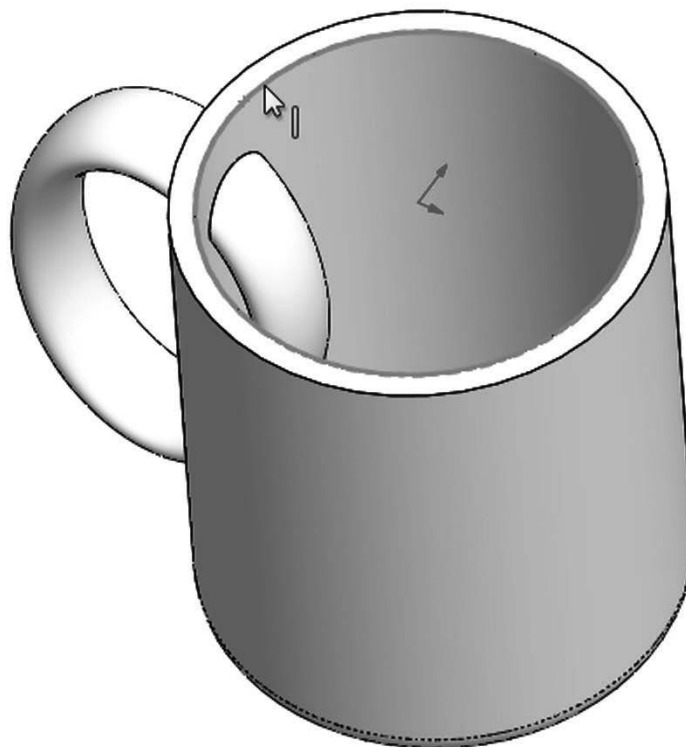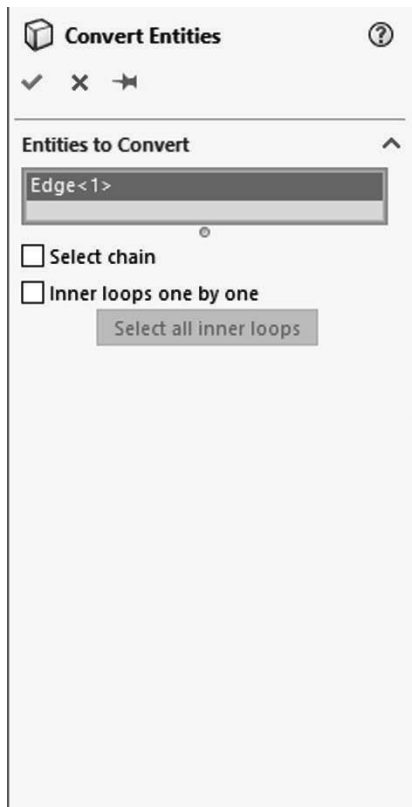
8.8. - Select the "**Extruded Cut**" icon and use the "Up to next" end condition from the drop down selection list. This end condition will make the cut until it finds the next face (the bottom) effectively cutting the part of the handle inside the cup.
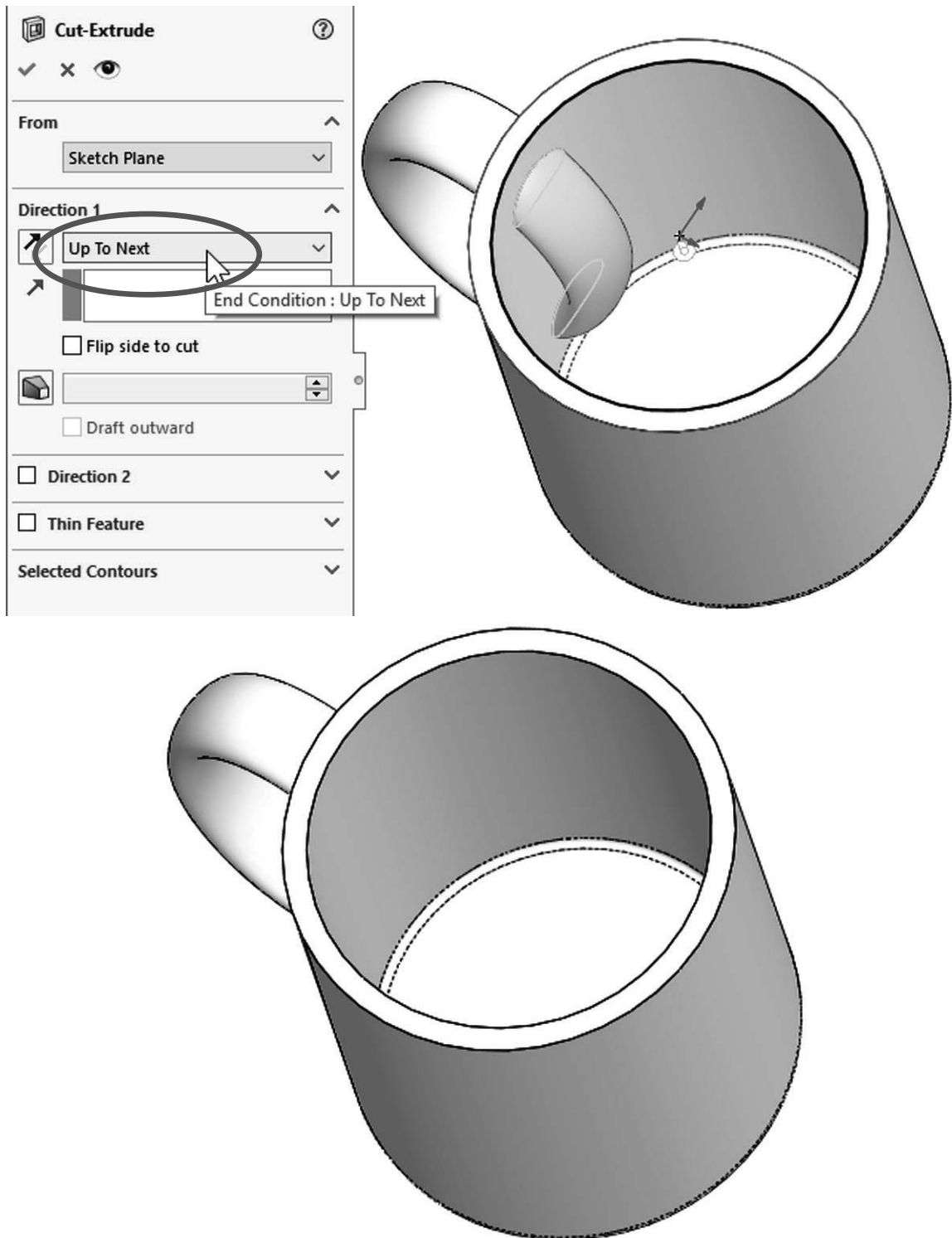
8.9. - Now we need to round the flat face at the top lip of the cup. To do this we'll add a "**Fillet**" using the "**Full Round Fillet**" option. The full round will essentially remove the flat face at the top (the middle face) and replace it with a rounded face tangent to the start, top, and end faces.

Before   After
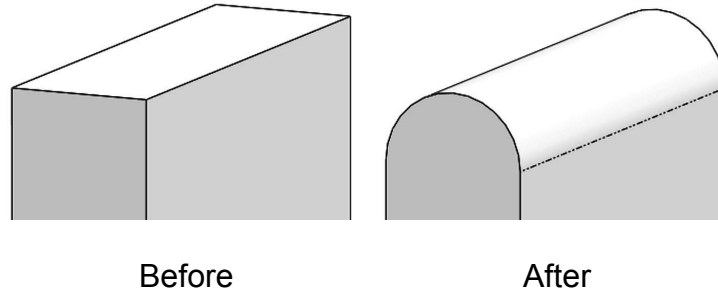
We need to select three faces; the middle face will be the one replaced by the fillet. After selecting the Fillet command, select the "Full round fillet" option to reveal the selection boxes. With the first selection box active select the outside face of the cup. Click inside the second selection box 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.
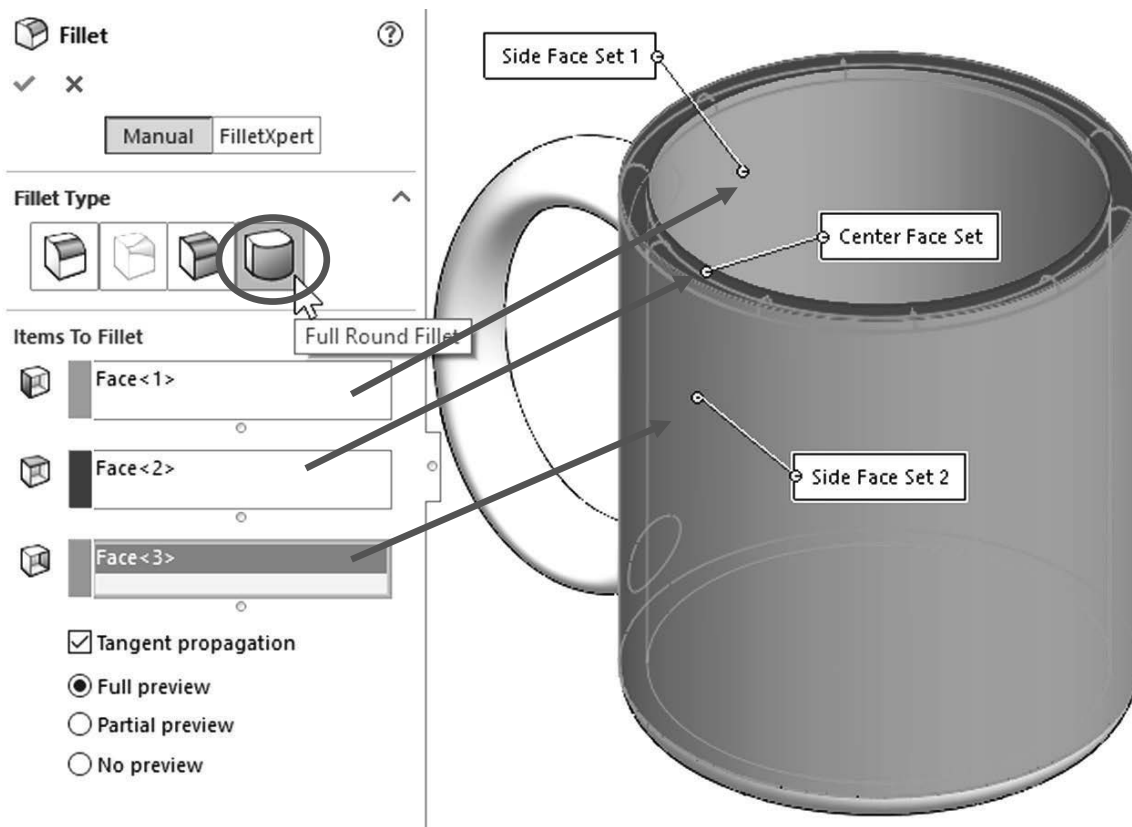
8.10. - To finish the cup, add a fillet to round the edges where the handle meets the cup. Select the fillet command with the "Constant Radius" option, select the handle's surface, and change the radius to 0.25″. Click OK to finish.
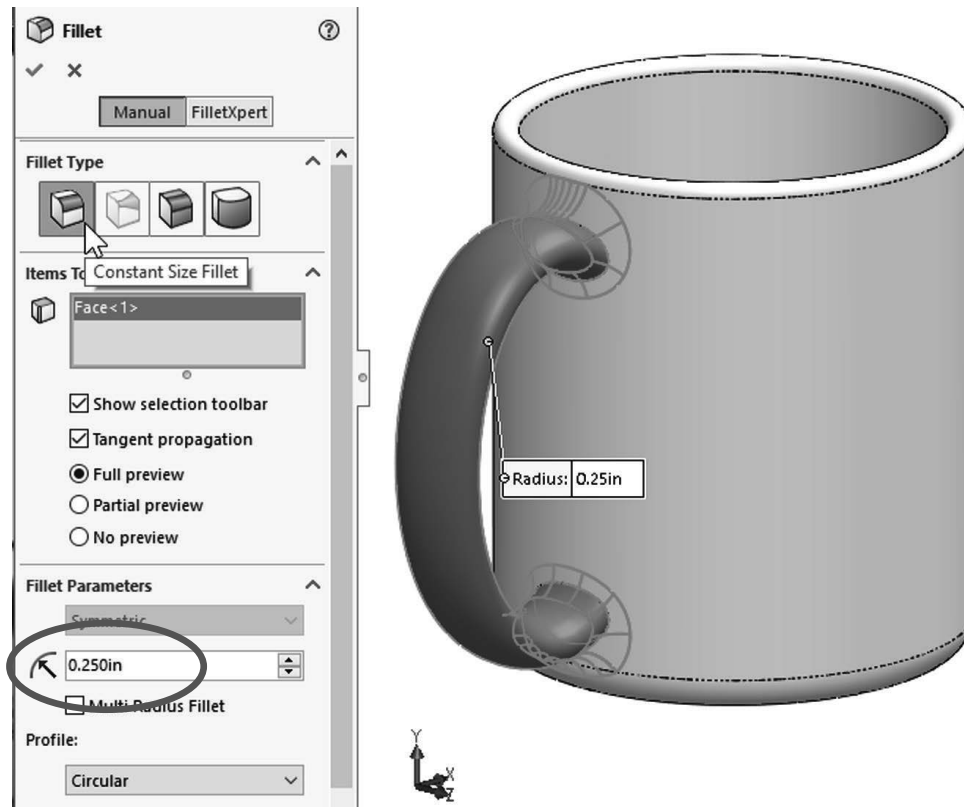
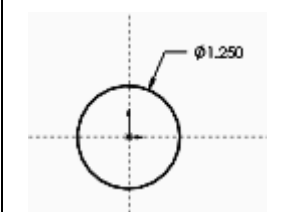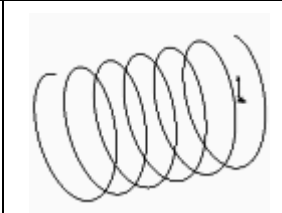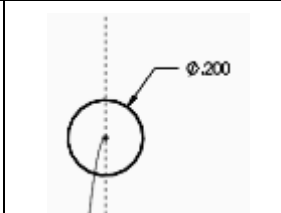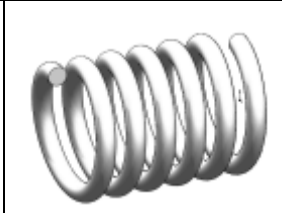**8.11. –** Set the material as "Ceramic Porcelain" and save the finished part as *'Cup'*.
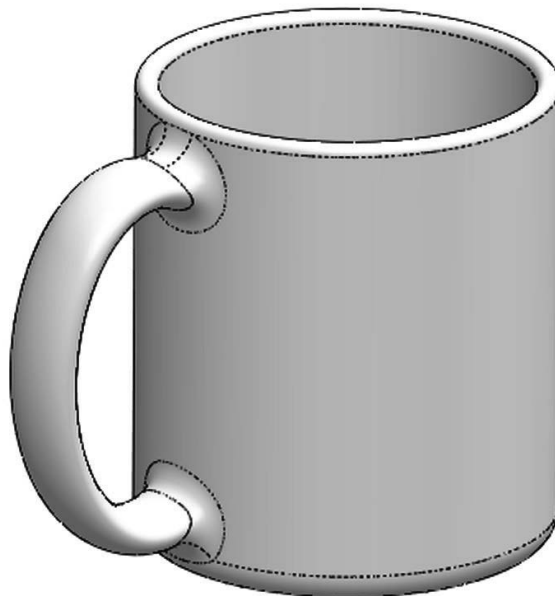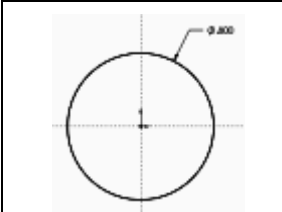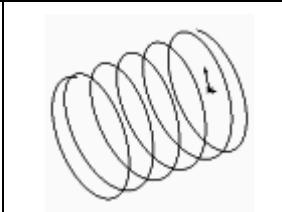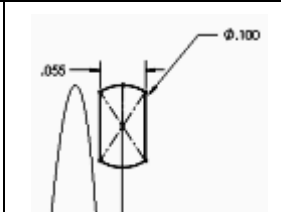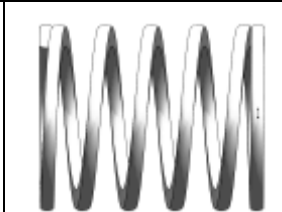
Notes:

Simple Spring

In the next exercise we are going to show how to make a simple and a variable pitch spring. To make these springs we'll have to learn how to make a simple and a variable pitch helix to be used as a sweep path. This is the sequence of features to complete the springs:

