

Includes  
CSWP Preparation Material

# SolidWorks® 2015

## Part II - Advanced Techniques

Parts, Surfaces, Sheet Metal, SimulationXpress,  
Top-Down Assemblies, Core and Cavity Molds



Paul Tran CSWE, CSWI

Visit the following websites to learn more about this book:

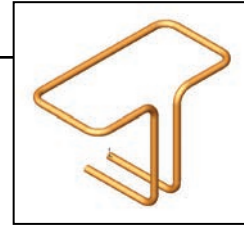


Google books

**BARNES & NOBLE**

# CHAPTER 1








## Introduction To 3D Sketch

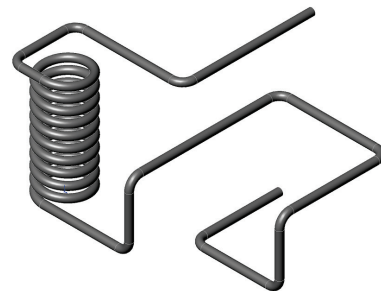
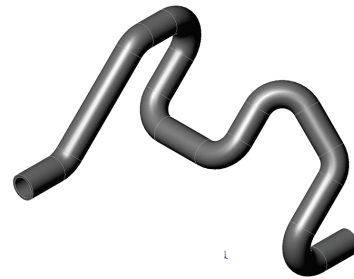


### Introduction to 3D Sketch

SolidWorks enables you to create 3D sketches. A 3D sketch consists of lines and arcs in series and splines. You can use a 3D sketch as a sweep path, as a guide curve for a loft or sweep, a centerline for a loft, or as one of the key entities in a piping system. Geometric relations can also be added to 3D Sketches.

#### Parameters

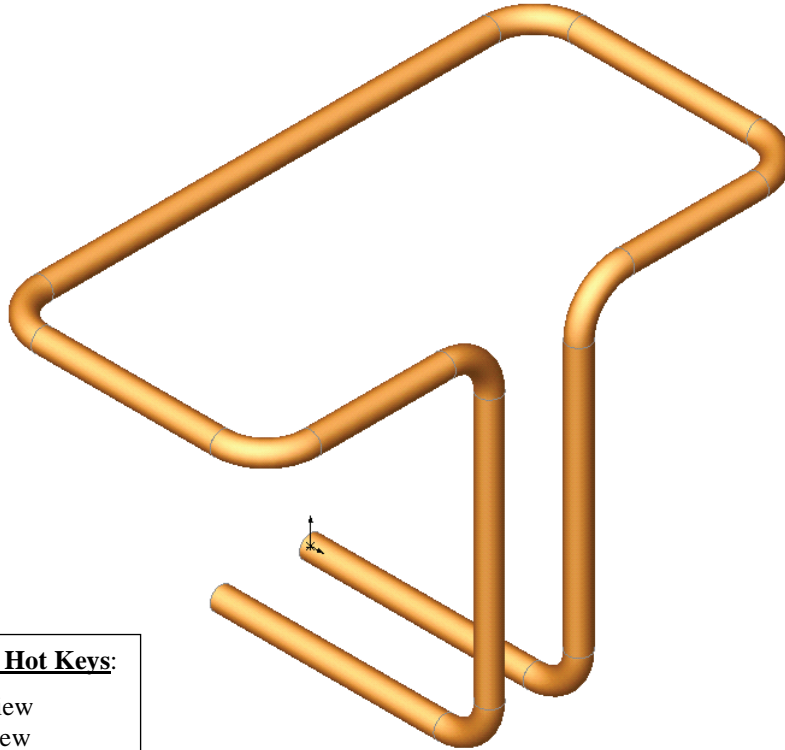
-  **X Coordinate**
-  **Y Coordinate**
-  **Z Coordinate**
-  **Curvature** (Spline curvature at the frame point)
-  **Tangency** (In the **XY** plane)
-  **Tangency** (In the **XZ** plane)
-  **Tangency** (In the **YZ** plane)



#### Space Handle

When working in a 3D sketch, a graphical assistant is provided to help you maintain your orientation while you sketch on several planes. This assistant is called a *space handle*. The space handle appears when the first point of a line or spline is defined on a selected plane. Using the space handle you can select the axis along which you want to sketch.

# Introduction to 3D Sketch



## View Orientation Hot Keys:

Ctrl + 1 = Front View  
Ctrl + 2 = Back View  
Ctrl + 3 = Left View  
Ctrl + 4 = Right View  
Ctrl + 5 = Top View  
Ctrl + 6 = Bottom View  
Ctrl + 7 = Isometric View  
Ctrl + 8 = Normal To Selection

Dimensioning Standards: **ANSI**  
Units: **INCHES** – 3 Decimals

## Tools Needed:



3D Sketch



2D Sketch



Sketch Line



Circle



Dimension



Add Geometric Relations



Sketch Fillet

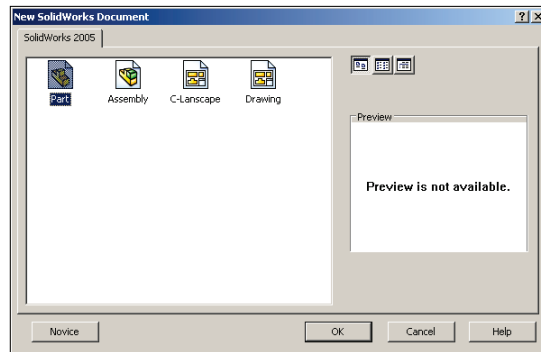


Tab Key






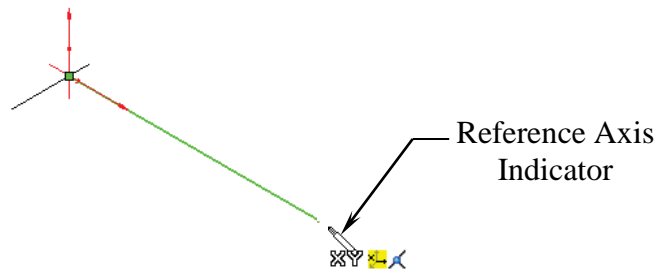
Base/ Boss Sweep

1. Starting a new part file: Select **File / New / Part / OK**.

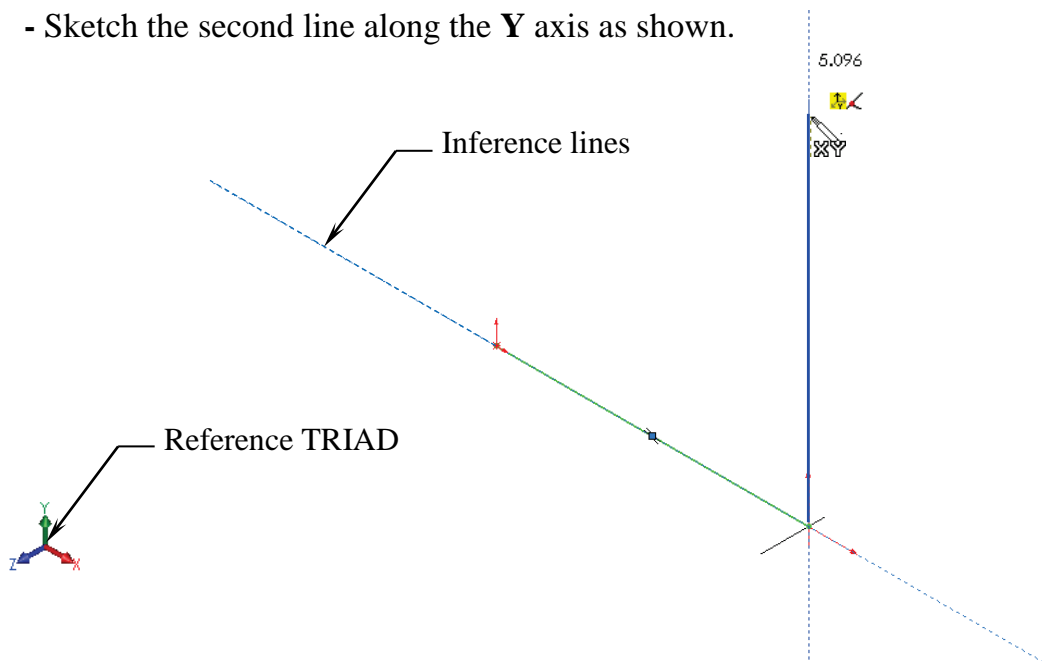


2. Using 3D Sketch:

- Click  or select **Insert / 3D Sketch**, and change to **Isometric view** .
- Select the Line tool  and sketch the first line along the **X** axis.

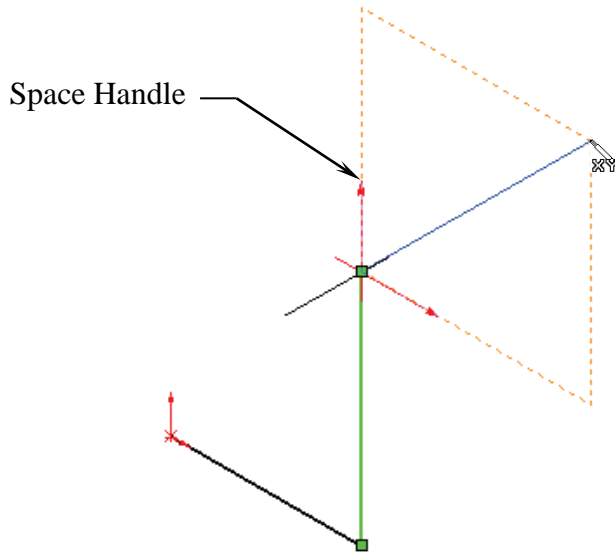


- Sketch the second line along the **Y** axis as shown.



### 3. Changing direction:

- By default your sketch is relative to the default coordinate system in the model.
- To switch to one of the other two default planes, press the **TAB** key and the reference origin of the current sketch plane is displayed on that plane.

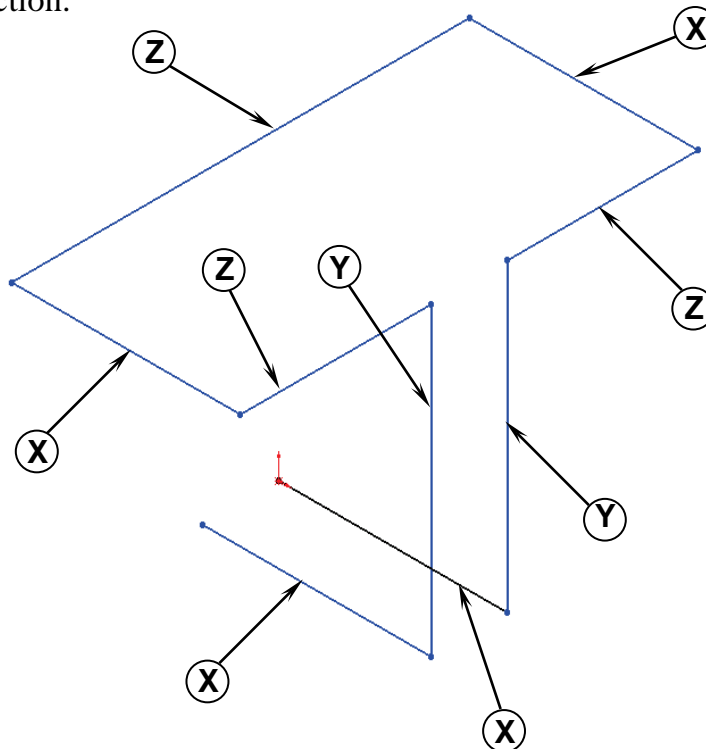


#### The TAB key


While sketching the lines, press the **TAB** key to switch to other planes/directions.

