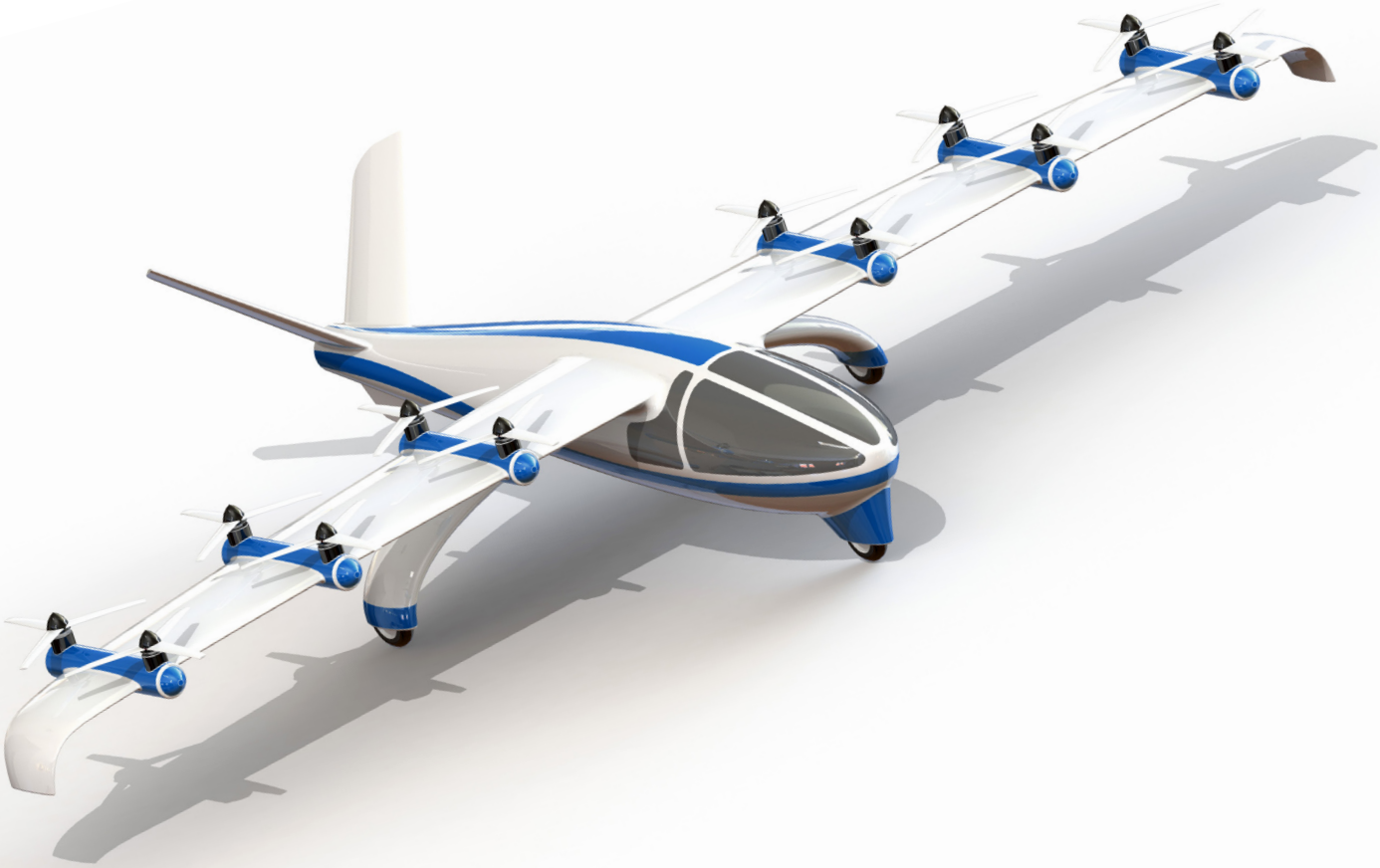
SOLIDWORKS® 2024 Advanced Techniques

Mastering Parts, Surfaces, Sheet Metal, SimulationXpress, Top Down Assemblies, Core & Cavity Molds



Paul Tran CSWE, CSWI

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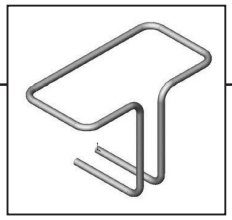
CHAPTER 1