Simple Spring

|			
Draw circle	Make Helix	Draw Spring Profile Sketch	Make Sweep

Variable Pitch Spring

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

9.1.- First Make a new part in inches. To make a helix, we need to make a sketch with a circle. This circle is going to define the helix's diameter. Select the "*Front Plane*" and make a sketch using the following dimensions. Exit the sketch when done.
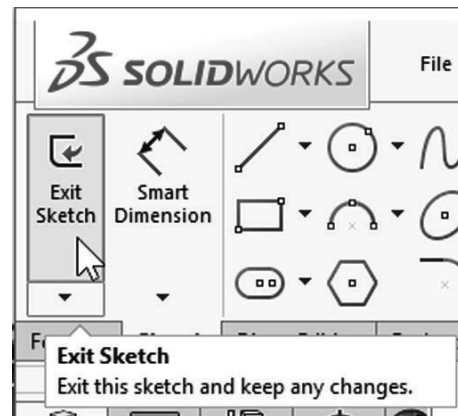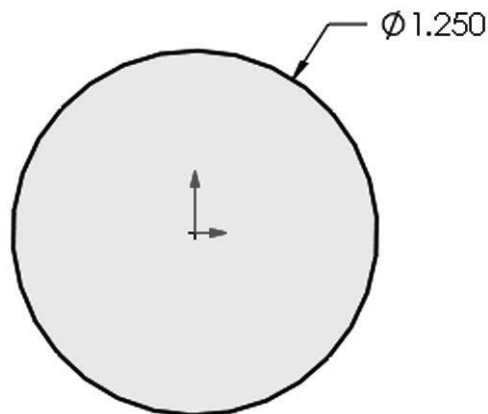
9.2. - In the Features tab select "**Curves**, **Helix and Spiral**" from the drop-down icon or the menu "**Insert, Curve, Helix/Spiral**." If asked to select a plane or a sketch, select the sketch we just drew.
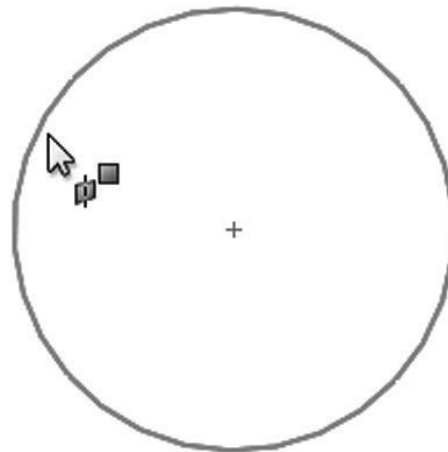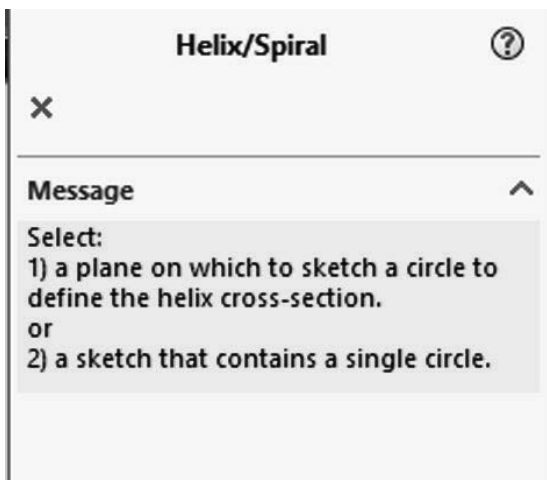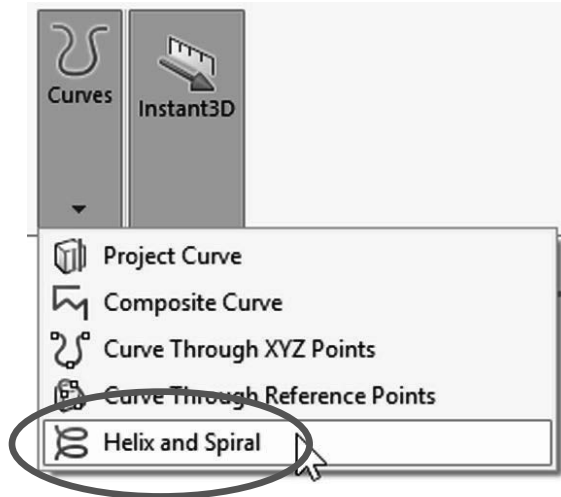
If we don't exit the sketch, we'll only have the "Helix and Spiral" option from the "Curves" command. If we select the "**Helix and Spiral**" before exiting the sketch, the sketch will be automatically selected.

The helix is defined by selecting two parameters between Pitch, Revolutions, and Height; the third parameter is automatically calculated from the other two. For this example, we'll select "Pitch and Revolution" from the "Defined By:" drop down menu and make the pitch 0.325" with 6 Revolutions. The "Start Angle" value defines where the helix will start. By setting it to 90 degrees we are defining the start of the helix 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.) Click OK to complete the helix.
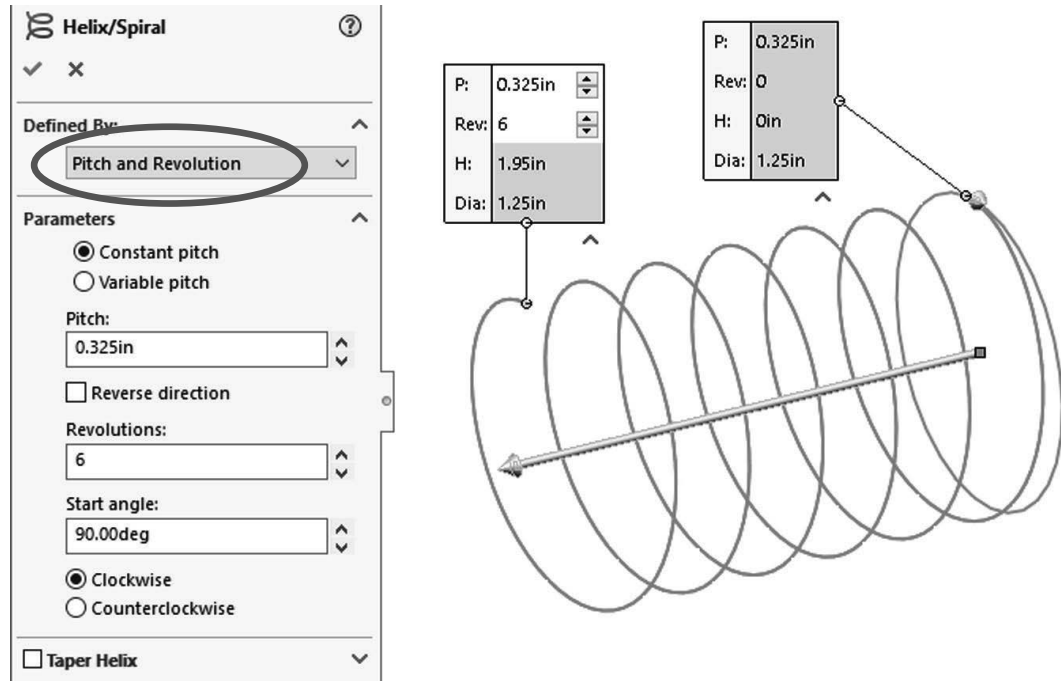
Note the Helix command has options to make it Counterclockwise, Clockwise, Tapered, Variable pitch and reversed (going right or left).

9.3. - 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…
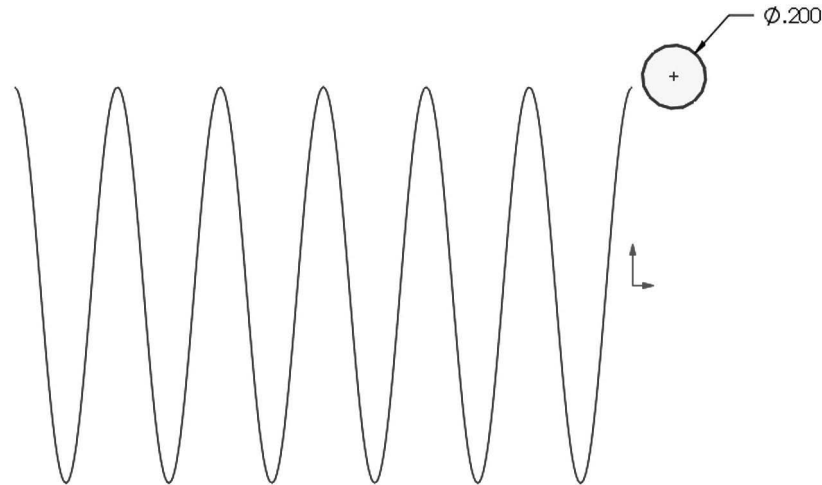

… and add a "**Pierce**" geometric relation between the center of the circle and the helix. This way the profile will be located at the beginning of the helix. This relation will make the sketch fully defined. Exit the profile sketch when done.
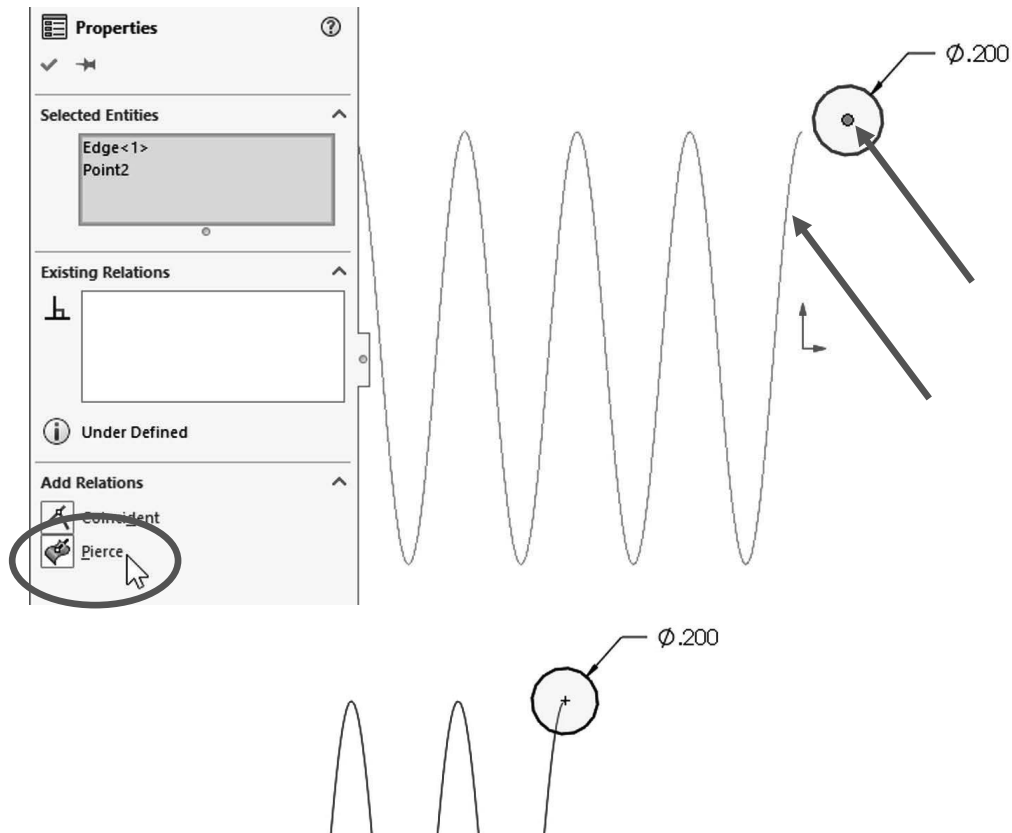
A pierce relation is done between an element that is oblique or perpendicular to the sketch plane and a point (or endpoint) in the sketch. Think of it as a needle piercing through a fabric, the sketch being the fabric and the helix (or curve, model edge or another sketch element) the needle.

9.4. - 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.
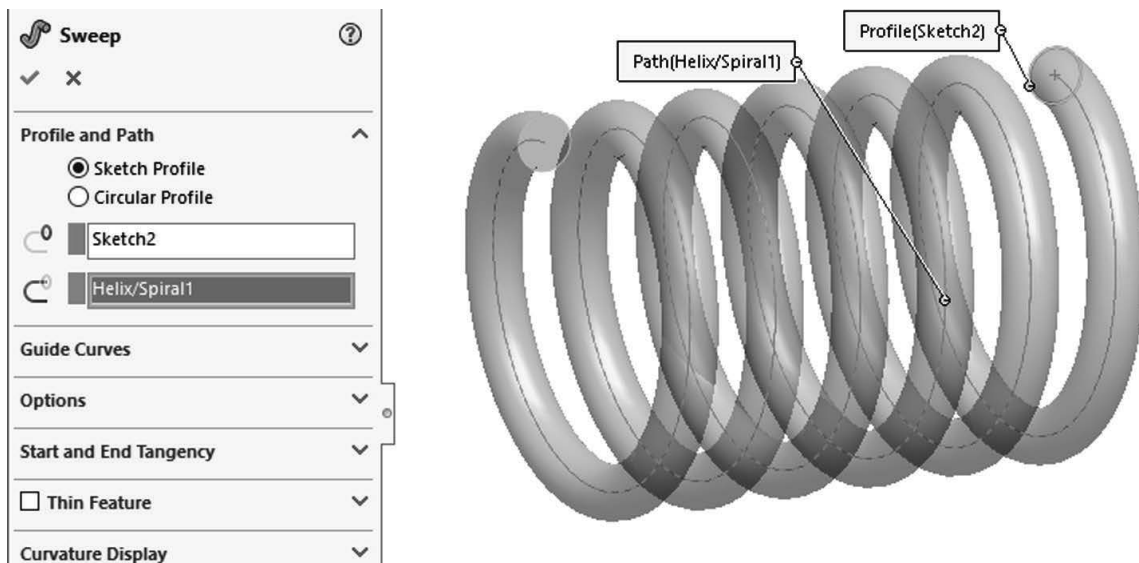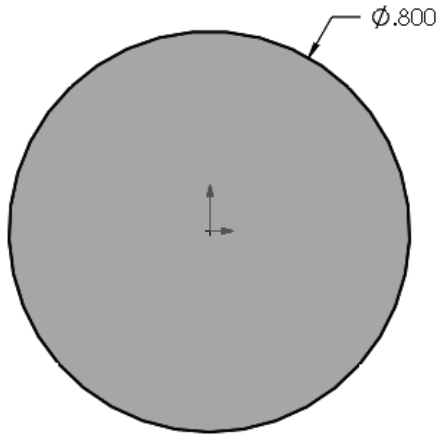
If needed/desired, hide the "*Helix/Spiral1*" in the FeatureManager, save the part as '*Spring*' and close the file.
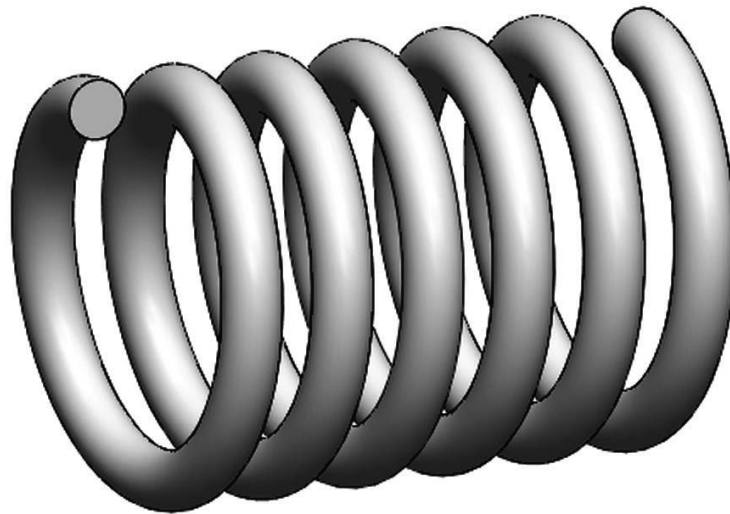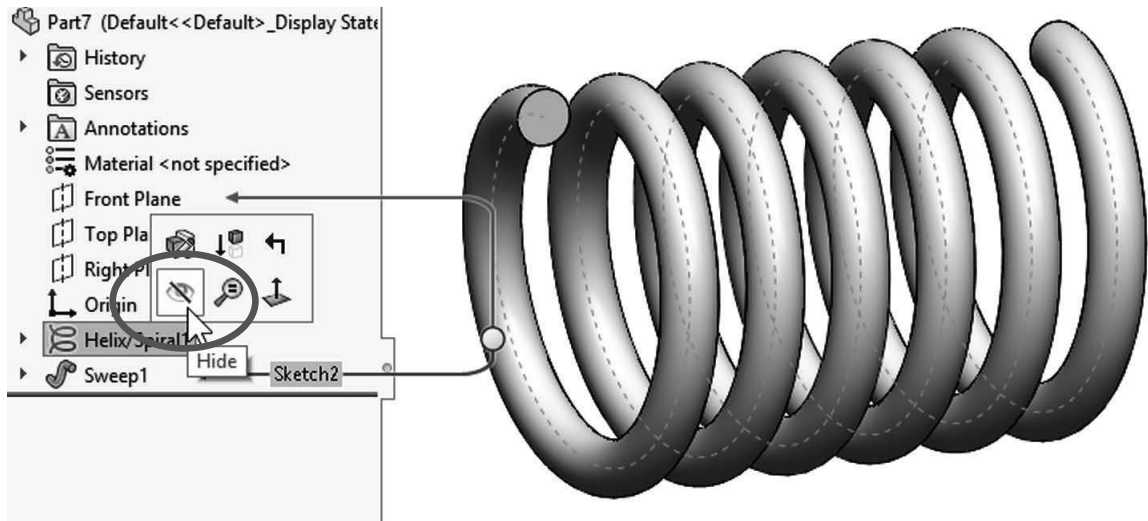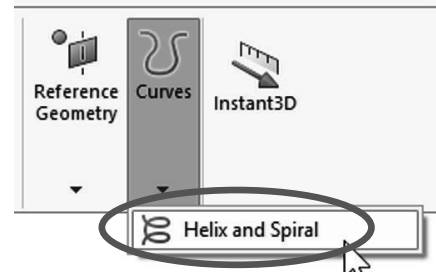
Notes:

Variable Pitch Spring

10.1. - 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.