### 4. Completing the profile:

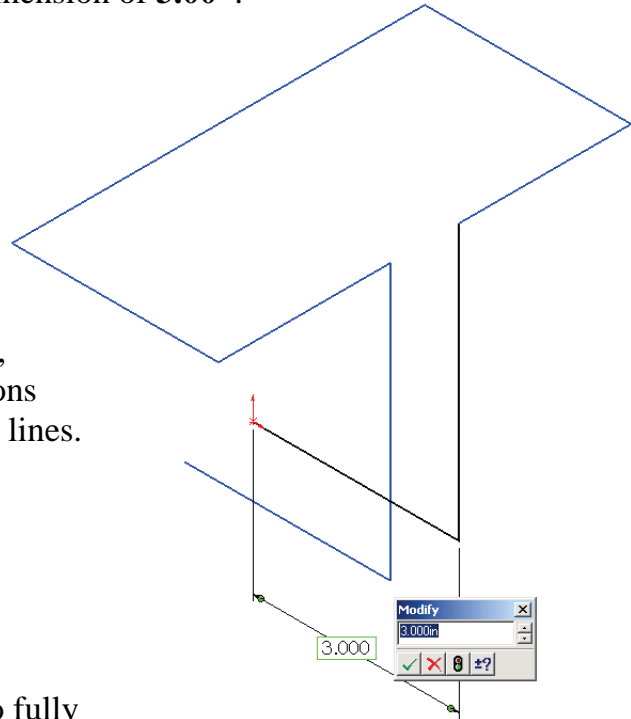
- Follow the axis as labeled; press **TAB** if necessary to change the direction.



## 5. Adding dimensions:

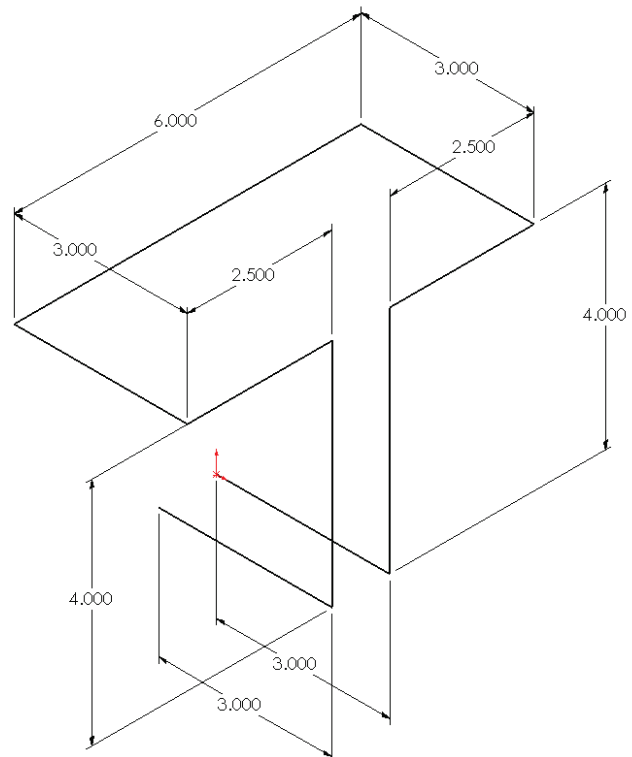
- Click  or select **Tools / Dimensions / Smart Dimension**.
- Click on the first line and add a dimension of **3.00"**.

- There is not a general sequence to follow when adding dimensions, so for this lesson, add the dimensions in the same order you sketched the lines.




- Continue adding the dimensions to fully define the 3D sketch as shown.

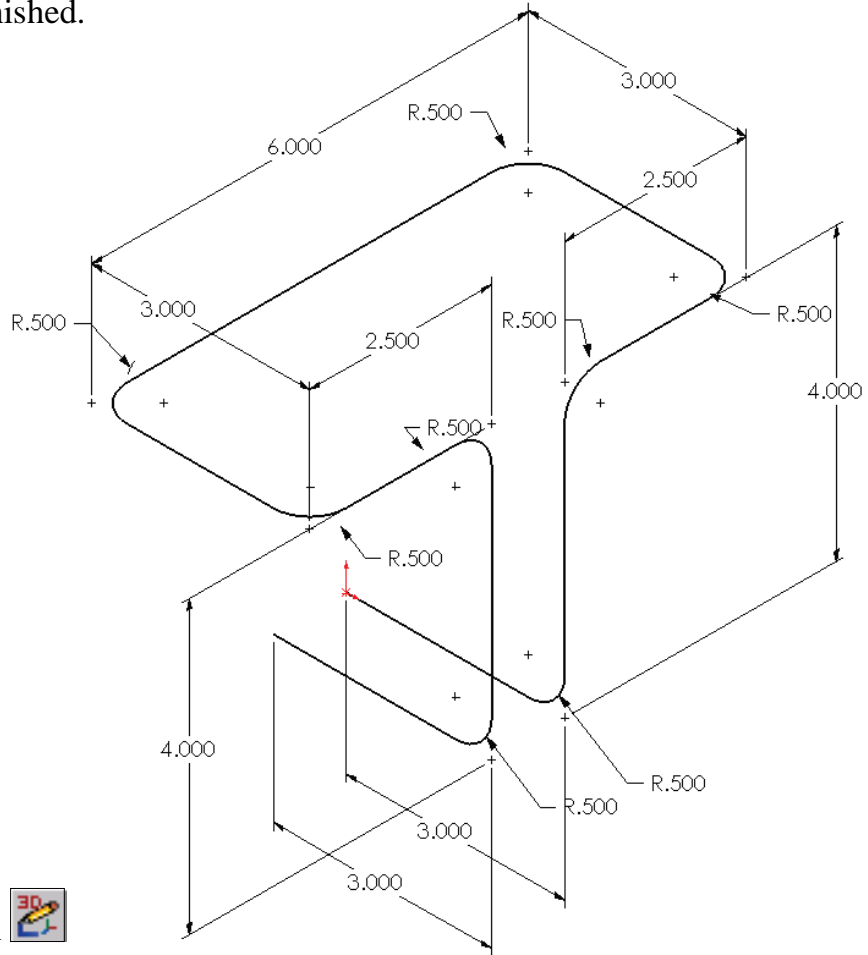
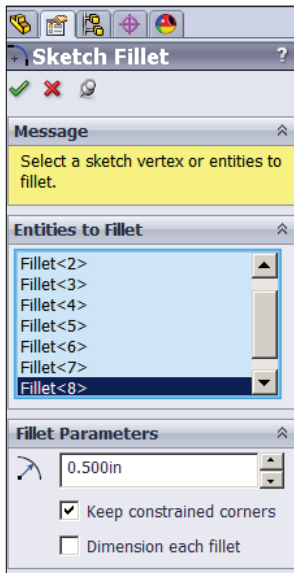
*(Note: To make the dimensions parallel to the lines as shown, select the lines, not the end points).*




- Re-arrange the dimensions so they are easy to read, which will make editing a little easier later on.

## 6. Adding the Sketch Fillets:

- Click  or select **Tools / Sketch Tools / Fillet**.
- Add **.500"** fillets to all the intersections as indicated.
- Enable the **Keep Constrained Corner** check box (Maintains the virtual intersection point if the vertex has dimensions or relations).
- Click **OK** when finished.

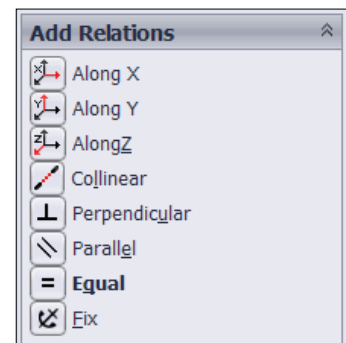


- **Exit** the 3D Sketch  or press **Control + Q**.

 **Geometric Relations**



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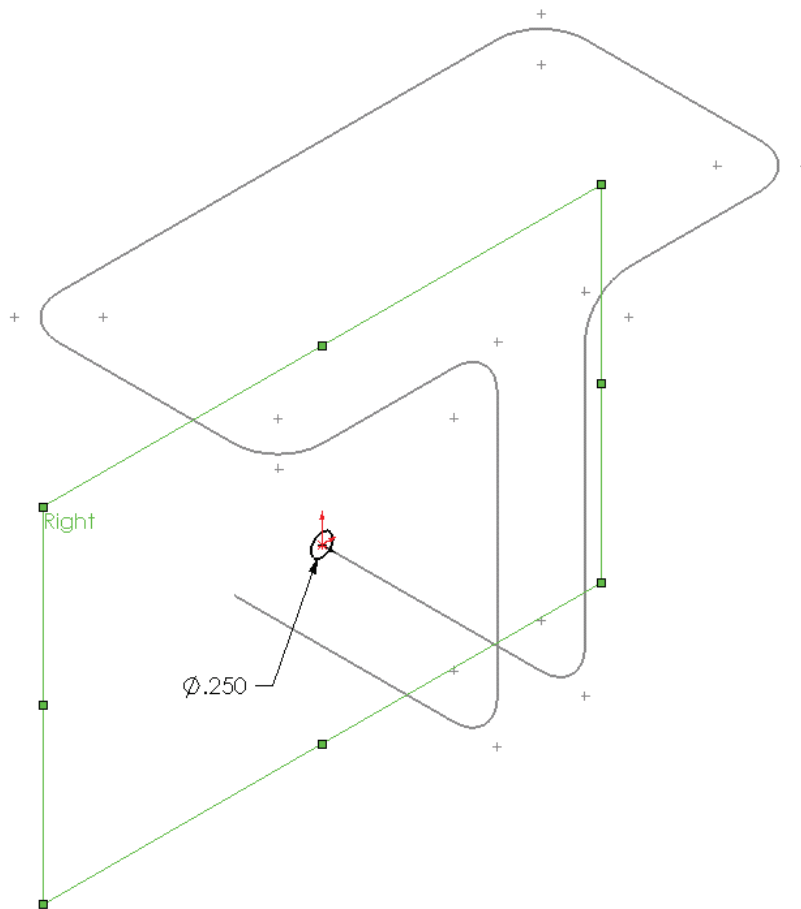
Geometric Relations such as Along Z and Equal can also be use to replace some of the duplicate dimensions.







## 7. Sketching the Sweep Profile:

- Select the Right plane from the FeatureManager tree.
- Click  to open a new 2D sketch or select **Insert / Sketch**.
- Sketch a Circle  using the Origin as the center. (The system automatically creates a Coincident relation between the Center of the circle and the Origin.)







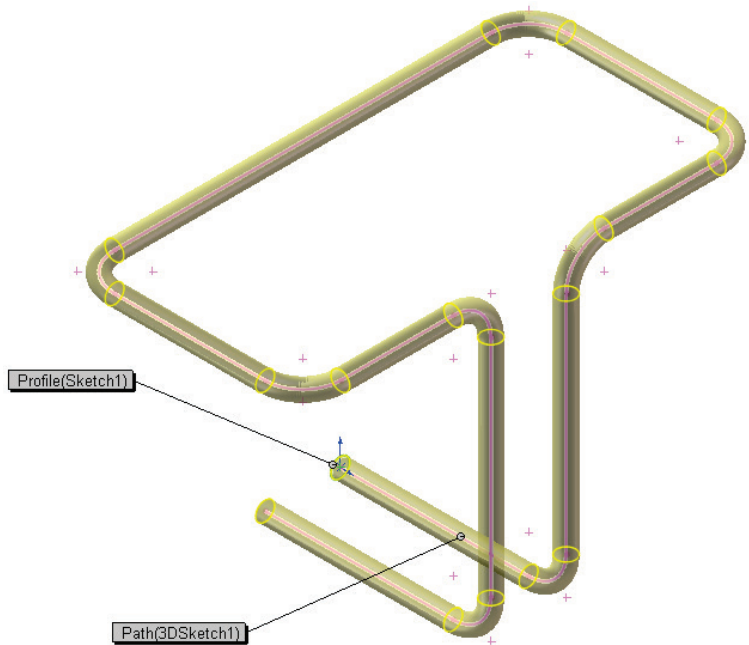
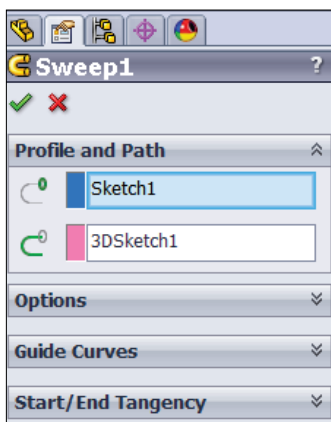
- Add a **Ø.250** dimension  to fully define the circle.
- **Exit** the Sketch  or select **Insert / Sketch**.

### **Note:**

- *The Sweep Profile should be Pierced or Coincident with the Sweep Path.*
- *The Swept Boss/Base command is only available when the sketch pencil is off.*

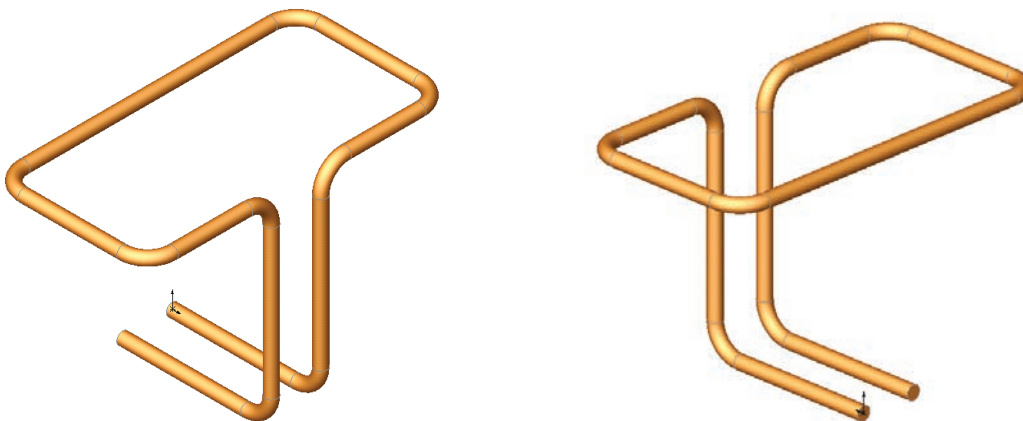
## 8. Creating the Swept feature:

- Click  or select **Insert / Boss-Base / Sweep**.
- Select the Circle as Sweep Profile  (Sketch1).
- Select the 3D Sketch to use as Sweep Path  (3Dsketch1).
- Click **OK** .



## 9. Saving your work:

- Select **File / Save As / 3D Sketch / Save**.



# Questions for Review

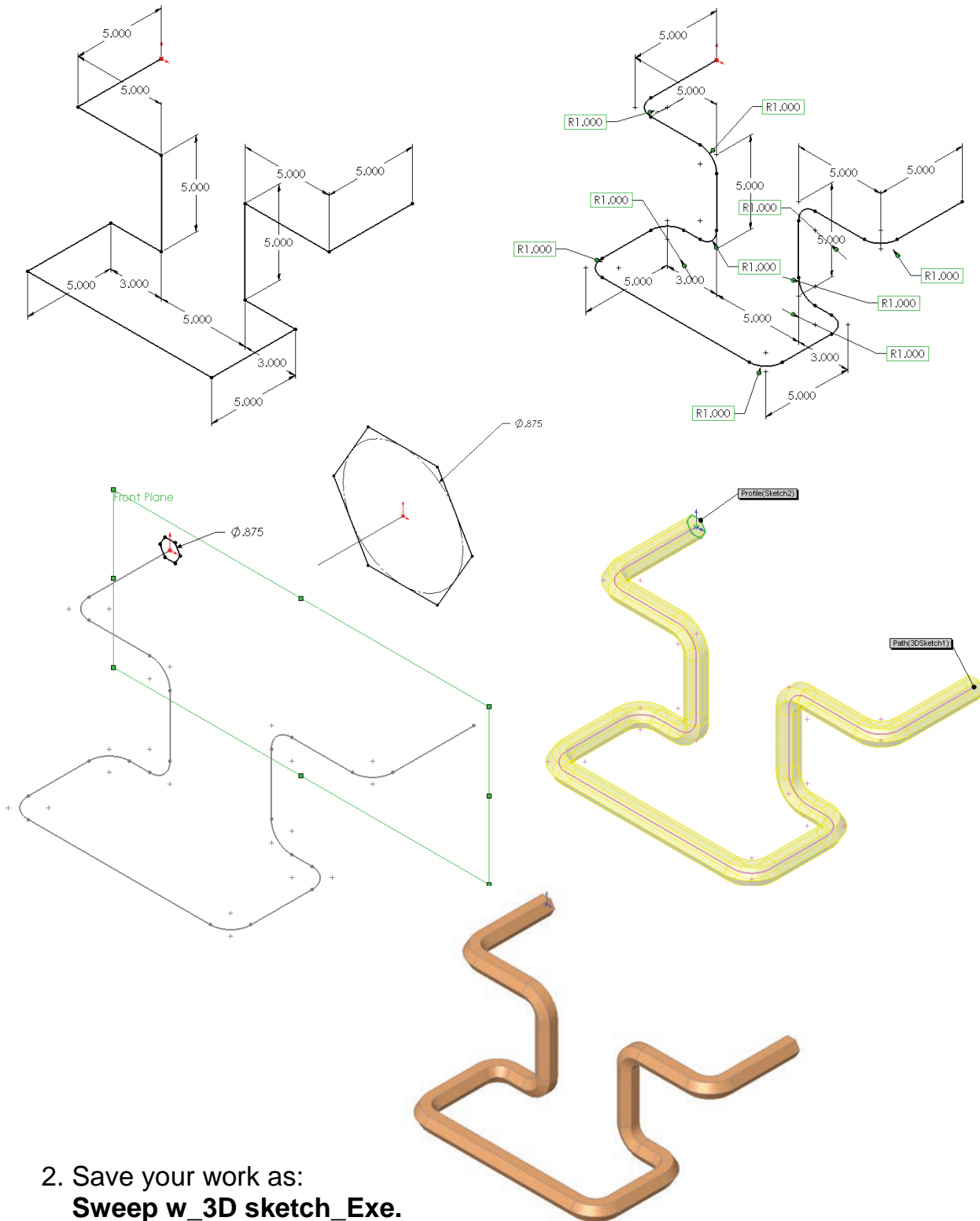
## Introduction to 3D Sketch

1. When using 3D Sketch you do not have to pre-select a plane as you would in 2D Sketch.
  - a. True
  - b. False
2. The space handle appears only after the first point of a line is started.
  - a. True
  - b. False
3. To switch to other planes in 3D Sketch mode, press:
  - a. Up Arrow
  - b. Down Arrow
  - c. TAB key
  - d. CONTROL key
4. Dimensions cannot be used in 3D Sketch mode.
  - a. True
  - b. False
5. Geometric Relations cannot be used in 3D Sketch mode.
  - a. True
  - b. False
6. All sketch tools in 2D Sketch are also available in 3D Sketch.
  - a. True
  - b. False
7. When adding sketch fillets, the option Keep Constrained Corner will create a virtual intersection point, but will not create a dimension.
  - a. True
  - b. False
8. 3D Sketch entities can be used as a path in a swept feature.
  - a. True
  - b. False

1. TRUE  
2. TRUE  
3. C  
4. FALSE  
5. FALSE  
6. FALSE  
7. FALSE  
8. TRUE

## Exercise: Sweep with 3D Sketch

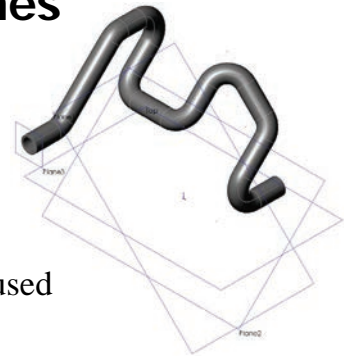
1. Create the part shown using 3D Sketch.



2. Save your work as:  
**Sweep w\_3D sketch\_Exe.**

## Exercise: 3D Sketch & Planes


A 3D sketch normally consists of lines and arcs in series, and splines. You can use a 3D sketch as a sweep path, as a guide curve for a loft or sweep, a centerline for a loft, or as one of the key entities in a routing system.



The following exercise demonstrates how several planes can be used to help define the directions of 3D Sketch Entities.

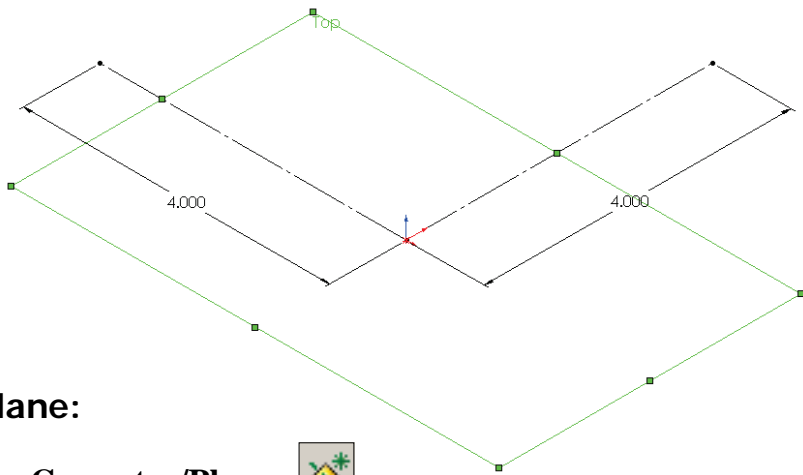
### 1. Sketching the reference Pivot lines:

- Select the Top plane and

open a new sketch  .


- Sketch **2 Centerlines** 

and add Dimensions  as shown.

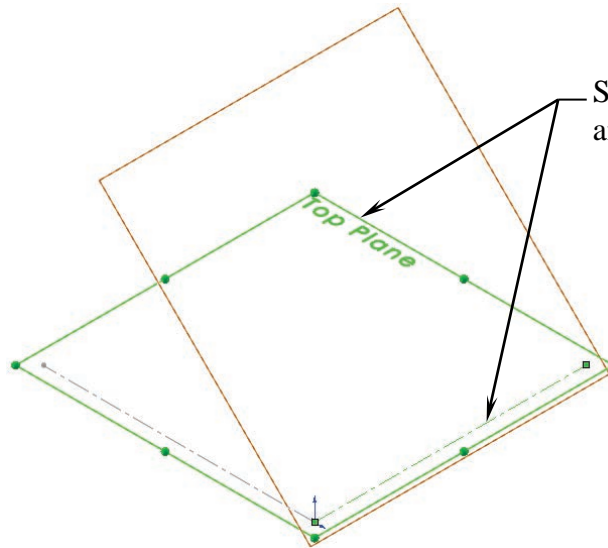
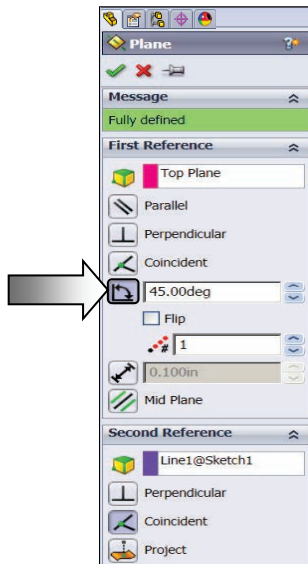


### 2. Creating the 1st 45° Plane:

- Select **Insert/Reference Geometry/Planes**  .

- Click the **At Angle** option and enter **45** for Angle  .



- Select the **top** plane and the **vertical line** as noted.