10.2. - 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.


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 using the next table. Click OK when finished to build the helix.

	P	Rev	H	Dia
1	0.01in	0	0in	0.8in
2	0.01in	0.5	0.005i	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	0.915i	0.8in
7				0.8in

10.3. - 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. Locate the sketch to either side of the helix.
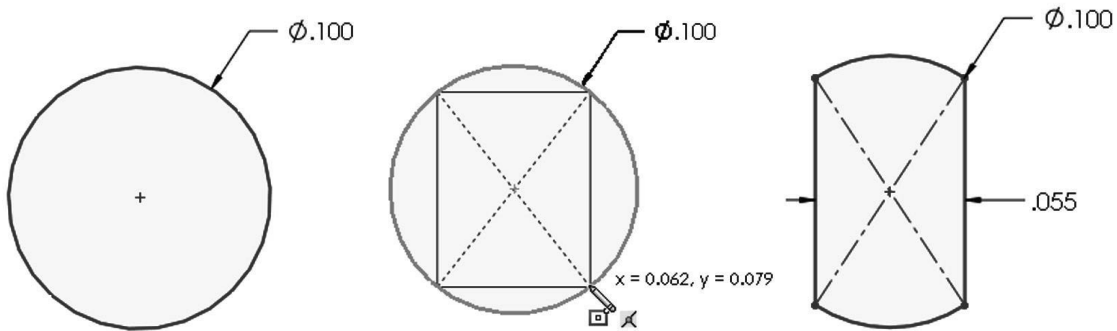
Add a centerline from the center to the top line (make sure it is coincident). Add a "**Pierce**" geometric relation between the top endpoint of the centerline and the helix to fully define the sketch. Exit the sketch and optionally rename it *'Profile'*.
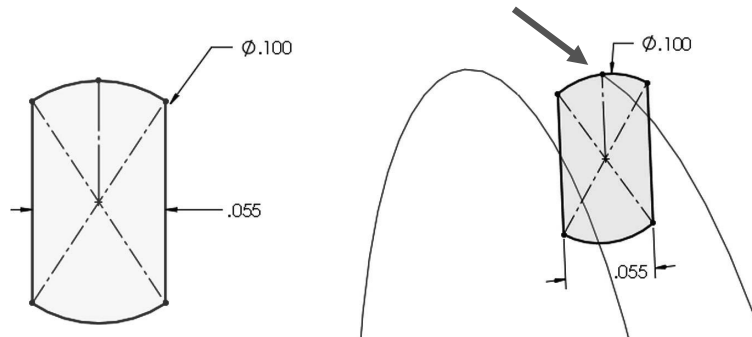
The Pierce relation allows us to fix the profile to the path and can be added to any part of the sketch. We chose to add it to the top because the original sketch used for the helix is the spring's outside diameter.

10.4. - Just as we did before, select the **"Sweep"** command, add the path and profile as shown and click OK to finish and hide the "*Helix/Spiral1*" feature.
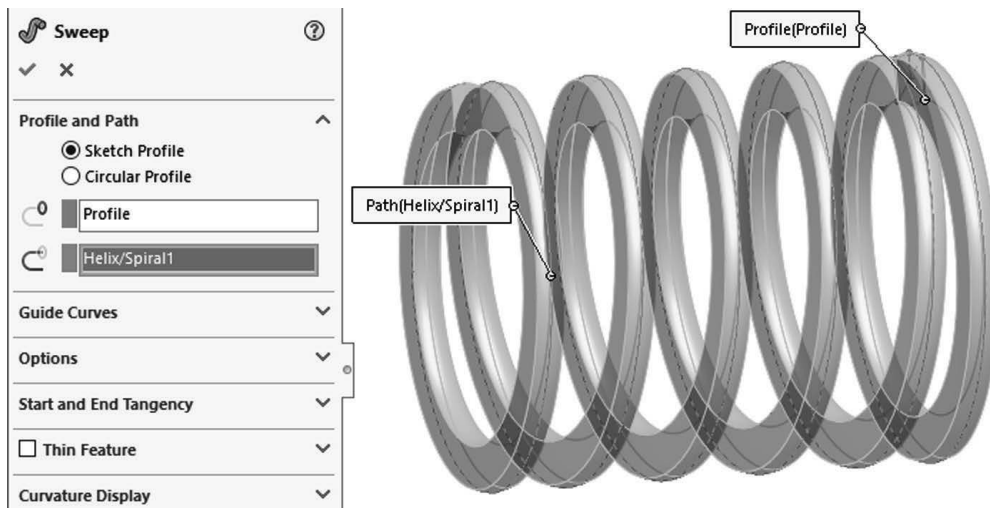
10.5. - As a finishing touch we'll make a cut to flatten the sides of the spring. Change to a Right view and add a sketch in the "*Right Plane*." Draw a single line starting at the midpoint of the indicated edge and long enough to cross the part.
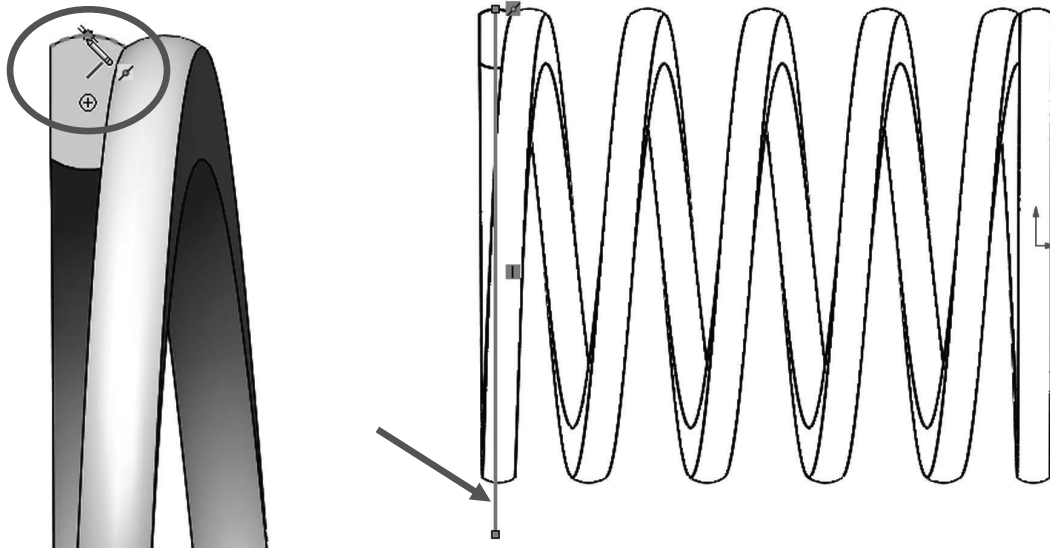

10.6. - Select the "**Extruded Cut**" command; if we use an open sketch to make a cut, selecting either the "Through All" or the "Through All - Both" option, it automatically activates the "Direction 2" checkbox with the "Through All" end condition and one side of the model is cut using the open sketch. The small arrow located at the center of the line indicates which side of the model will be cut. Use the "Flip side to cut" option if needed to cut the left side. Click OK to complete the cut and repeat the same sketch to cut the right side. Note the cutting plane is shown in the graphics area.
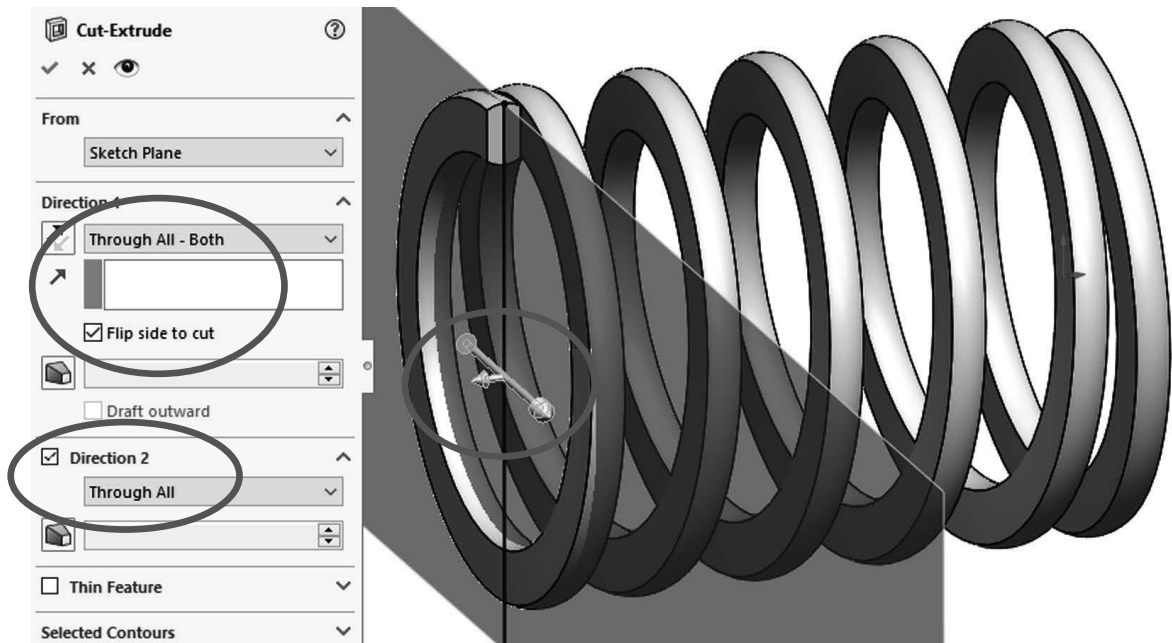
When using an open sketch to cut a model, we can only use a single open profile. If we have multiple open profiles, we cannot make the Cut Extrude using these options; we would get multiple thin cuts with a different behavior.

**10.7. -** Save the model as *'Variable Pitch Spring'*.
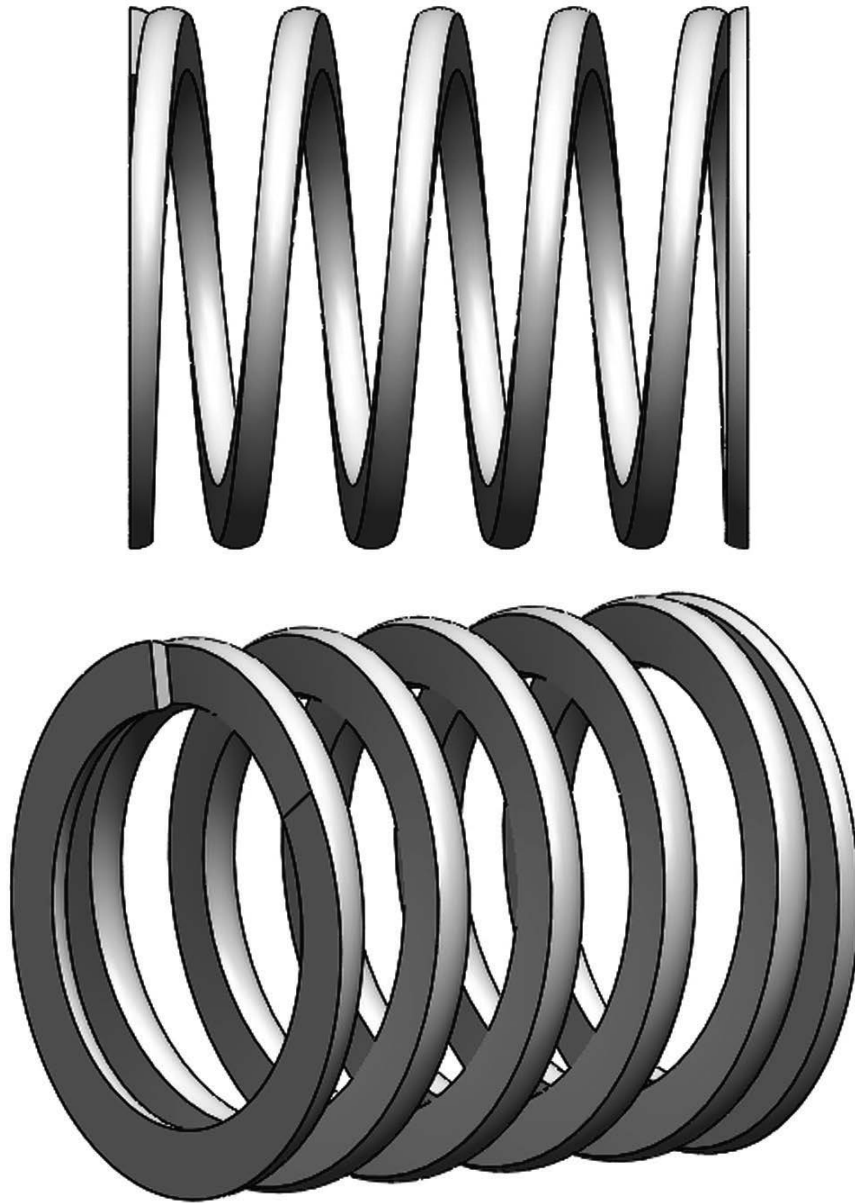
Thread Feature

Threads can be added to a model using a Sweep Boss or a Sweep Cut with a helix, and defining every thread parameter, or simplify the process using the "**Thread**" tool. 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 using a Revolved Boss/Cut or a Cosmetic Thread is usually enough. It is strongly advised to only add threads when required by the model, as in the bottle exercise later in this lesson.

11.1. - To learn how to use the "**Thread**" command open the part '*Threads*' from the accompanying files. This part has two cylinders; the first one (Blue) is the size of a ¼-20 screw's <u>minor</u> diameter, and the second (Green) is the size of a ¼-20 screw's <u>major</u> diameter. The hole (Pink) is the size needed to make a ¼-20 tapped hole.

1/4-20 Thread
<u>Minor</u> Diameter

1/4-20 Thread
<u>Mayor</u> Diameter

1/4-20 Tapped Hole
<u>Minor</u> Diameter
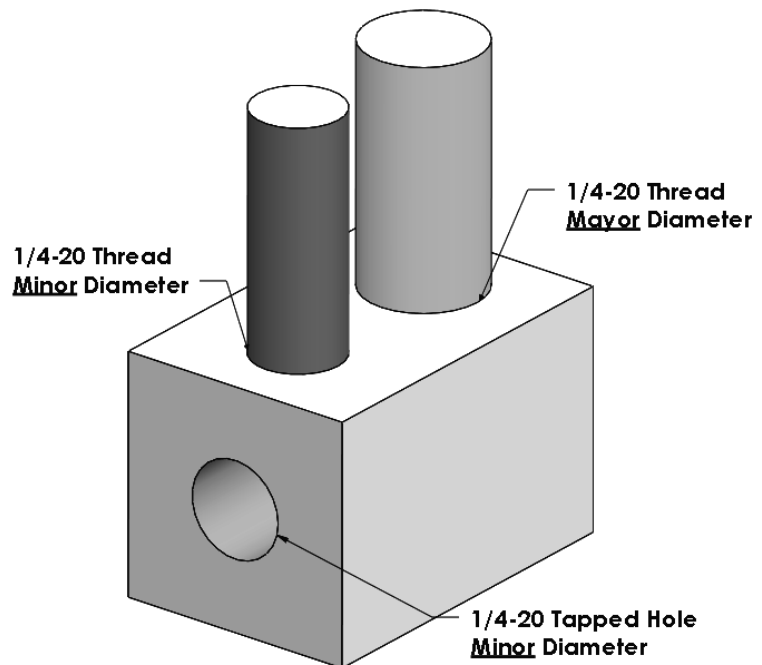
The "**Thread**" command is located in the menu "**Insert, Features, Thread.**" This command allows us to select a standard thread size and apply it either as an external thread (male) to a cylindrical boss or an internal thread (female) to a hole.
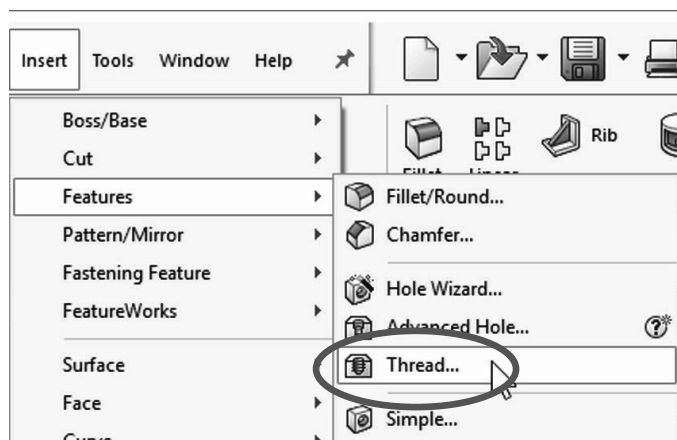
There are two ways to add an external (male) thread: The first way is to add (**extrude**) the thread in a cylinder the size of the thread's *minor* diameter. The second way is to start with a cylinder the same size as the screw's *major* diameter and **cut** away the thread.

267
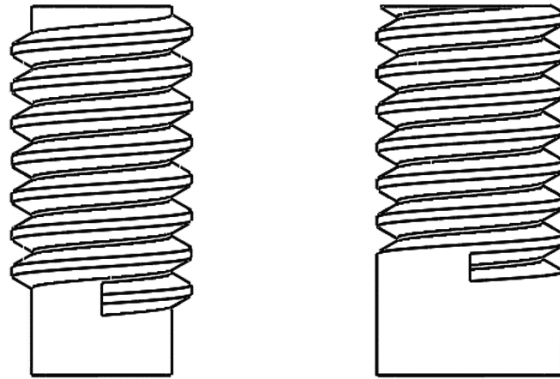
Extrude - Thread added
to Minor Diameter

Cut -Thread removed
from Major Diameter

11.2. - Select the Blue cylinder's top edge to add the thread. In this case we'll add the thread using the following settings; click OK when done to finish the thread feature:

- **Thread Location** is to define the edge to start the thread. Select the top edge of the blue cylinder's face.
- **End Condition**: The thread's length, number of revolutions or up to surface. Set to "Blind" and define its length to 0.375".
- **Specification** is the standard and size of the thread. Select "Inch Tap" to add a thread, and "0.2500-20" to define the thread size as ¼-20.
- In **Thread Method** use the "Extrude thread" to add the thread on top of the cylindrical face.
- In **Thread Options** select Right-hand thread.
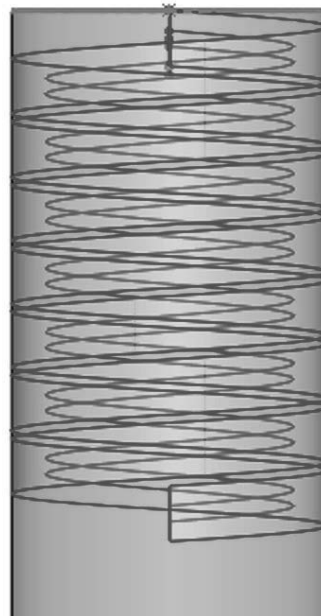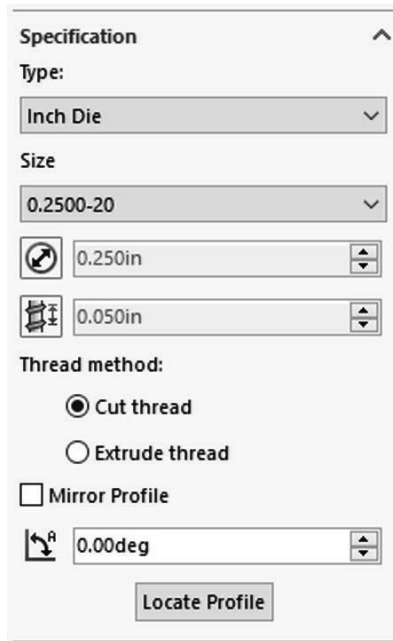
Our first finished thread looks like this:
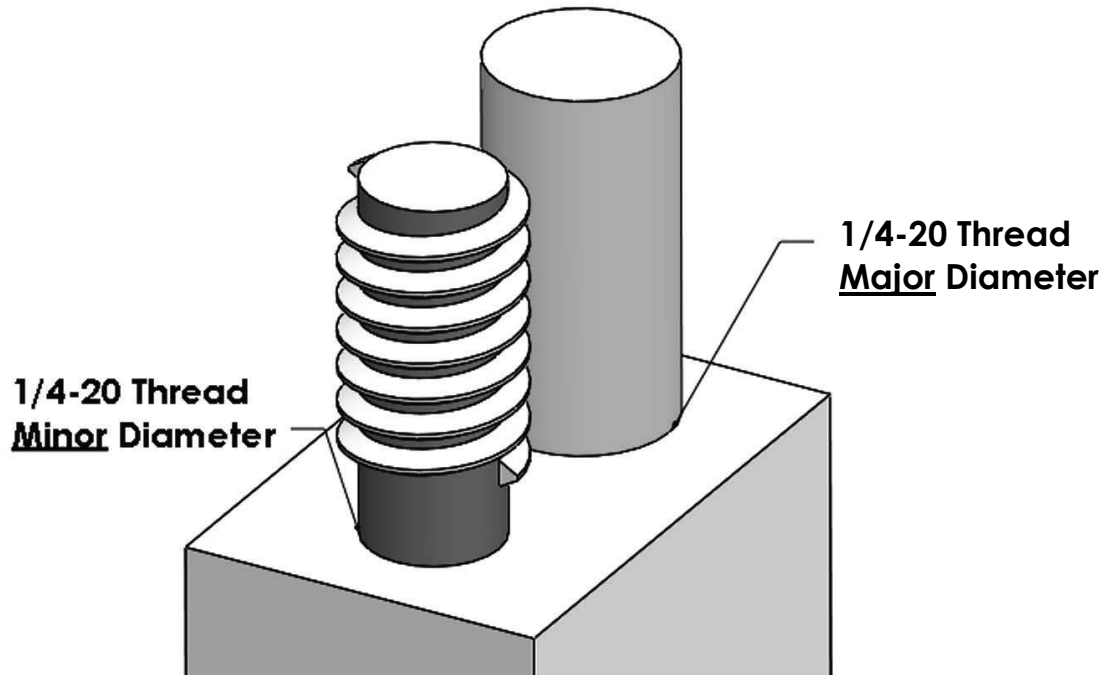
11.3. - To illustrate the second way to add an external thread, in this step the thread will be added to the green cylinder using the same settings as the previous one, except under "Specification" we'll use the "Inch Die" and "Cut thread" options to cut into the cylinder instead of adding material.

Another option that will be changed in the second thread is to turn on the "Offset" option under "Thread Location." The reason to turn this option ON is to have a thread that starts to cut the cylinder outside, making a more true-to-life thread. Change the direction of the offset and make it 0.040" as indicated; this way the thread will be cut starting away from the face. Optionally we can also control the angle where the thread starts by changing the "Start Angle" value.


Offset option **OFF**   Offset option **ON**
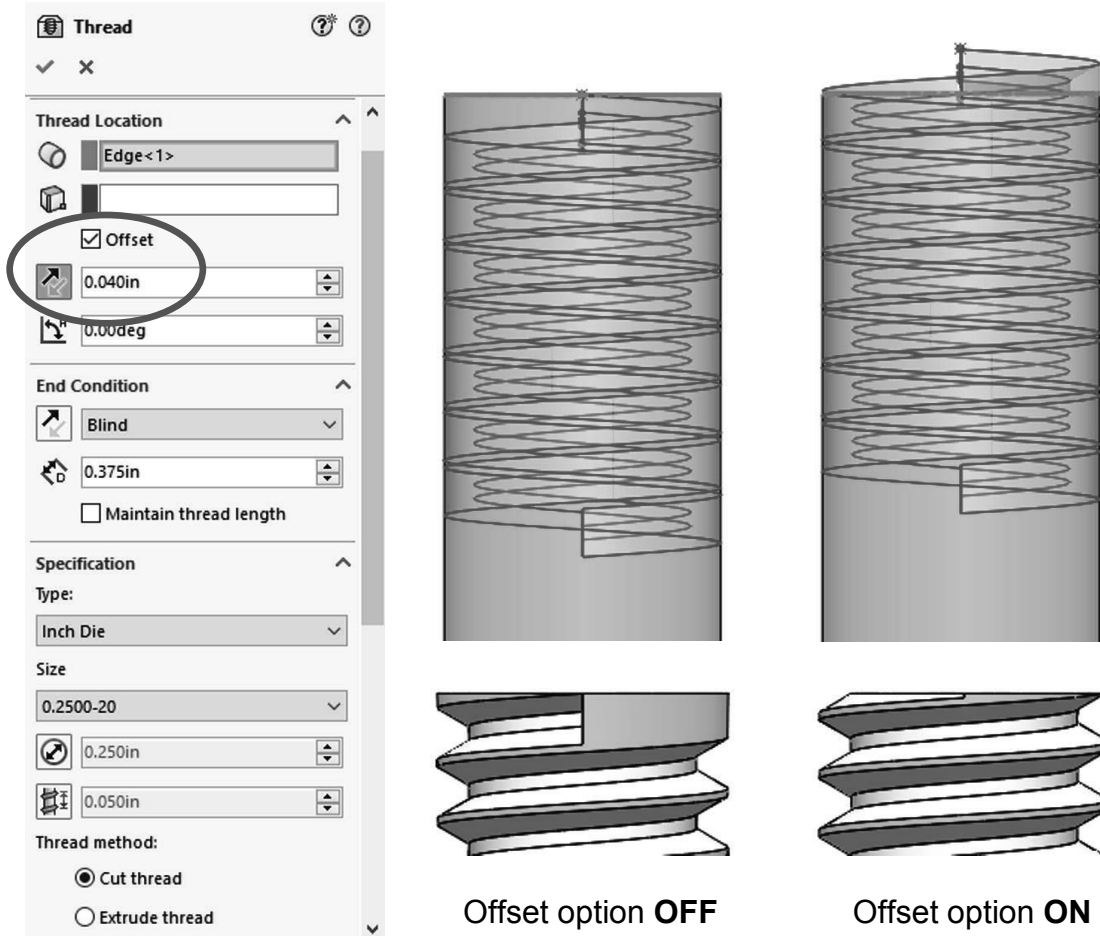
11.4. – The last thread will be the internal (tapped) thread. In this case the hole in the part has a 0.201" diameter (#7 drill bit). This is the drill size we'd use in real life to make a ¼-20 tapped hole. Select the "**Thread**" command, and this time select the pink hole's edge. These are the options we'll use:

- **Thread Location:** select the edge and turn on the "Offset" option to have the same result as in the previous step to start the thread outside the hole.
- In **Specification** select "Inch tap," size "0.2500-20."
- **Thread Method:** use the "Cut thread" option to cut into the part.
- For **End Condition** select "Up To Selection," and select the face at the other end of the block to make the thread go through the entire block, also using the "Offset" option to make the thread go past the end face.
- Under **Thread Method** select "Cut thread."
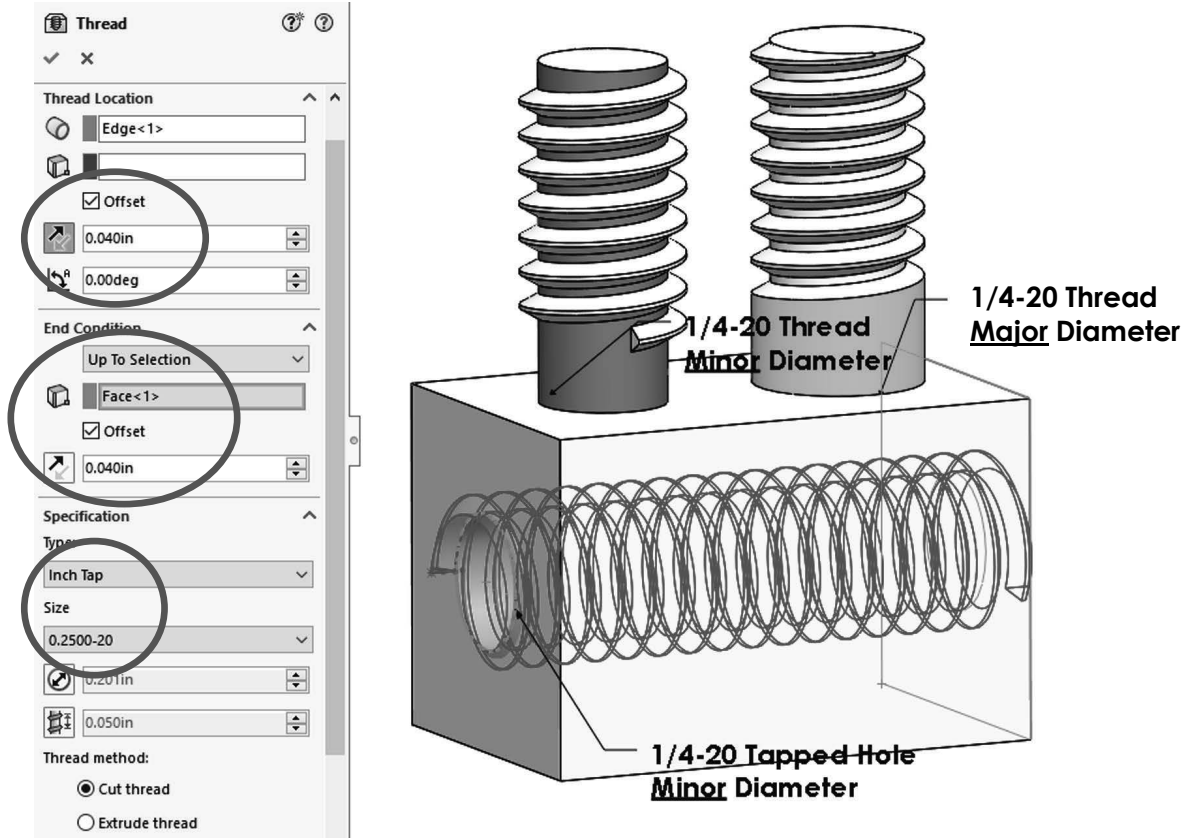
After finishing all three threads, orient the part to a Top view using the Hidden Lines Visible option. Here we can see the external threads made at the top fit perfectly inside the tapped hole.
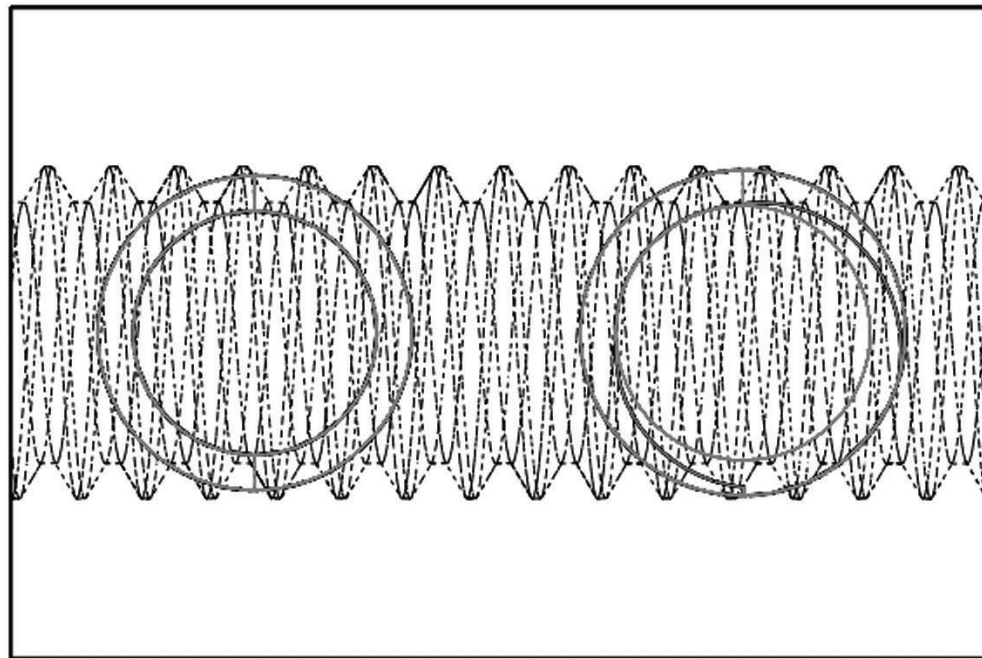
Notes:

Sweep: Bottle

For the next exercise we'll build the following bottle; notice the complete absence of straight linear edges and *almost* no flat faces in the model. This is a good example of a model that would be very difficult, if not impossible to complete using simple prismatic features.