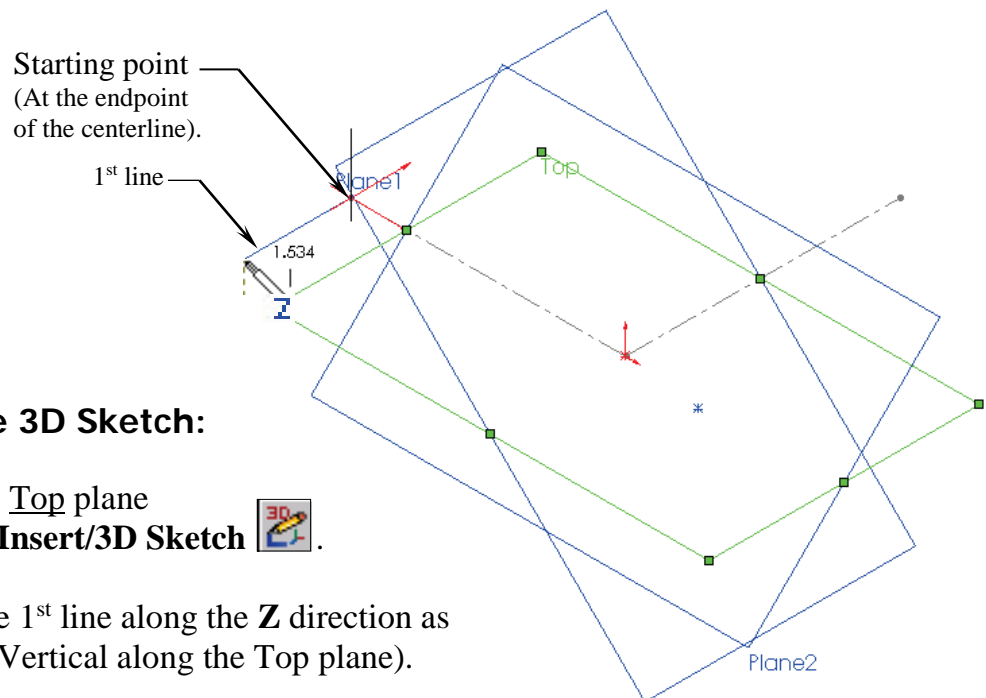
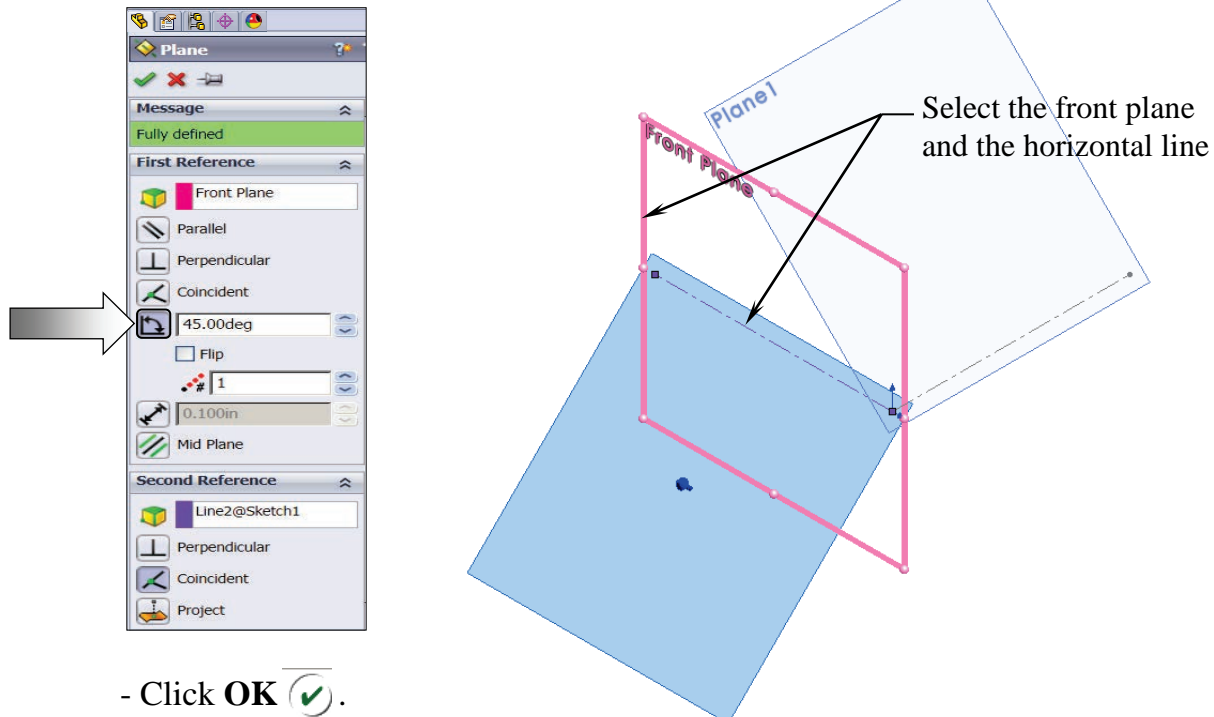


Select the top plane and the vertical line...


- Click **OK**  .

### 3. Creating the 2nd 45° Plane:


- Select **Insert/Reference Geometry/Planes** .
- Click the **At Angle** option and enter **45** for Angle .
- Select the **front** plane and the **horizontal line** as noted.



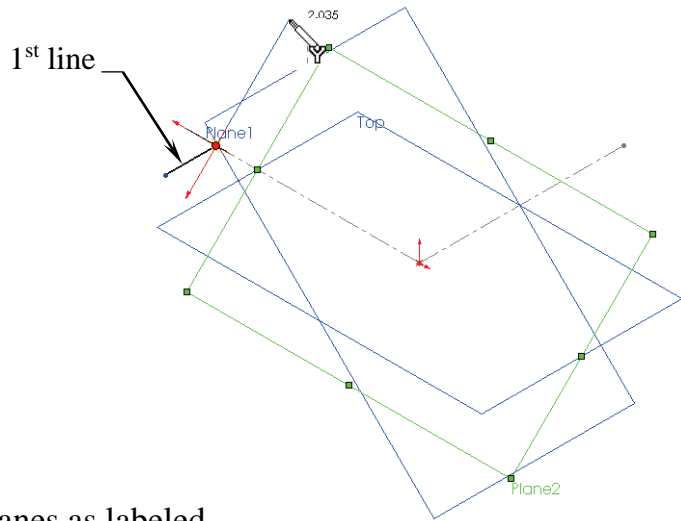
### 4. Creating the 3D Sketch:

- Select the Top plane and click **Insert/3D Sketch** .
- Sketch the 1<sup>st</sup> line along the **Z** direction as noted (or Vertical along the Top plane).

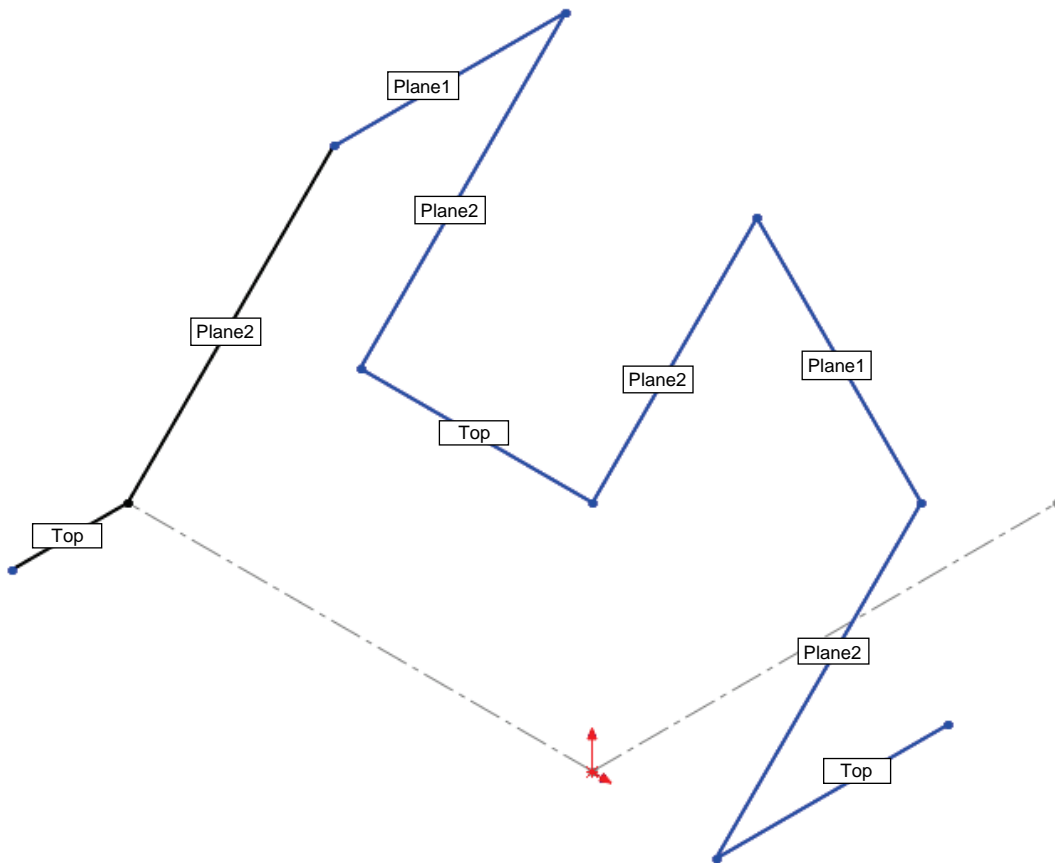
- Select the **Plane2** (45 deg.) from the Feature Manager tree and Sketch the 2<sup>nd</sup> line along the **Y** direction (watch the cursor feedback symbol).

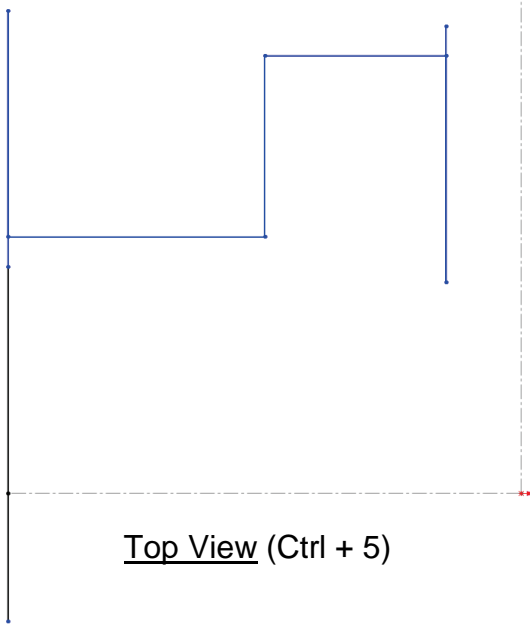
 **Switching Planes**

While sketching the lines, hold the **Control** key and click a plane to switch from one plane to another, or simply select them from the Feature tree each time.

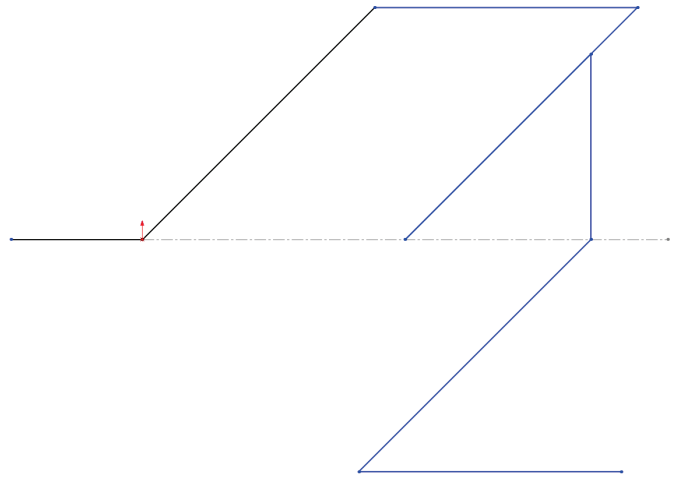


- Sketch the rest of lines on the planes as labeled.
- For clarity, hide all the planes (select the **View** menu and click off **Planes**). We will select the planes from the FeatureManager tree when needed.