Image made using RealView graphics

In this model we'll use two sweep features, one for the body and one for the thread. For the body we'll make a sweep with two guide curves, therefore we need to make four sketches for the first sweep feature: 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 for the sweep feature to work as expected.

12.1. – Make a new part in inches with three decimal places, set the material to a clear plastic, "ABS PC." Add the Path sketch as indicated in the "*Front Plane*" and Exit the sketch. Rename *'Path'*.

7.000
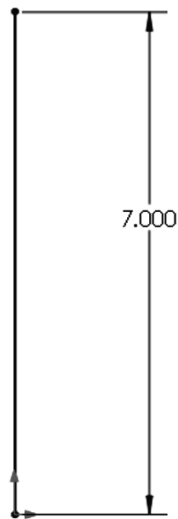
12.2. - Add a second sketch also in the "*Front Plane*." Make arcs and lines tangent to each other; use centerlines as reference for tangency. Add geometric relations to make all arcs equal. Exit the sketch and rename it *'Guide 1'*.

.500

R1.500

1.500

1.750
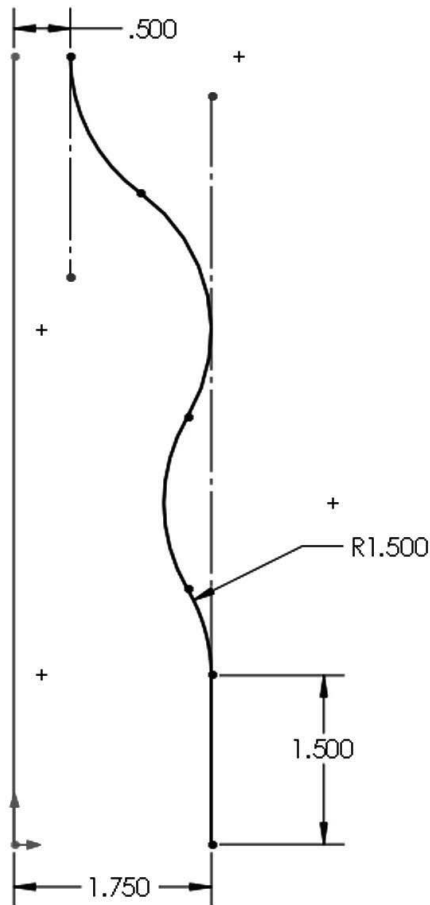
12.3. – Switch to a Right view and add a third sketch in the "*Right Plane*." Add geometric relations to previous sketches to maintain design intent. Make the topmost endpoint Horizontal to the topmost endpoint of the '*Path*' sketch. Arcs are equal size and tangent. Add a horizontal geometric relation to the endpoints indicated. Only geometric relations and two dimensions are needed to fully define the sketch. Exit the sketch and rename it '*Guide 2*.'

Right view   *View orientation between front and right view*
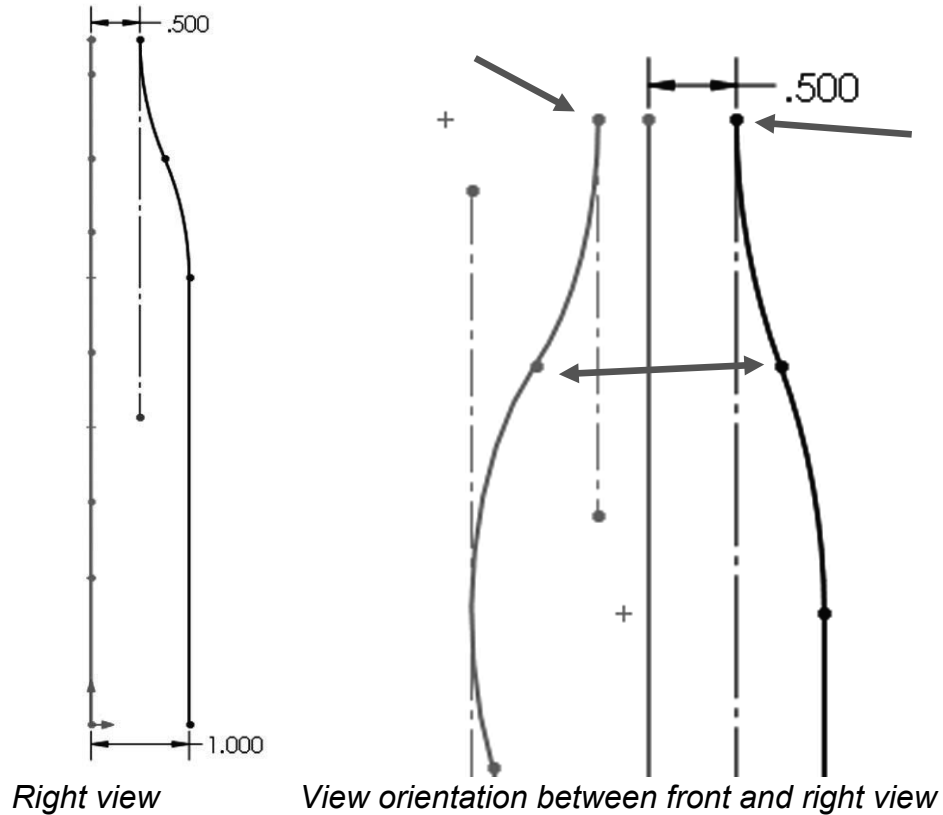
12.4. - Add a new sketch in the "*Top Plane*." Make an ellipse starting at the origin adding Pierce relations between the ellipse's major and minor axes to *'Guide 1'* and *'Guide 2'*. View is in Isometric for clarity. Exit the sketch and rename it *'Profile'*.

275
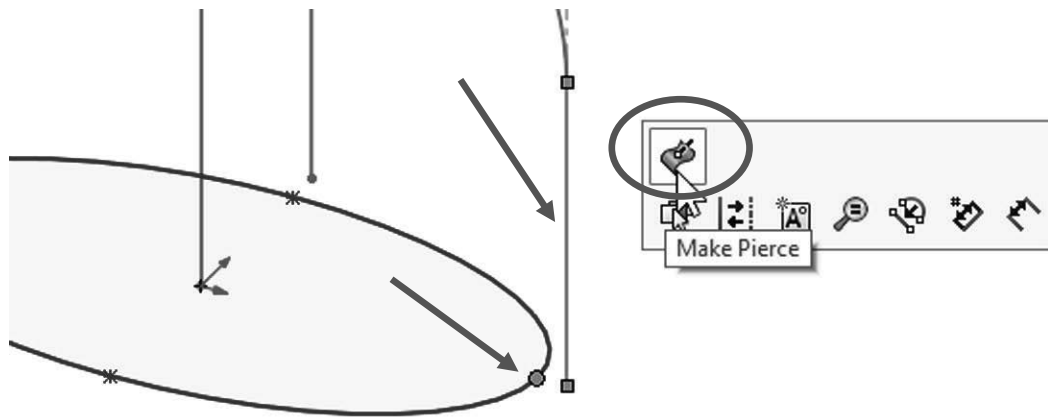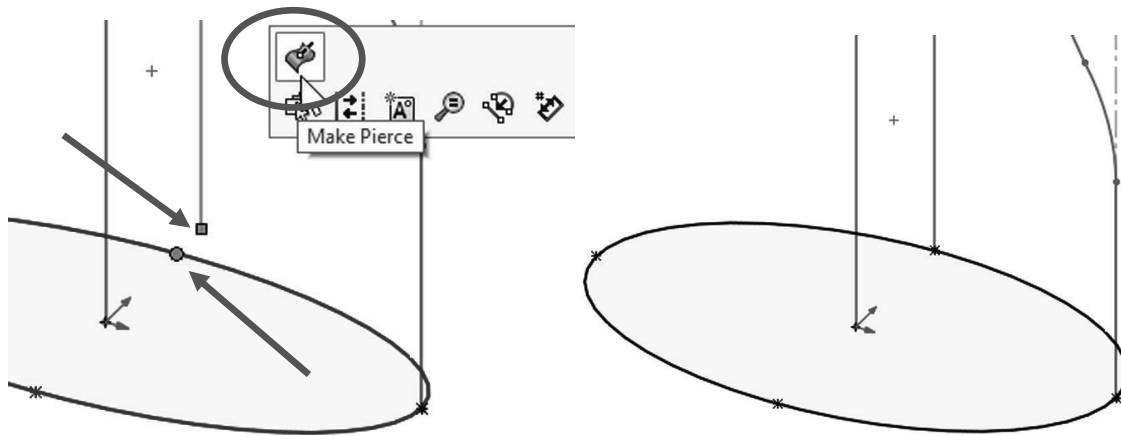
12.5. - Select the "**Sweep Boss/Base**" icon, add the *'Path'* sketch to the "Path" selection box and the *'Profile'* sketch to the "Profile" selection box. Expand the "Guide Curves" selection box and add both guides to it. Click OK to finish.
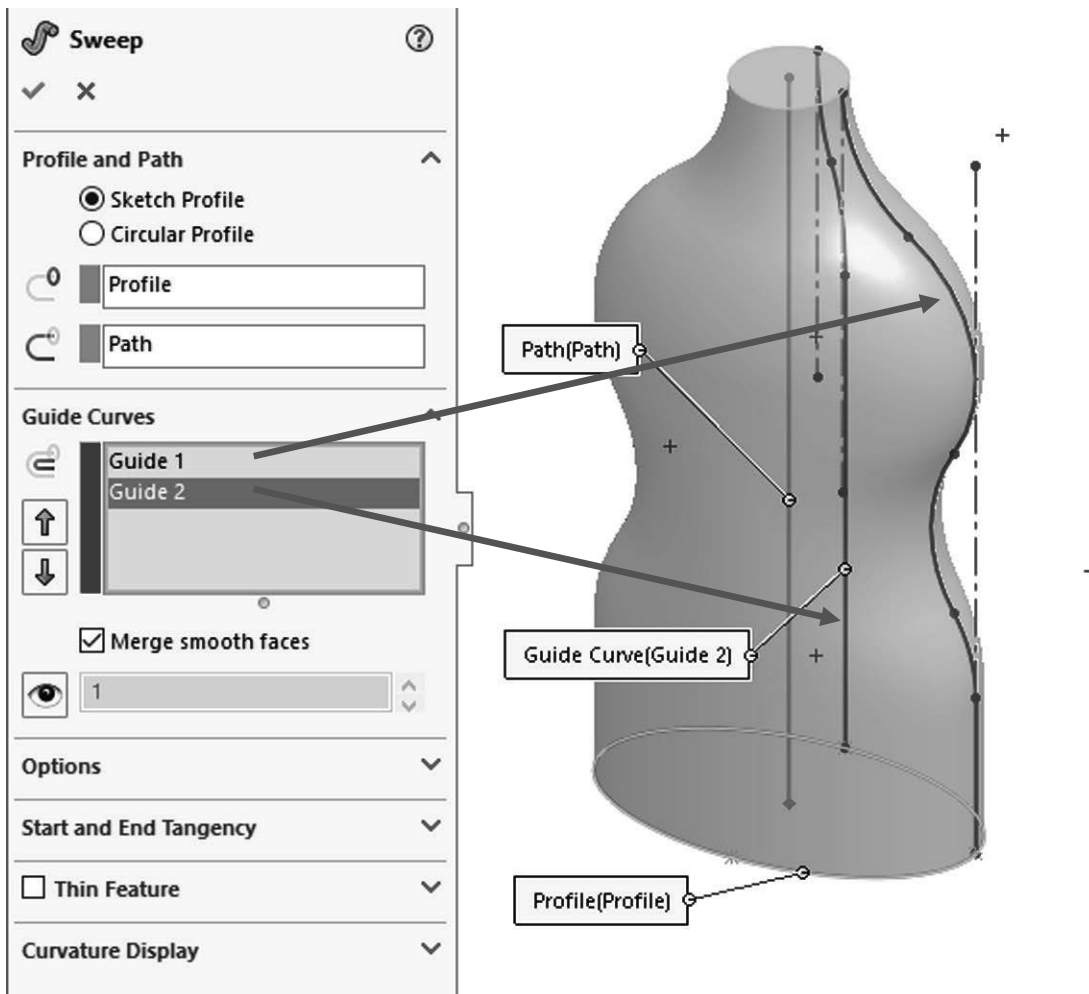
12.6. – To make the neck of the bottle, add a sketch on the top face, use "**Convert Entities**" to project the edge of the top circular face…
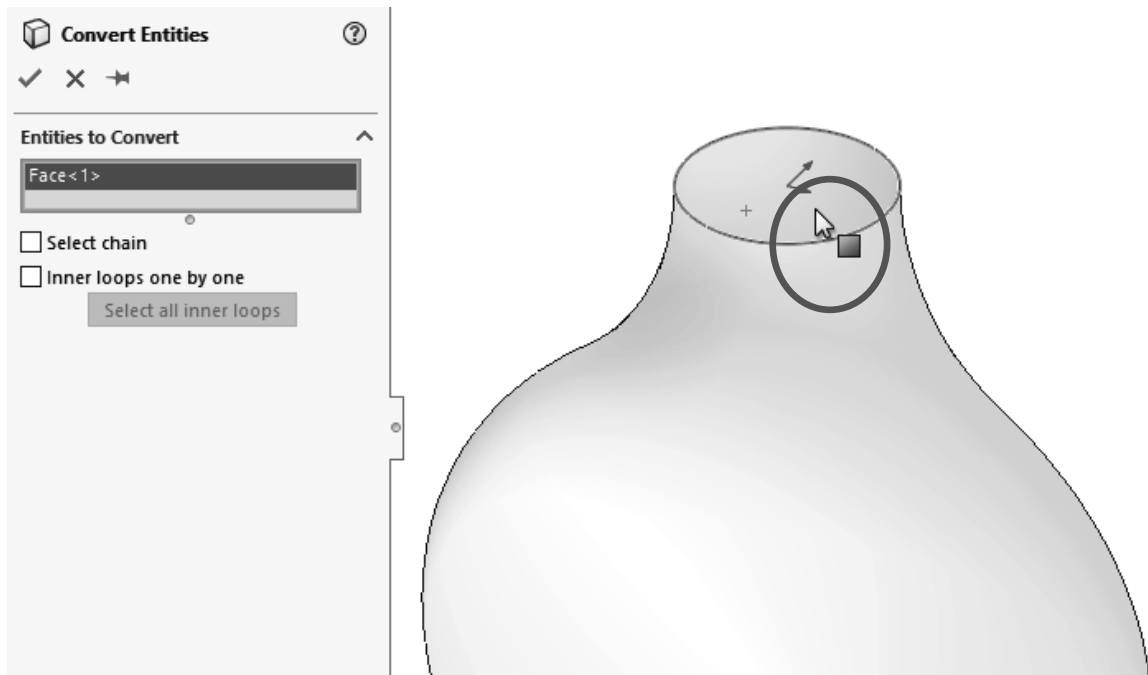
…and extrude it 1".
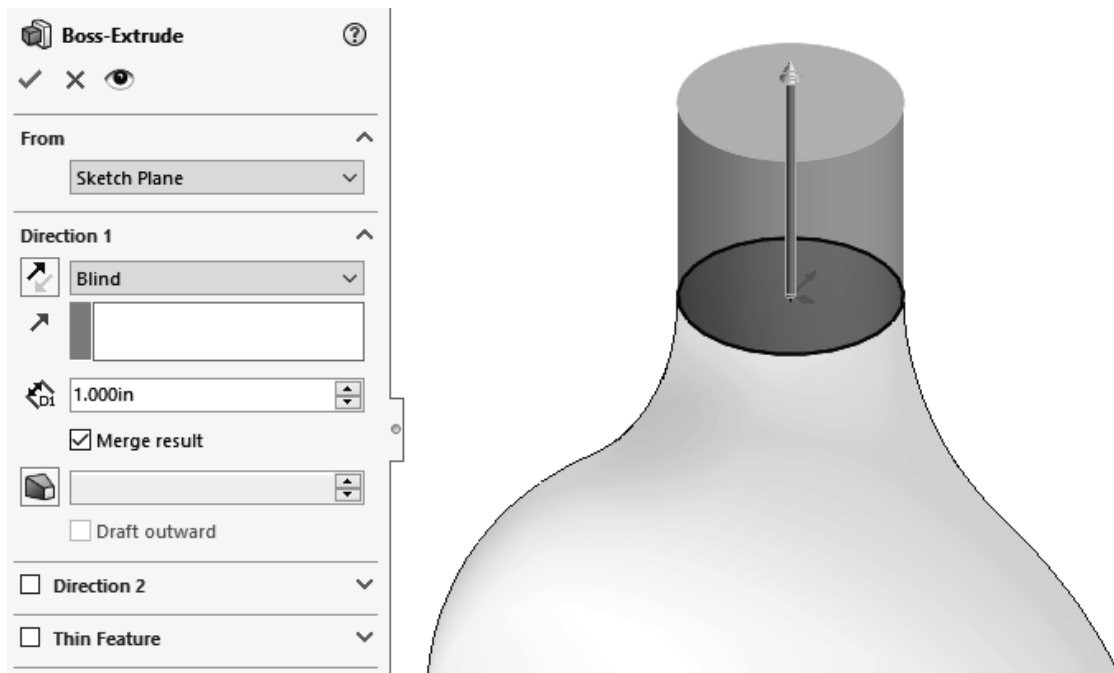
12.7. – Next, we need to add a fillet to the bottom of the bottle. Add a fillet to the bottom edge using the "Asymmetric" option. An asymmetric fillet allows us to define the distance the fillet goes into each of the faces. Activate the option and set the first distance to 0.375" going up, and 0.25" at the bottom of the bottle. Be aware that Distance 1 and 2 can be reversed.
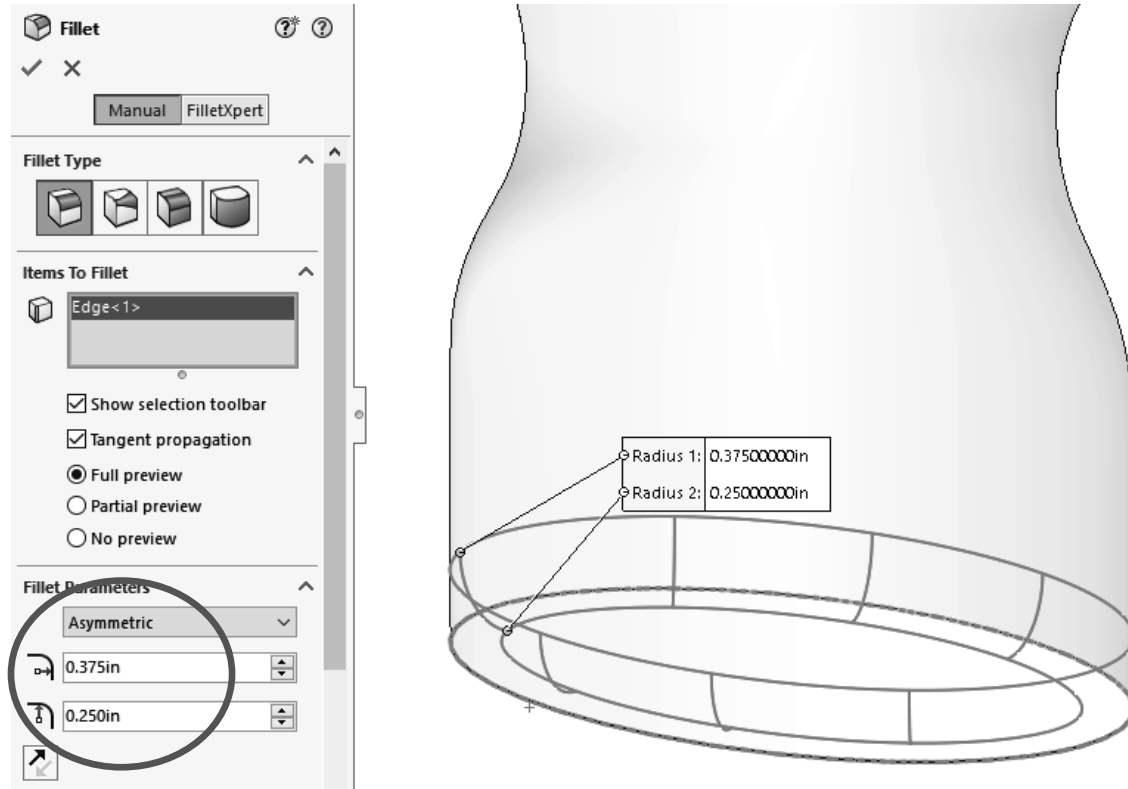