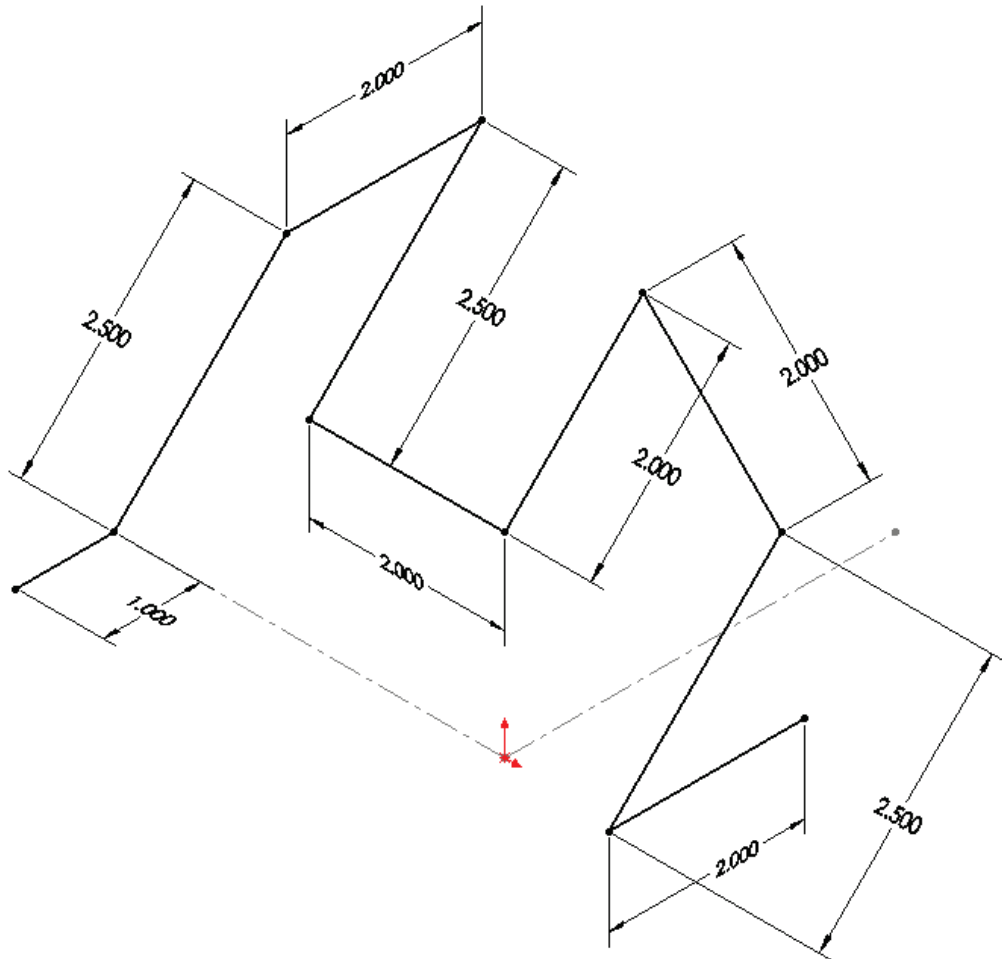


Top View (Ctrl + 5)



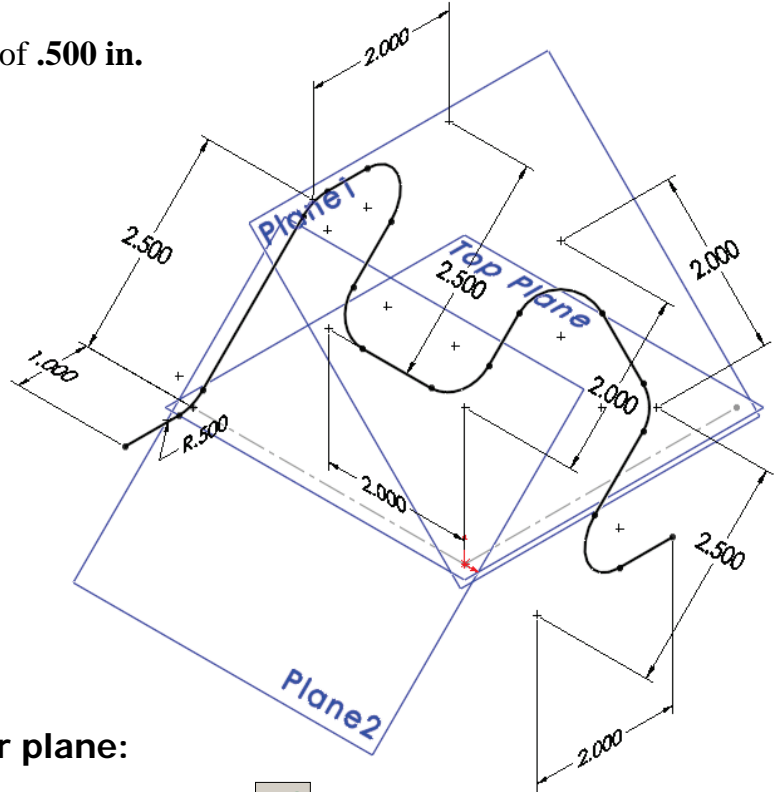
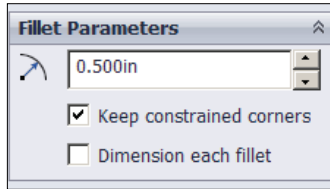
Right View (Ctrl + 4)

- Add Dimensions  to fully define the sketch.






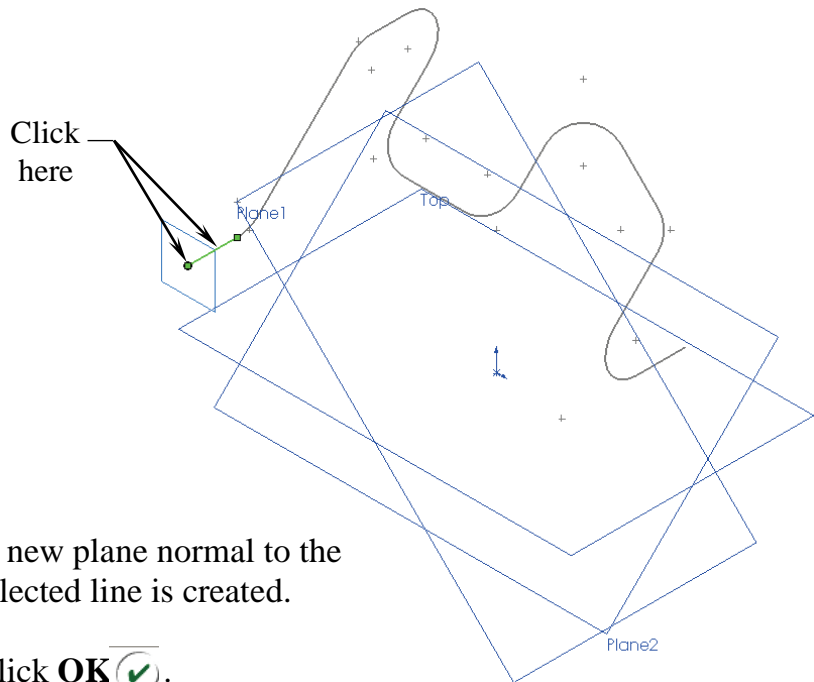
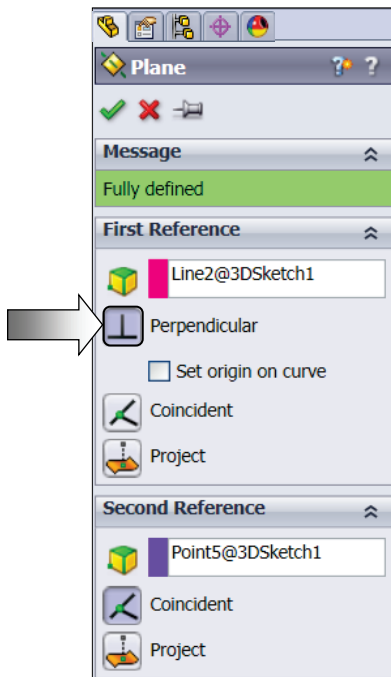
- Add **Sketch Fillets**  of **.500 in.** to all corners.




- **Exit** the 3D Sketch or press **Ctrl+Q**.


## 5. Creating a Perpendicular plane:


- Select **Insert/Reference Geometry/Plane** .
- Select the **line** and its **endpoint** approximately as shown.
- The **Perpendicular** option should be selected by default.

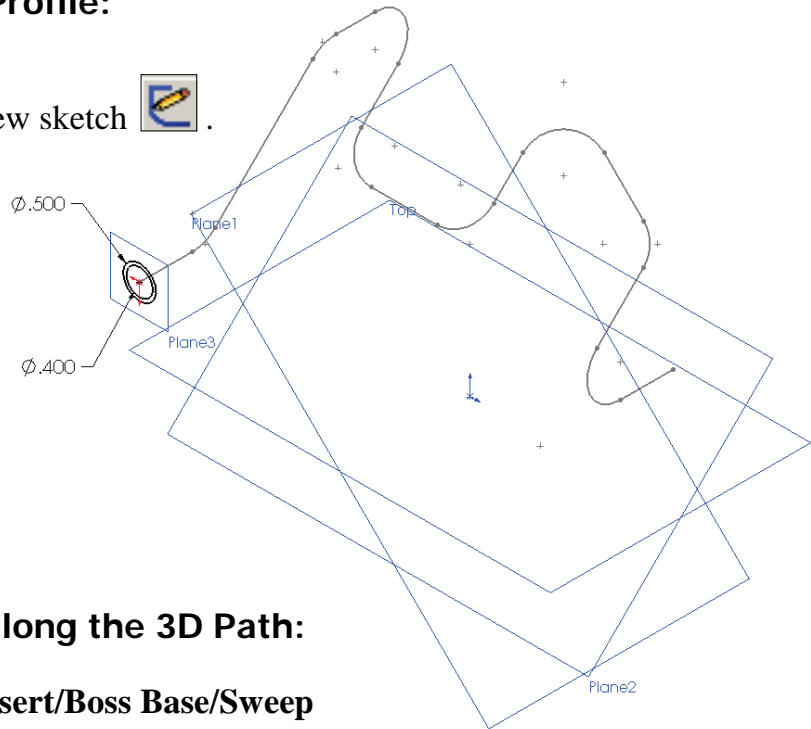


- A new plane normal to the selected line is created.
- Click **OK** .




## 6. Sketching the Sweep Profile:

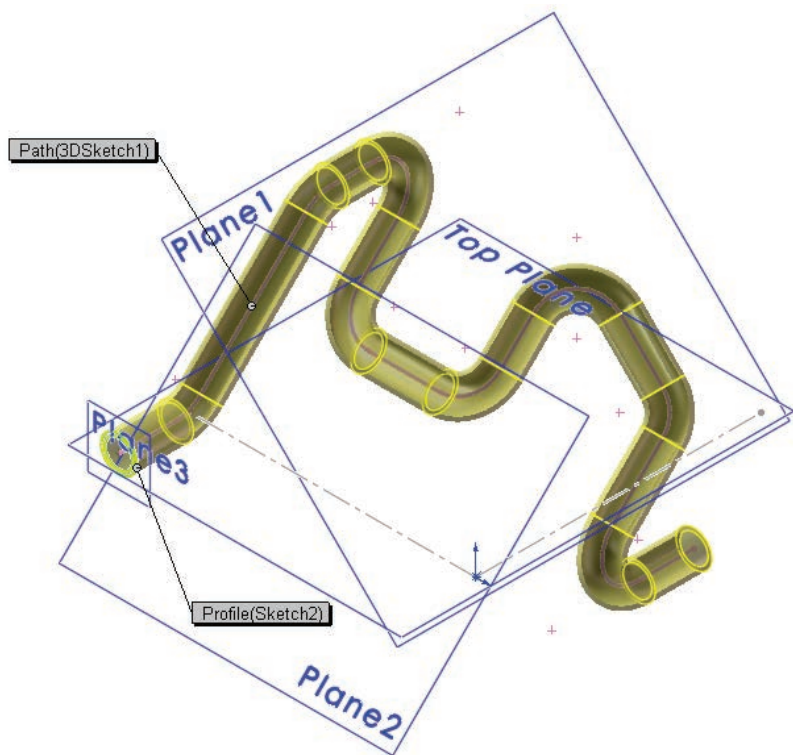
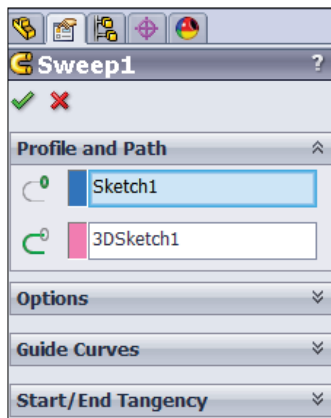
- Select the new plane (Plane3) and open a new sketch .

- Sketch **2 Circles**  on the same center and add the dimensions as shown to fully define the sketch.



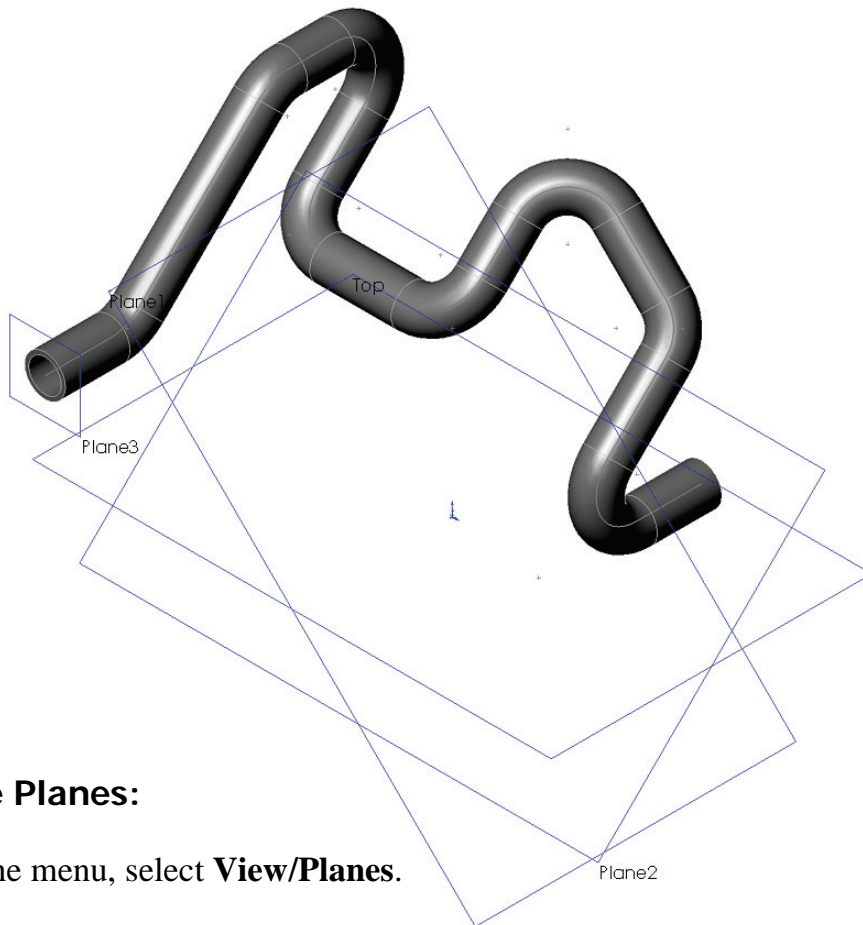
## 7. Sweeping the Profile along the 3D Path:

- Click  or Select **Insert/Boss Base/Sweep**
- Select the **Circles** as the Sweep Profile .
- Select the **3D Sketch** as the Sweep Path .



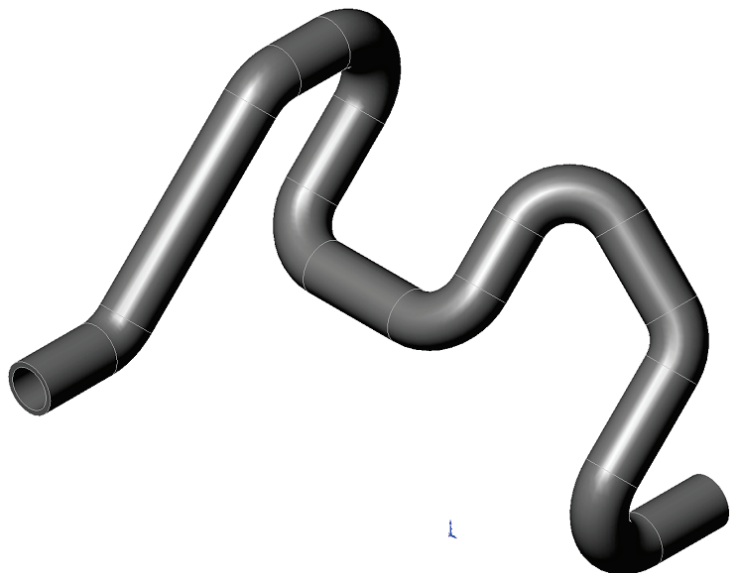
- Click **OK** .

- The resulting Swept feature.



### 8. Hiding the Planes:

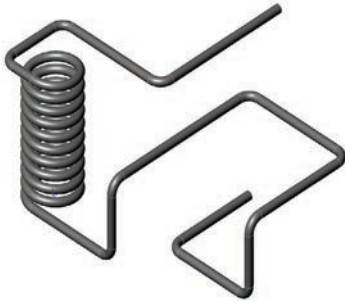
- From the menu, select **View/Planes**.
- The planes are temporarily put away from the scene.



### 9. Saving your work:

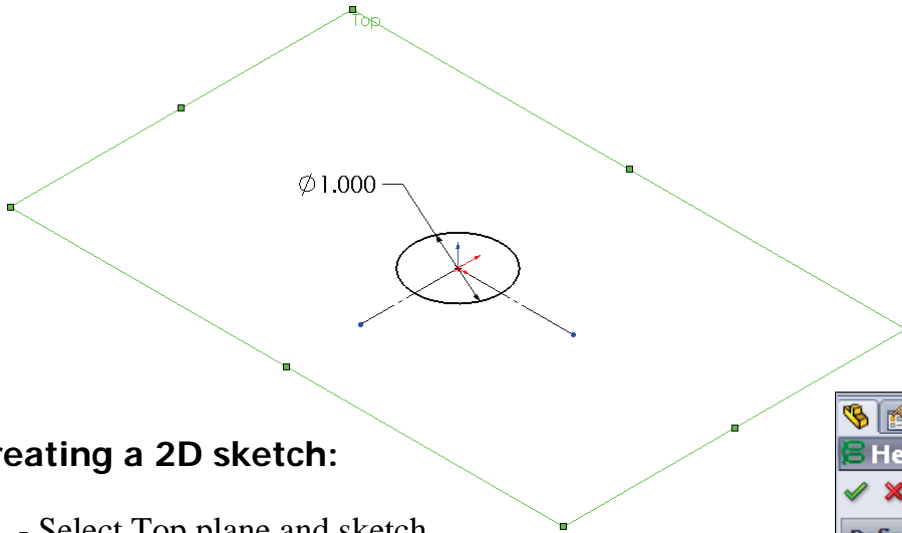
- Click **File/Save As: 3D Sketch\_Planes**.
- Click **Save**.

## Exercise: 3D Sketch & Composite Curve



A 3D sketch normally consists of lines and arcs in series and Splines. You can use a 3D sketch as a sweep path, as a guide curve for a loft or sweep, a centerline for a loft, or as one of the key entities in a routing system.

The following exercise demonstrates how several 3D Sketches can be created, combined into 1 continuous Composite Curve, and used as a Sweep Path.



### 1. Creating a 2D sketch:

- Select Top plane and sketch

a 1.00" dia. Circle 

and 2 Centerlines .

### 2. Creating a Helix:

- Select **I**nsert/**C**urve/

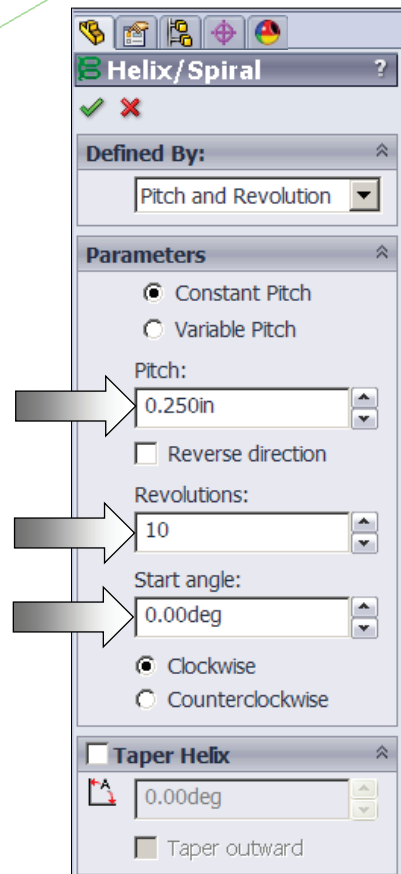
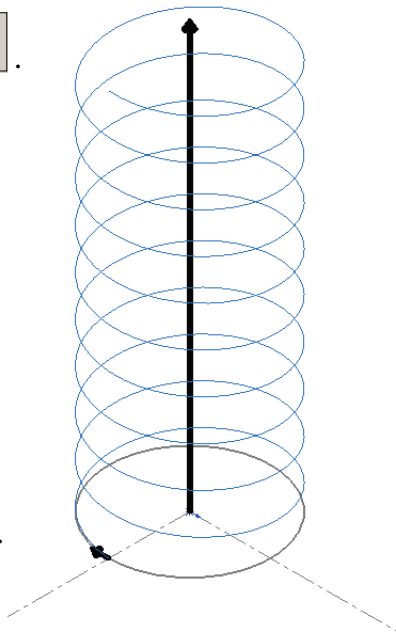
**H**elix-**S**piral .

- Pitch: .250 in.


- Revolution: 10.


- Starting Angle: 0 deg.

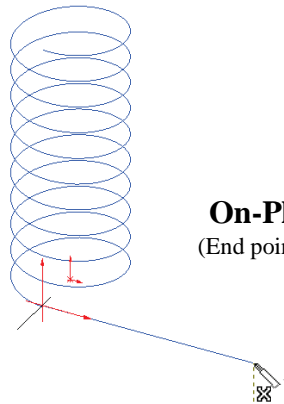
- Click **OK** .



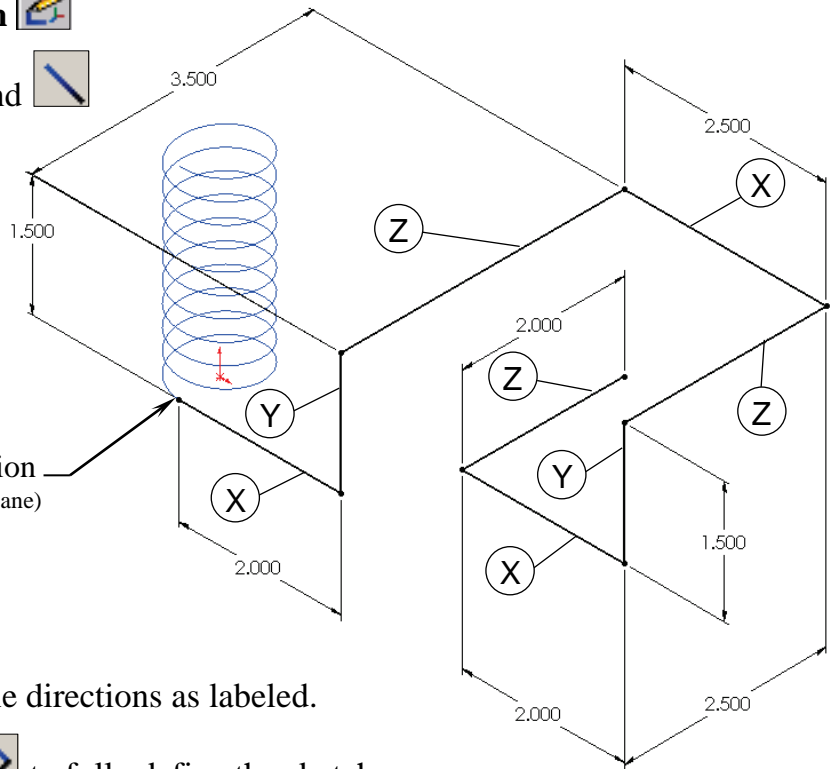
### 3. Creating the 1st 3D sketch:

- Select **Insert/3D Sketch** 


- Select the **Line** command  and sketch the 1<sup>st</sup> line along the X direction.