12.8. - Add a 0.050" shell to the part removing the top face.
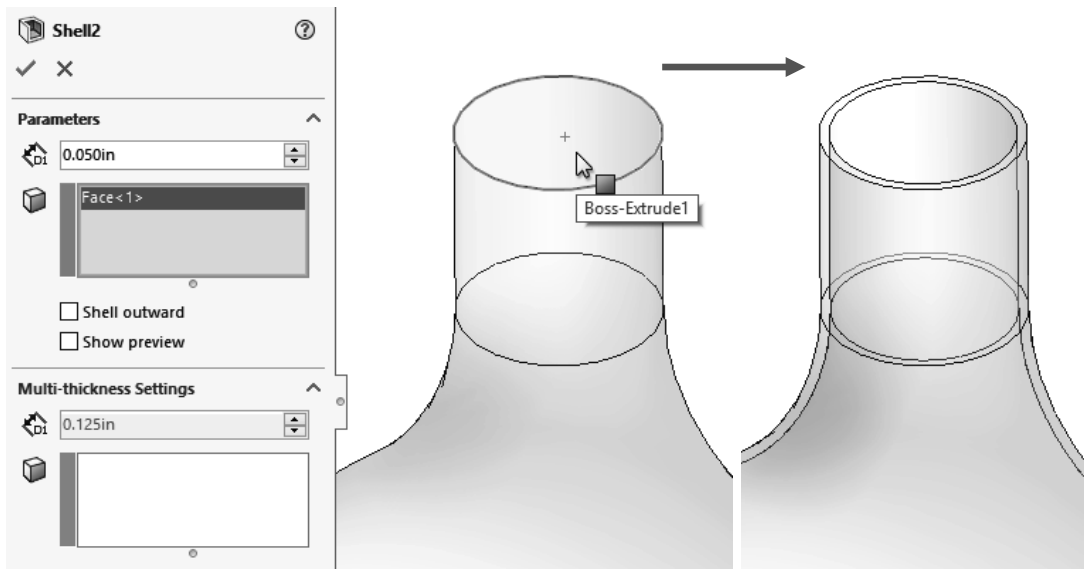
12.9. - Add a parallel auxiliary plane 0.125" below the topmost face.
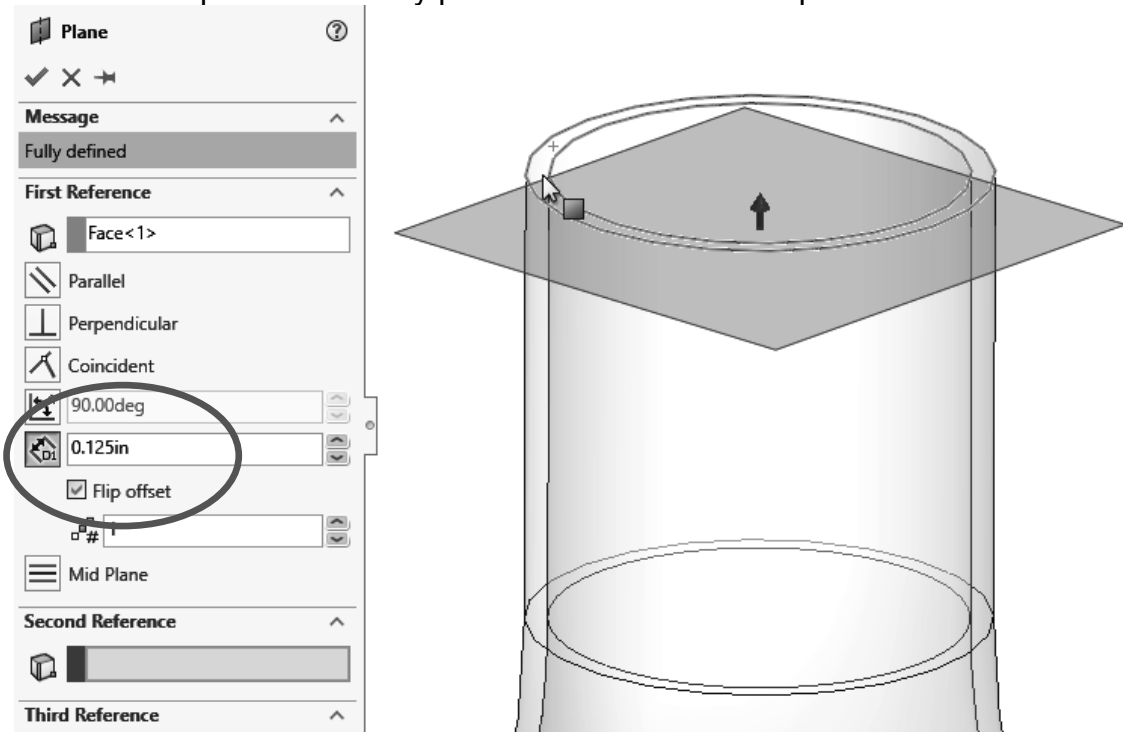
12.10. - Add a sketch in the new plane. Use "**Convert Entities**" to project the top outside edge and make a Helix defined by Pitch and Revolution using 0.2" for pitch and 2.5 revolutions. Make sure the Start Angle is set at 0 degrees.
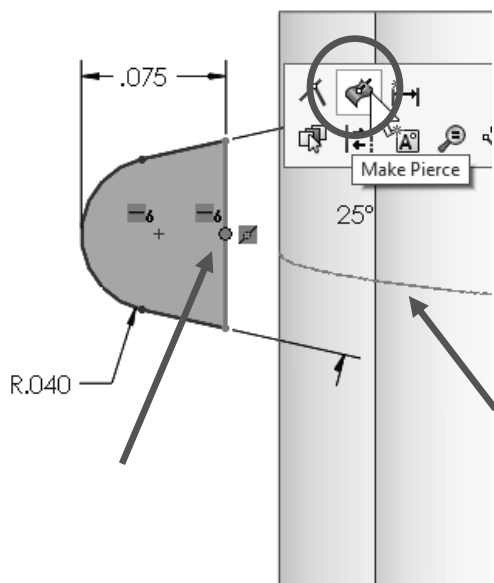

279
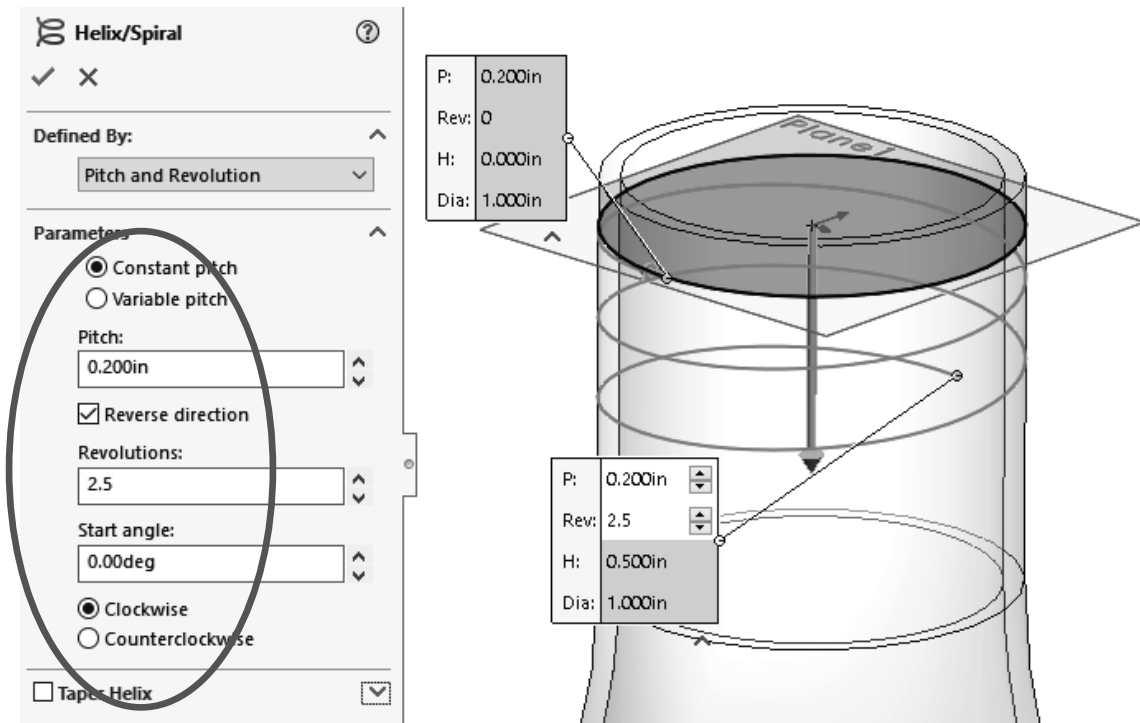
12.11. - Rotate to a Right View and add the following sketch in the "*Right Plane*" near the top of the bottle, close to the helix's start point; this will be the thread's Profile. For the vertical line we'll use a "**Midpoint Line"**; the point at the middle of the line will be used to add a horizontal relation with the arc's center point, and a Pierce relation to the helix. Draw, dimension and add the geometric relations indicated, then Exit the sketch and make a sweep feature using the helix curve and this thread profile. After creating the sweep, if needed, hide the helix curve.
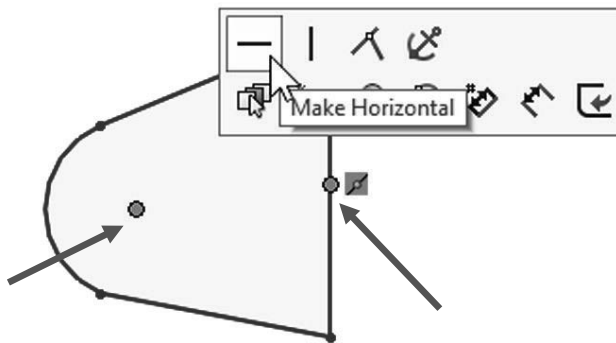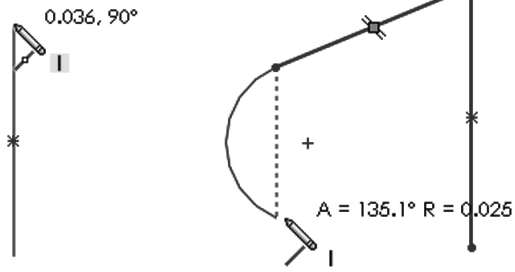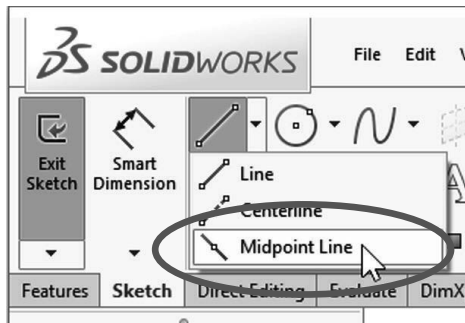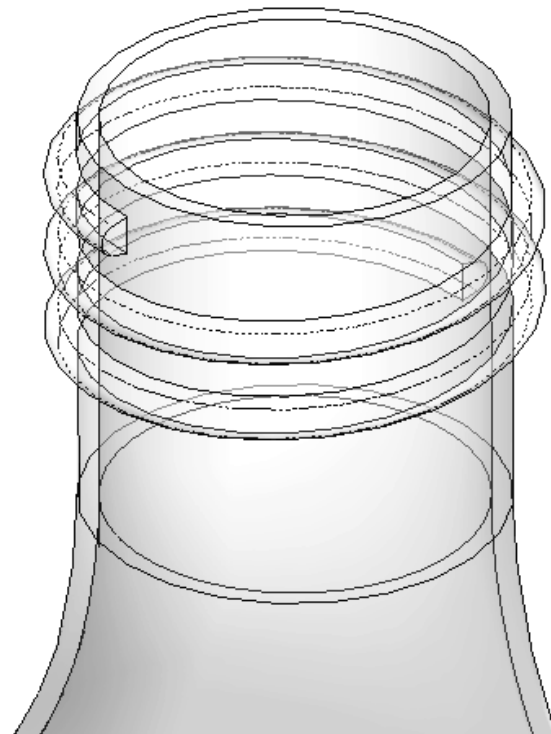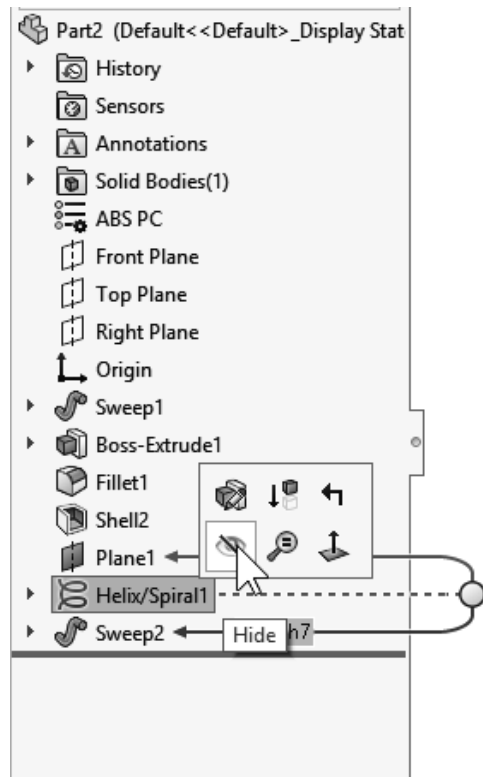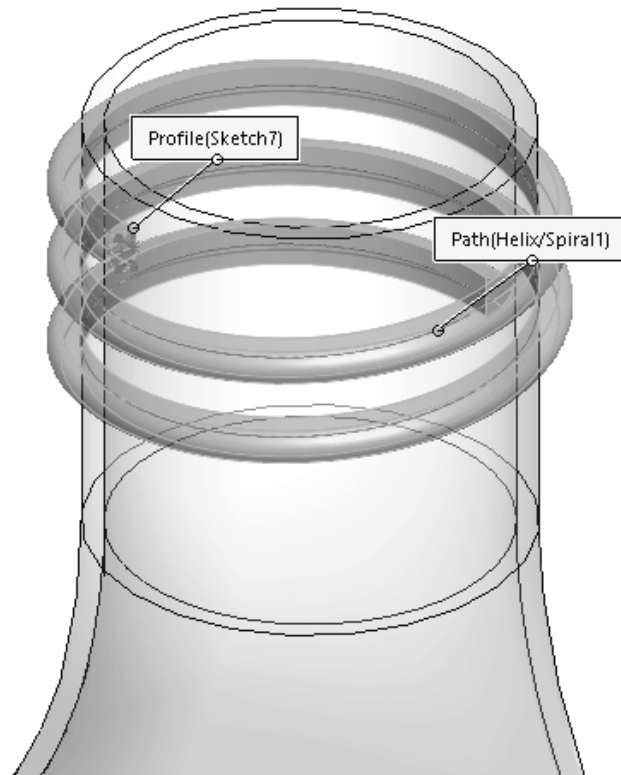
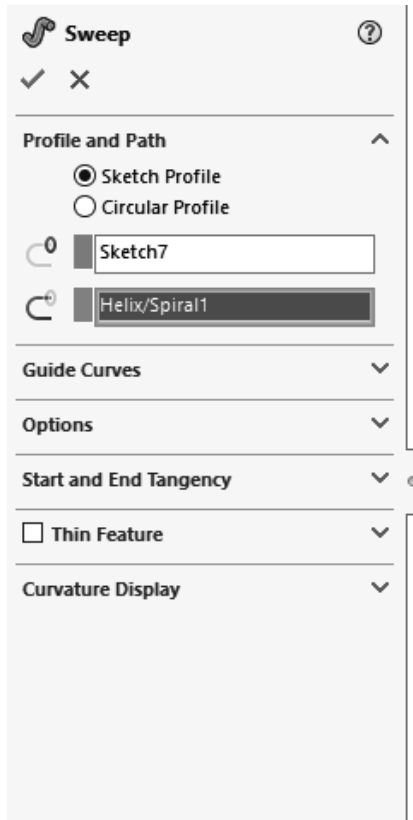Add Horizontal Relation   Add Dimensions (Use Shift for 0.075")
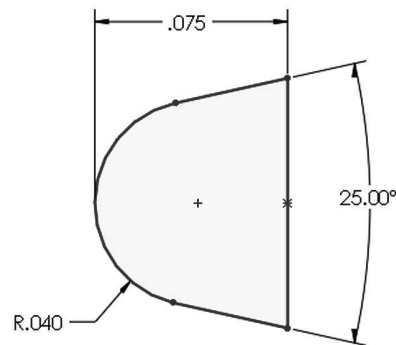
Add Pierce Relation   Finished Sketch
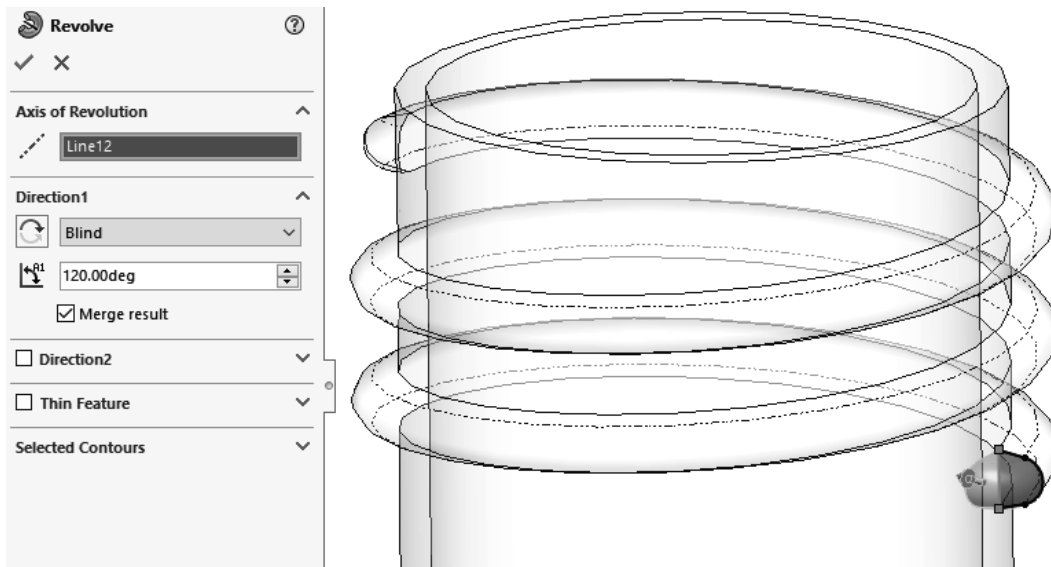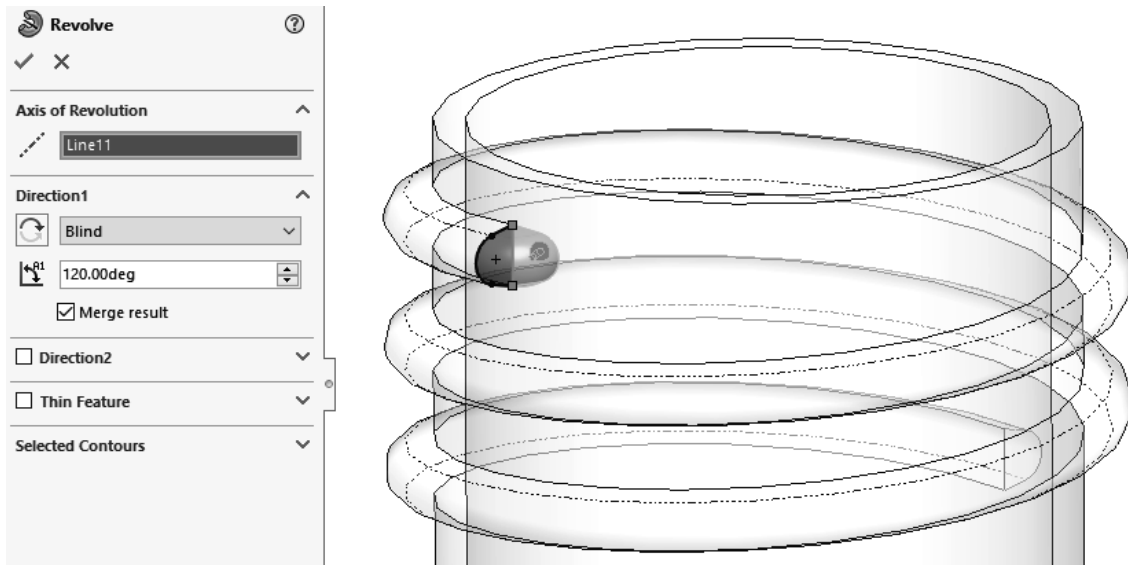
Sweep

✓ ✗

Profile and Path ⌃

◉ Sketch Profile
○ Circular Profile

Sketch7

Helix/Spiral1

Guide Curves ⌄

Options ⌄

Start and End Tangency ⌄

☐ **Thin Feature** ⌄

Curvature Display ⌄

Profile(Sketch7)

Path(Helix/Spiral1)

Part2 (Default<<Default>_Display Stat
▸ 🔁 History
   📷 Sensors
▸ 🅰 Annotations
▸ 📦 Solid Bodies(1)
   ⚙ ABS PC
   📄 Front Plane
   📄 Top Plane
   📄 Right Plane
   ⌞ Origin
▸ 🎣 Sweep1
▸ 📦 Boss-Extrude1
   📦 Fillet1
   📦 Shell2
   📦 Plane1 ◄
▸ 📧 Helix/Spiral1
▸ 🎣 Sweep2 ◄   Hide h7

Hide the Helix curve   Finished Sweep

281

12.12. - Add a new sketch in the flat face at the beginning of the thread and use the "**Convert Entities**" command to project the face's edges into the sketch.
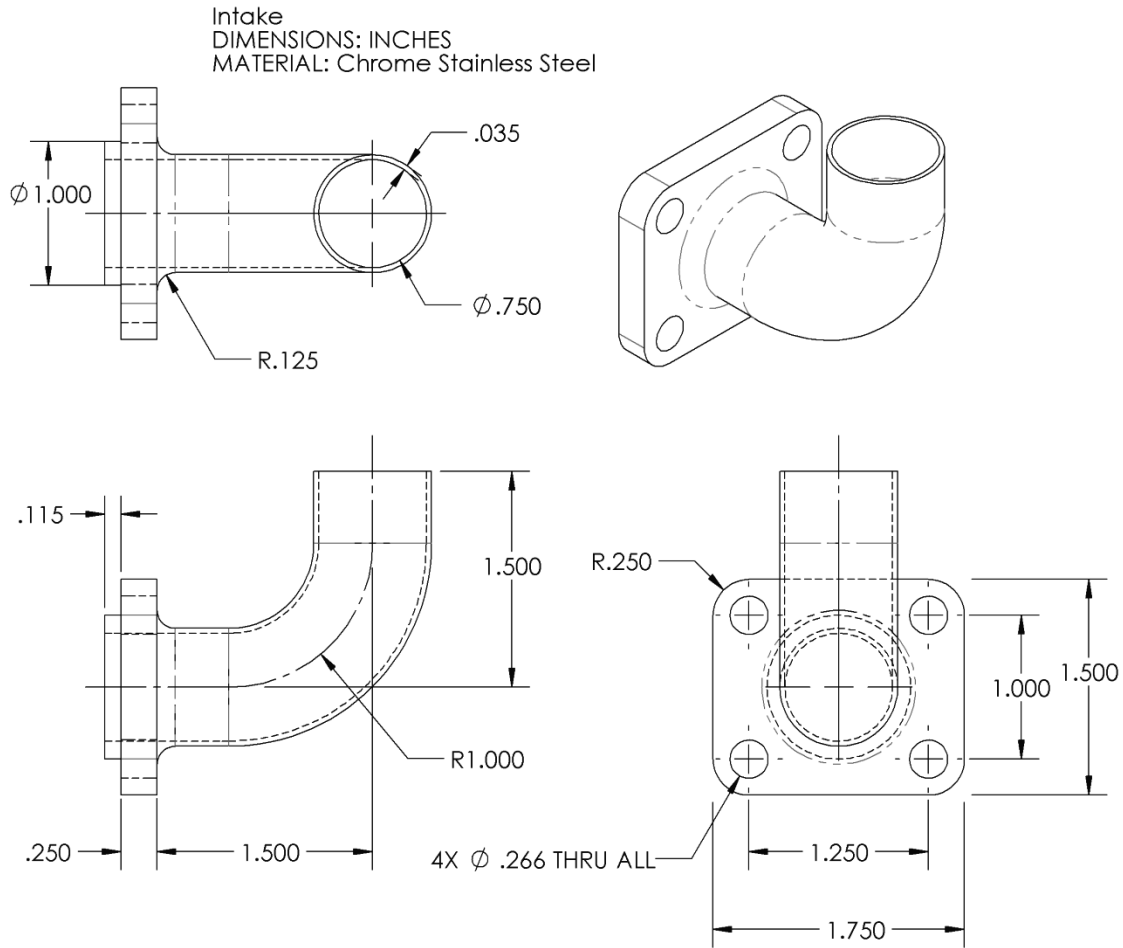
Make a "Revolved Boss" 120 degrees using the vertical sketch line as axis of revolution. The reason to make it 120 degrees is to completely merge it with the body of the bottle. Add another sketch at the end of the thread and repeat the same process to finish the thread.
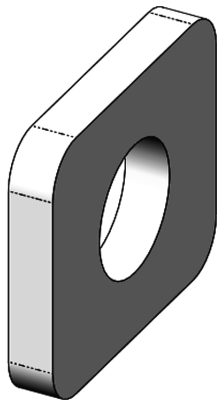
12.13. - Add a 0.02" fillet to the thread as a finishing touch. You may need to select multiple edges. Save the part as '*Bottle Sweep*' and close it.
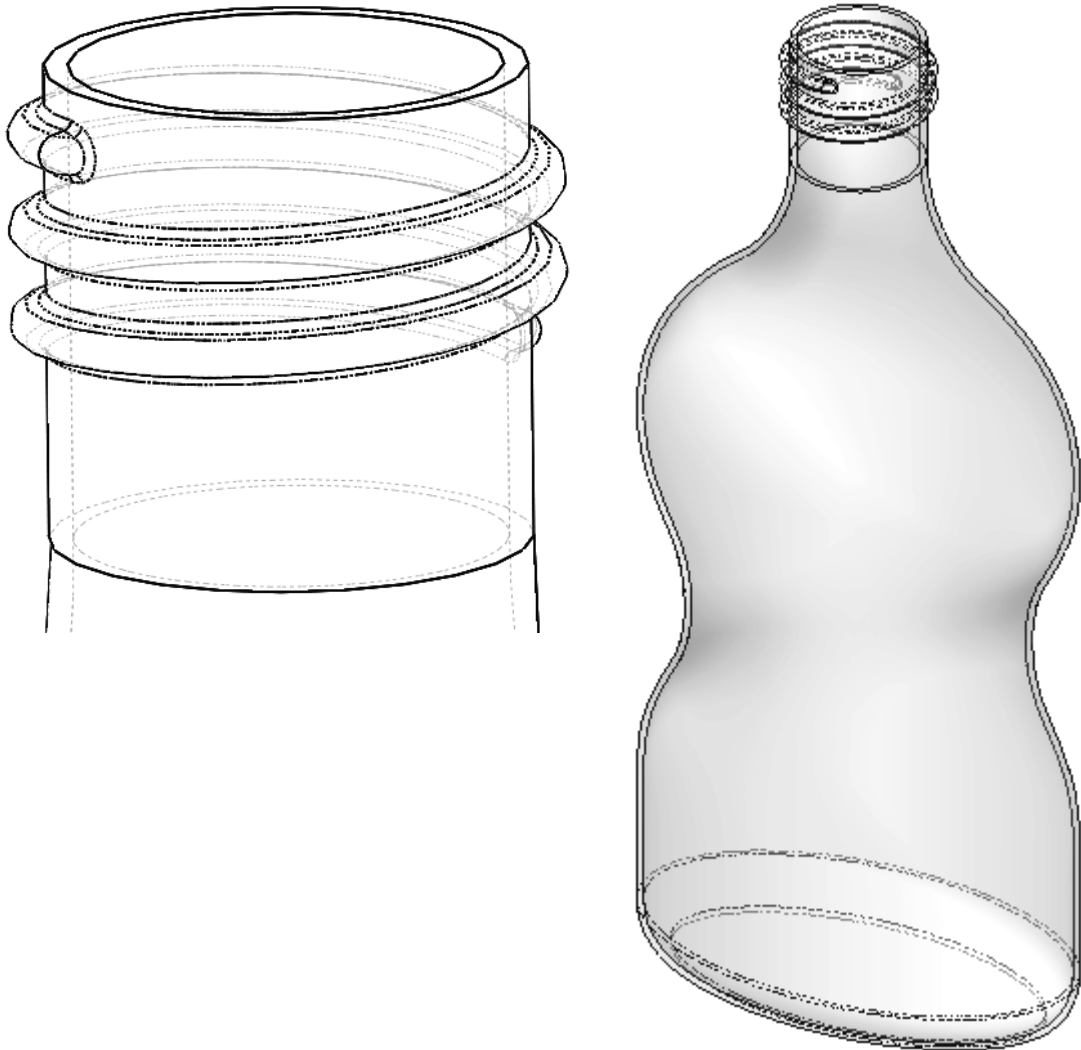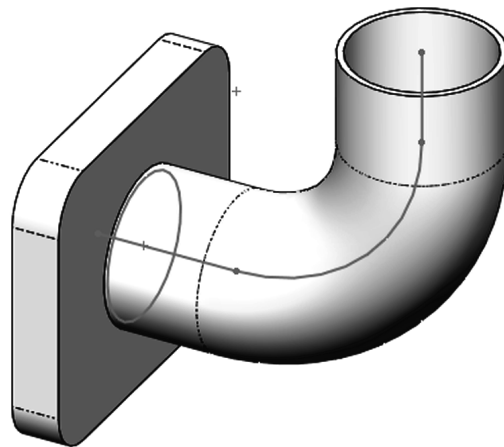
Engine Project Parts: Make the following components to build the engine. Save the parts using the name provided. High resolution images are included on the exercise files.
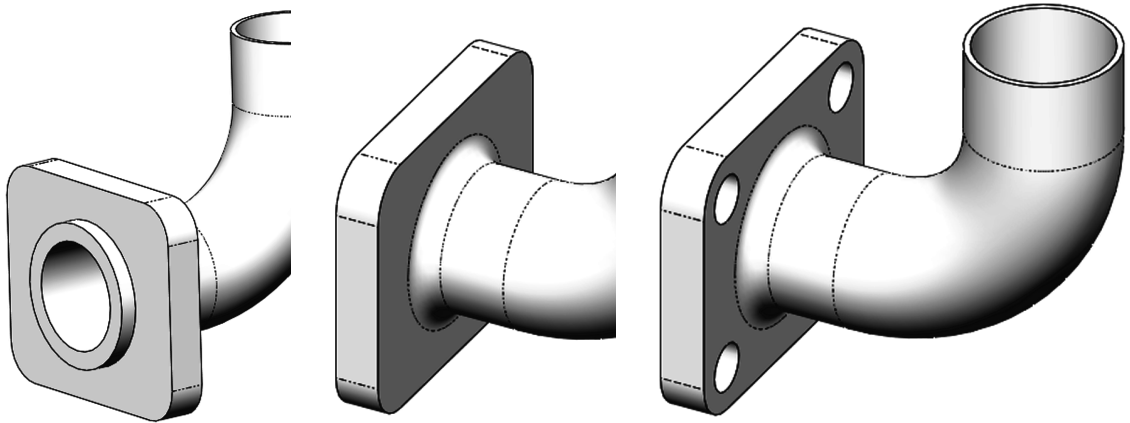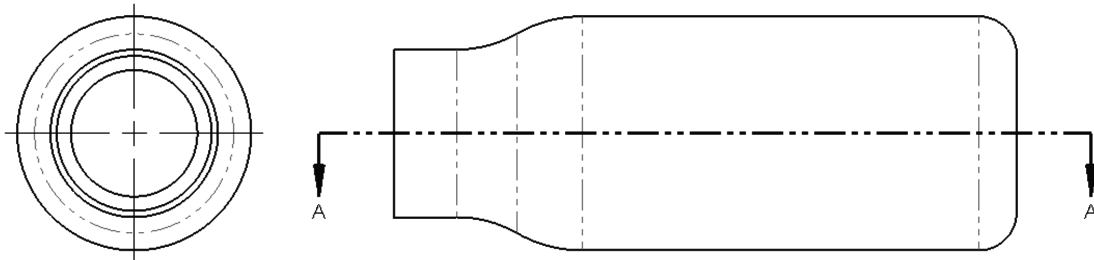
Intake
DIMENSIONS: INCHES
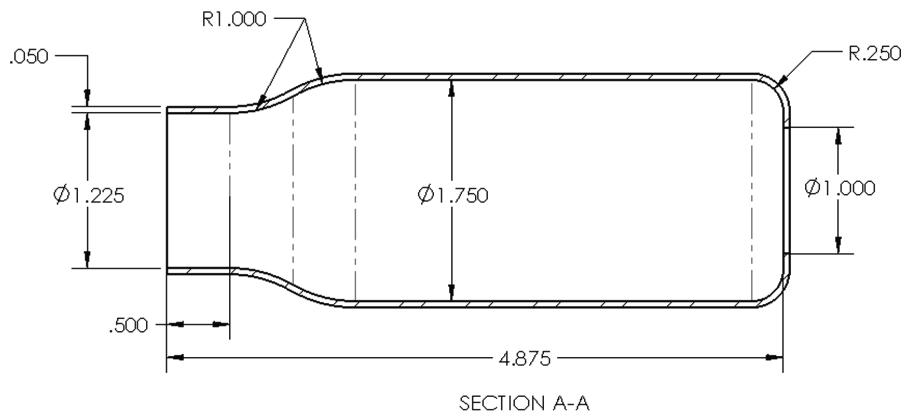MATERIAL: Chrome Stainless Steel