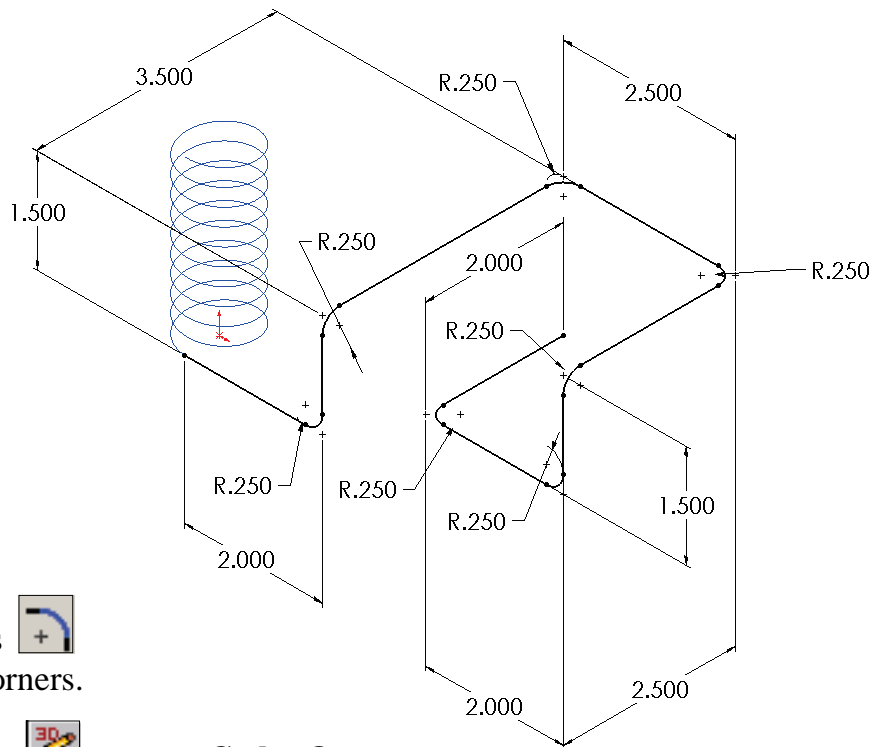
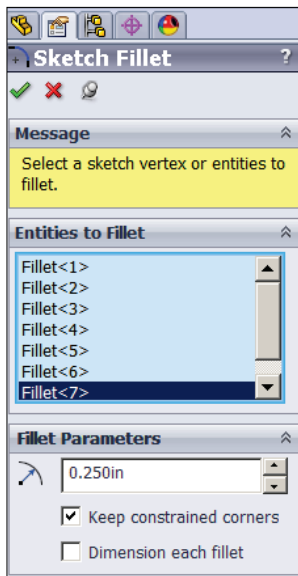



**On-Plane** relation  
(End point & Right plane)



- Add other lines in the directions as labeled.



- Add Dimensions  to fully define the sketch.

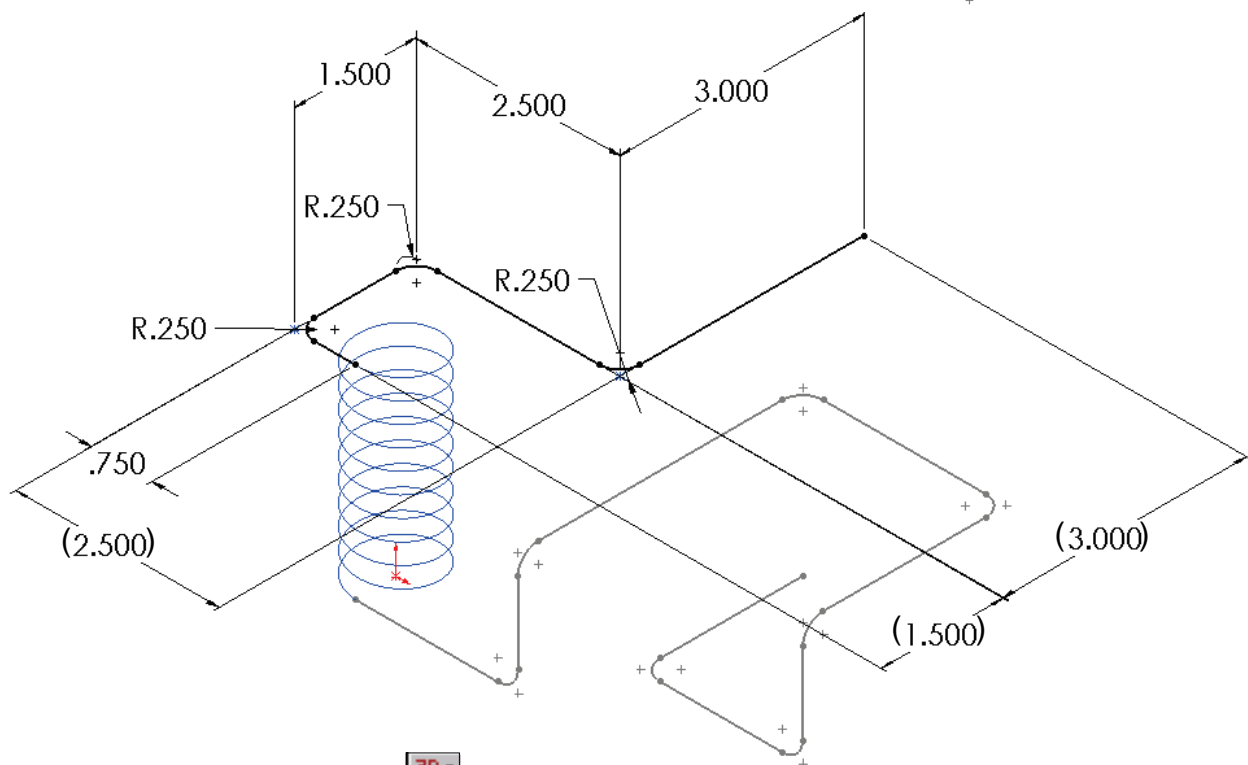
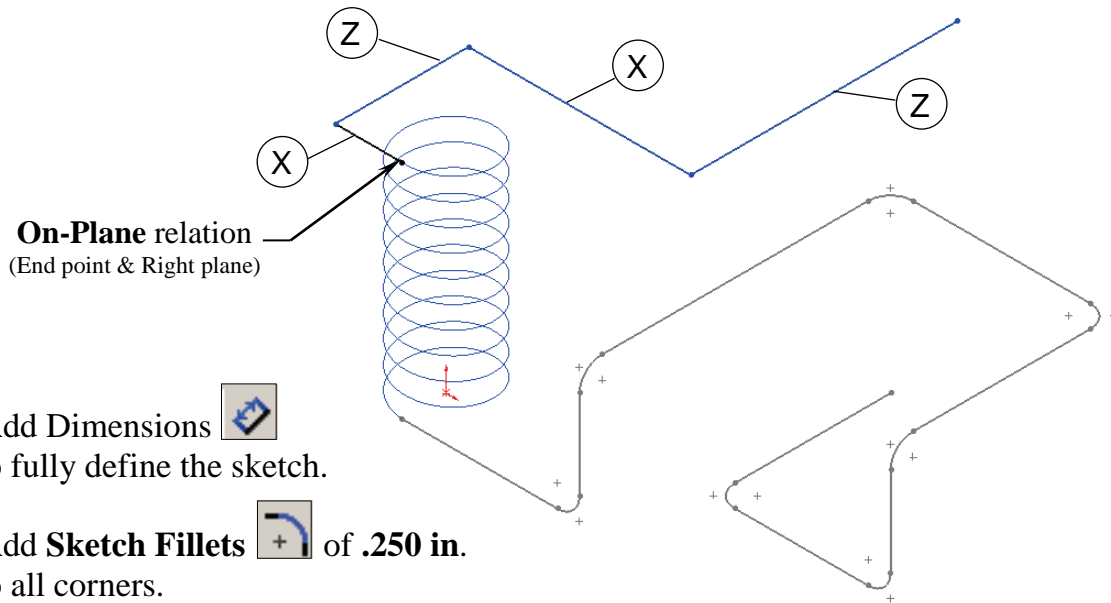


- Add Sketch Fillets  of **.250 in.** to all corners.

- **Exit** the 3D Sketch  or press **Ctrl + Q**.


#### 4. Creating the 2nd 3D sketch:

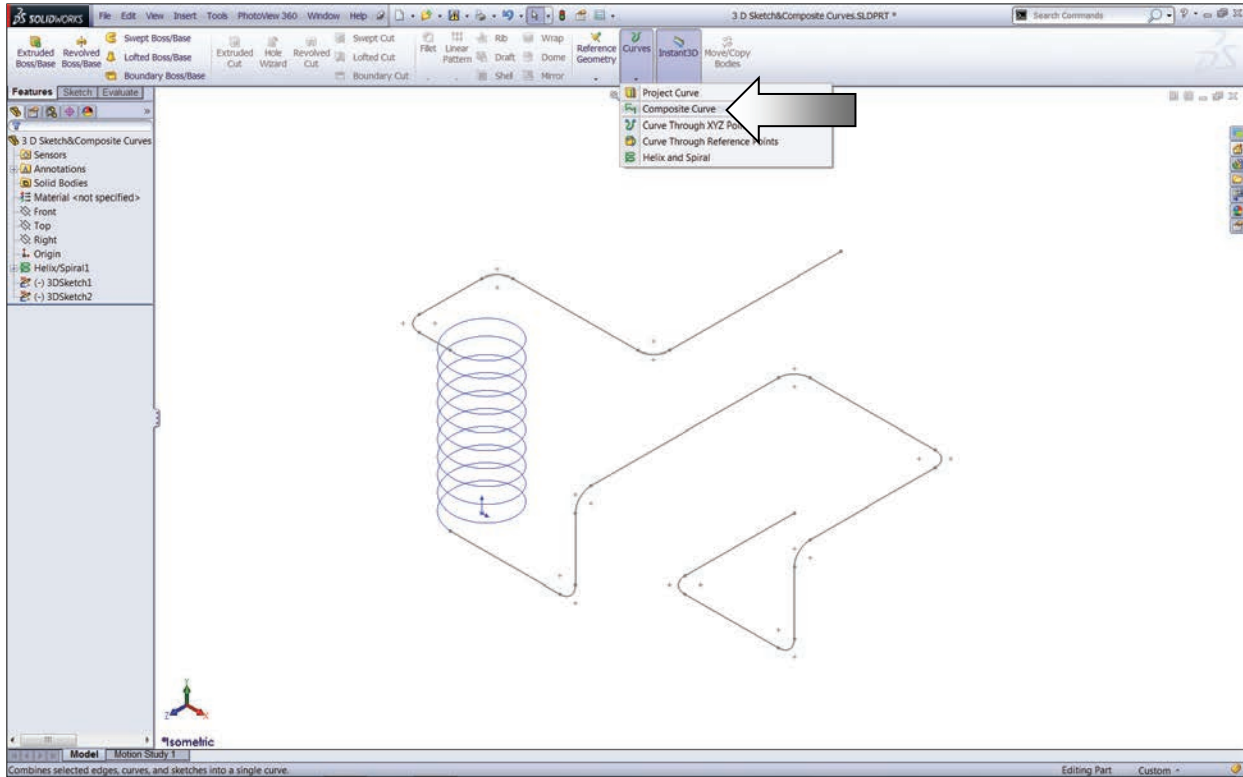
- Select **Insert/3D Sketch** .
- Select the **Line** command  and sketch the 1<sup>st</sup> line along the X direction.
- Sketch the rest of the lines following their direction shown below.