.035

$\emptyset$ 1.000

$\emptyset$ .750

R.125

.115

1.500

R.250

R1.000

.250

1.500

4X $\emptyset$ .266 THRU ALL

1.000

1.500

1.250

1.750

TIPS:

Extrude base

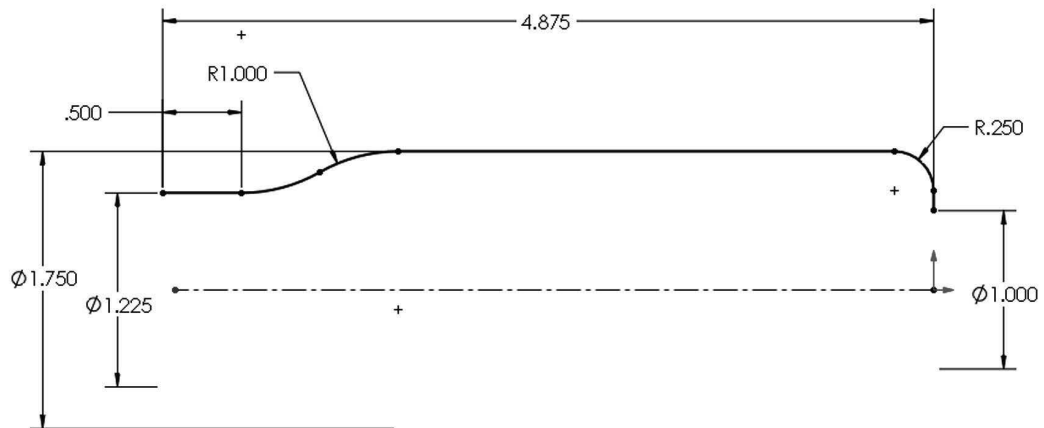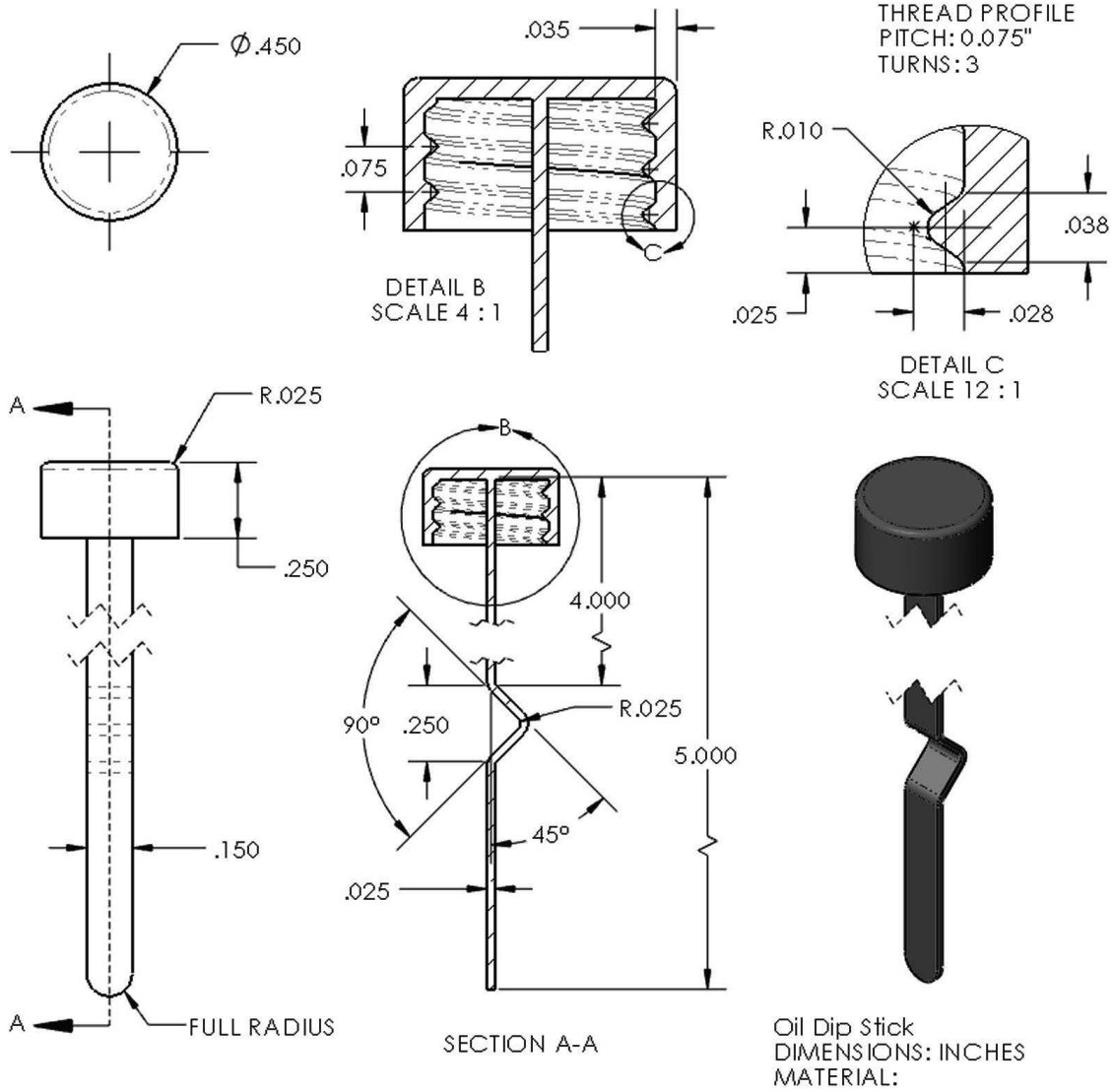Add Path & Profile sketches. Create a sweep using the "Thin Feature" option

Exhaust Cover_
DIMENSIONS: INCHES
MATERIAL: AISI 1020

R1.000
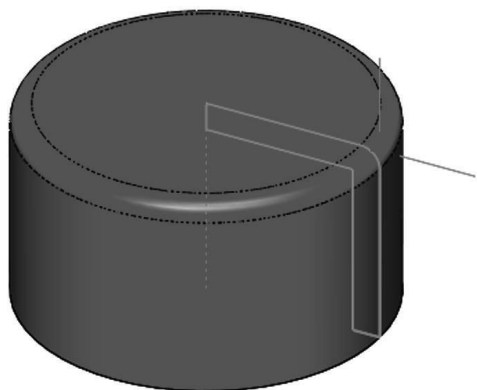
.050

Ø1.225

.500

Ø1.750

R.250

Ø1.000

4.875

SECTION A-A

A — A

TIPS:

4.875

R1.000

.500

R.250

Ø1.750

Ø1.225

Ø1.000

Make a "Revolved Boss" using the "Thin Feature" option, adding material outside

Ø.450

.035

THREAD PROFILE
PITCH: 0.075"
TURNS: 3

.075

DETAIL B
SCALE 4 : 1

R.010

.025

.038

.028

DETAIL C
SCALE 12 : 1

A

R.025

.250

.150

FULL RADIUS

B

4.000

90° .250

R.025

45°

.025

5.000

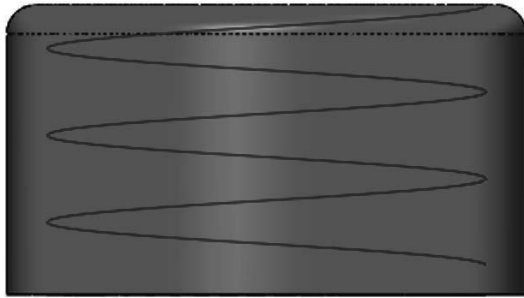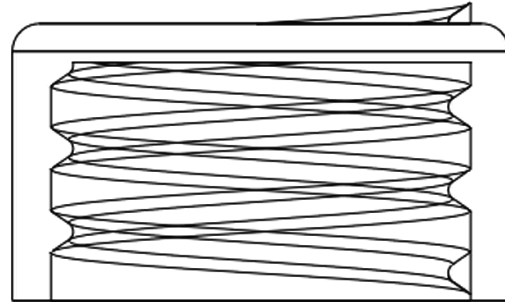SECTION A-A

A

Oil Dip Stick
DIMENSIONS: INCHES
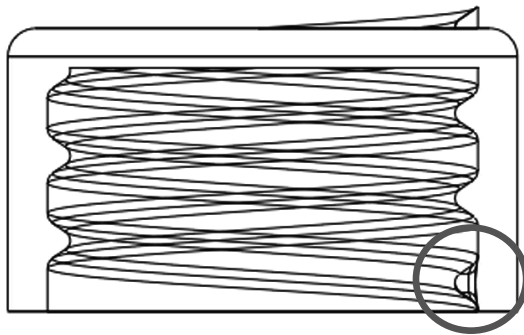MATERIAL:

TIPS:

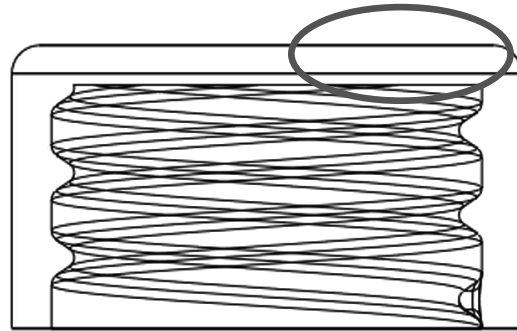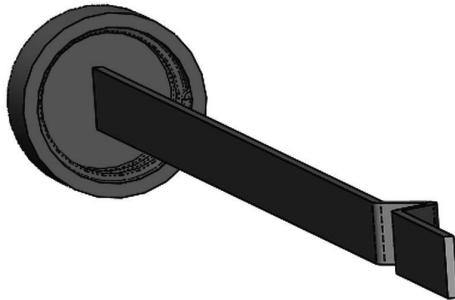Add Revolved feature
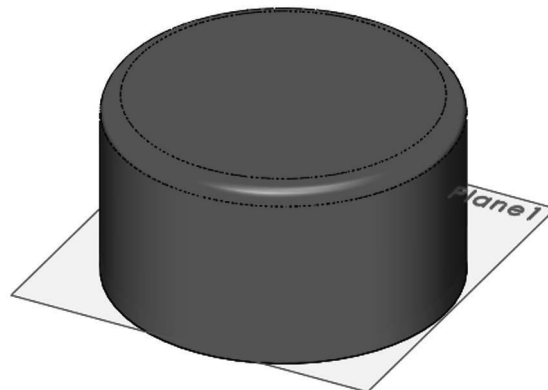
Add offset plane

Add Helix

Add thread profile and create sweep.
Thread will go through the top; it will be
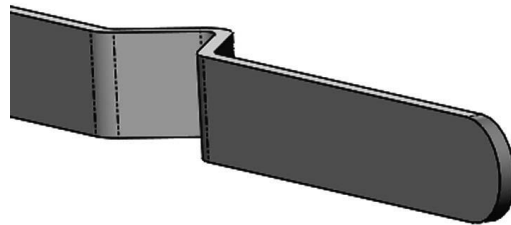trimmed later

Add Revolved boss and fillet to finish
thread

Trim the excess thread from the top

Add extrusion for dip stick using the
"Thin Feature" option, adding material
using the Mid-Plane option and the
Auto-Fillet corners option.

Add full round fillet at end of dip stick