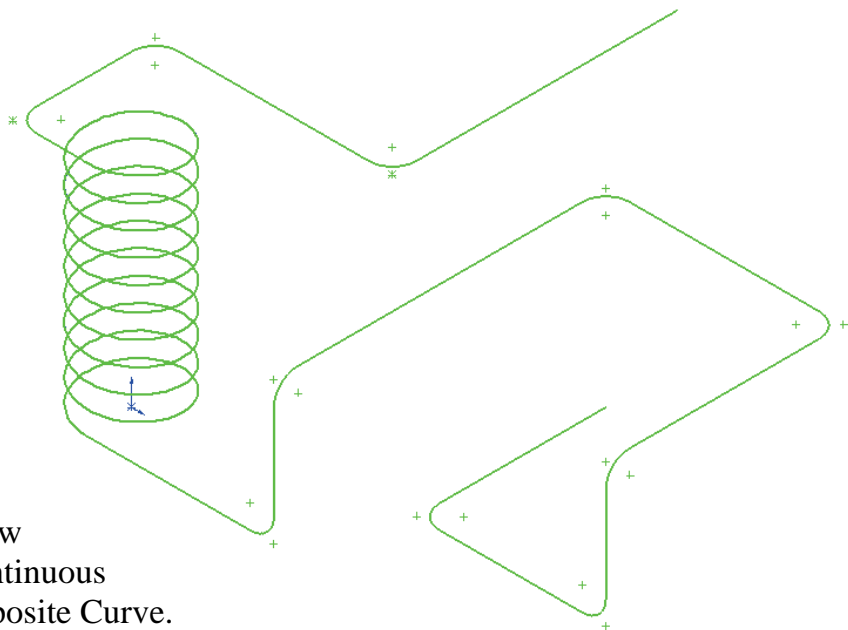
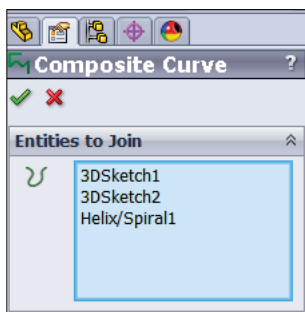
- **Exit** the 3D Sketch  or press **Ctrl+Q**.


## 5. Combining the 3 sketches into 1 curve:

- Select **Insert/Curve/Composite**  or select it from the Curves button on the Features toolbar.



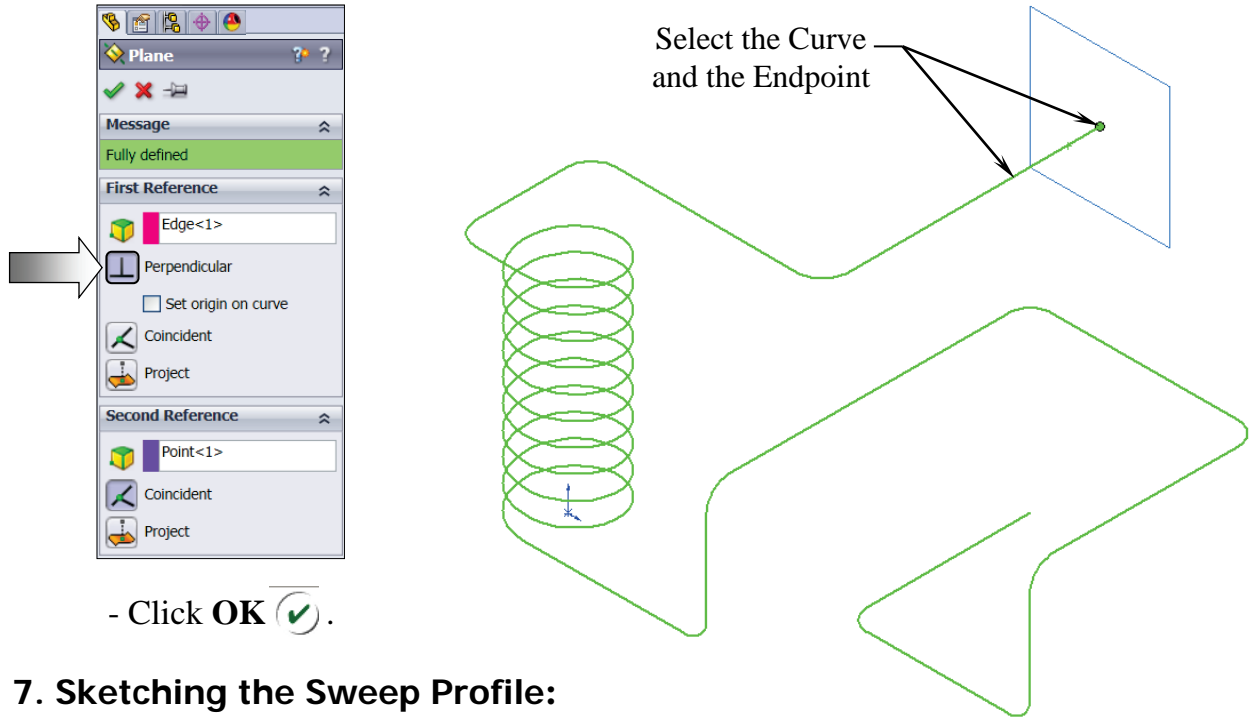
- Select the 3 Sketches either from the Feature Manager tree or directly from the graphics area.



- Click **OK** .
- The sketches are now combined into 1 continuous curve called a Composite Curve.




## 6. Creating a new work plane:

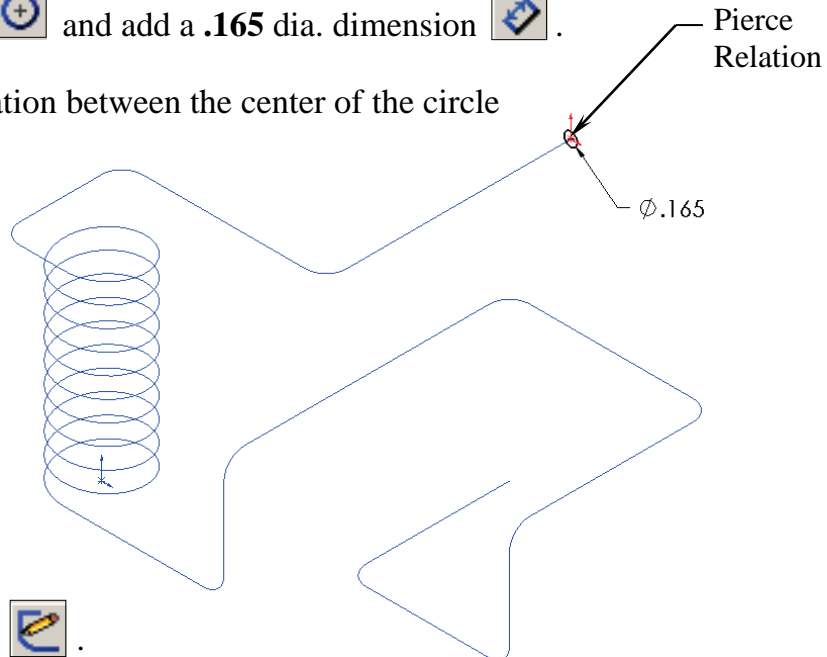
- Select **Insert/Reference Geometry/Plane** .
- Select the **edge** and **endpoint** as noted, the Perpendicular should be selected.



- Click **OK** .

## 7. Sketching the Sweep Profile:




- Select the new plane (Plane1) and open a new sketch .
- Sketch a **Circle**  and add a **.165** dia. dimension .
- Add a **Pierce** relation between the center of the circle and the curve.

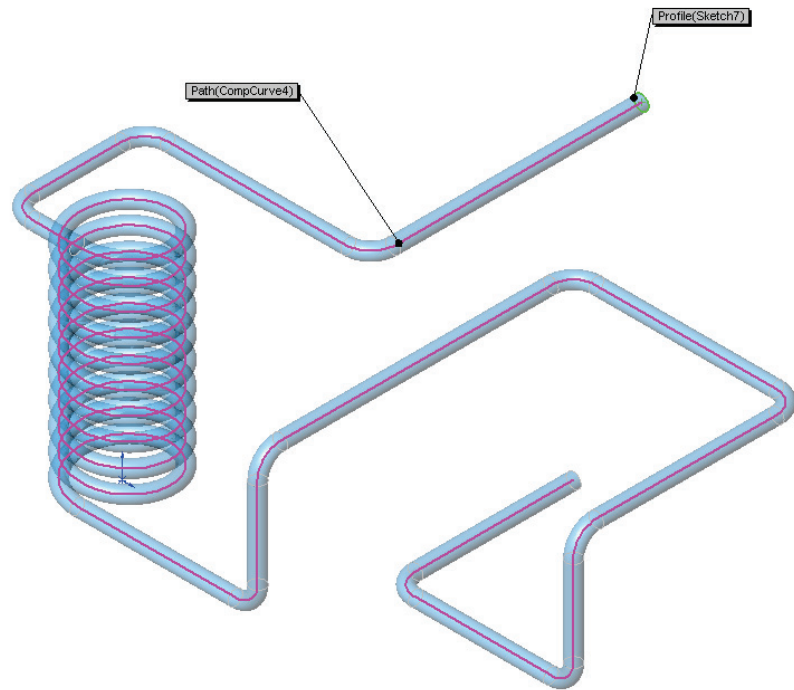
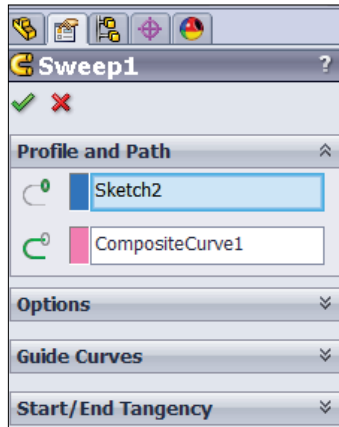


- **Exit** the Sketch .

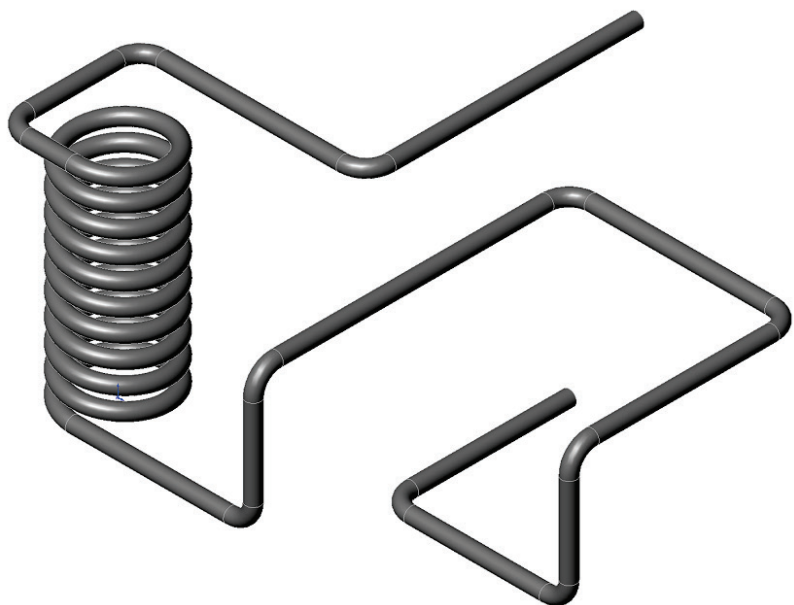


## 8. Sweeping the Profile along the Path:

- Select **Insert/Boss Base/ Sweep** .
- Select the **Circle** as the Sweep Profile .
- Select the **Composite Curve** as the Sweep Path .



- Click **OK** .



## 9. Saving your work:

- Click **File/Save As**.
- Enter **3D Sketch\_ Composite Curve**.
- Click **Save**.