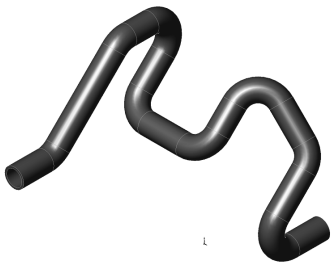
Introduction to 3D Sketch



Introduction to 3D Sketch

SOLIDWORKS has 3D sketch capabilities. 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

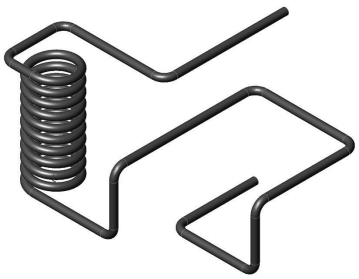
Z Coordinate

Curvature (Spline curvature at the frame point)

Tangency (In the **XY** plane)

Tangency (In the **XZ** plane)

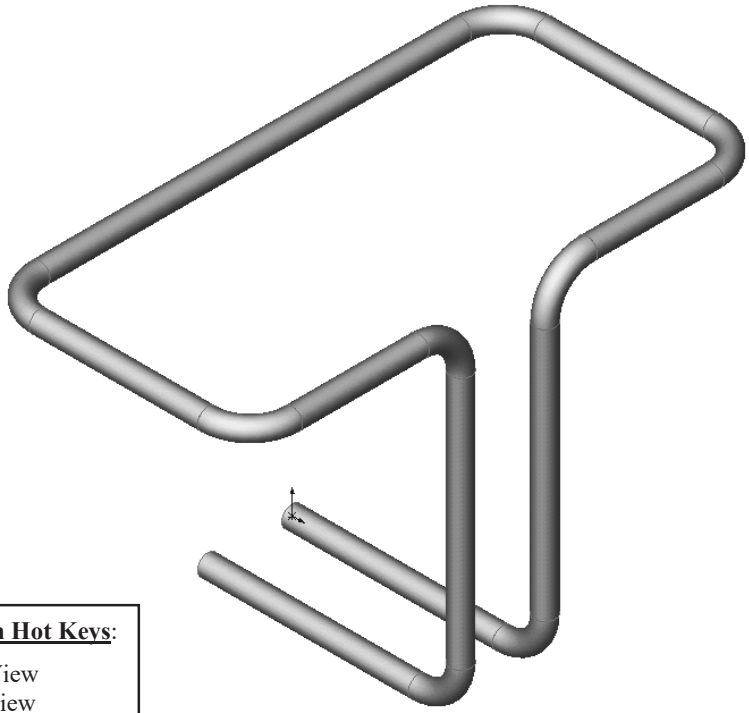
Tangency (In the **YZ** plane)



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:

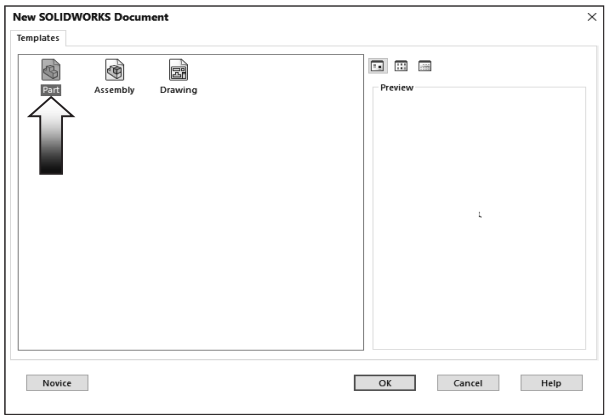
Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

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:



Click **File / New.**

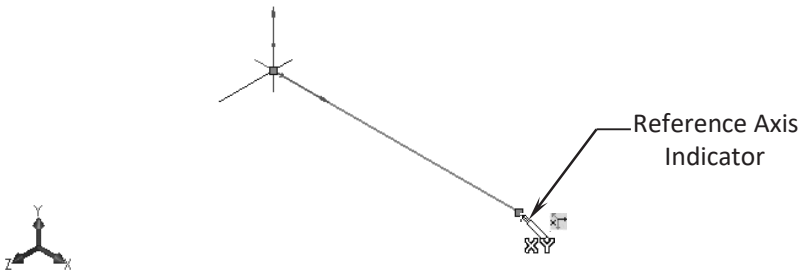
Select the **Part** template and click **OK**.

Set the Units to **IPS**, 3 decimals.

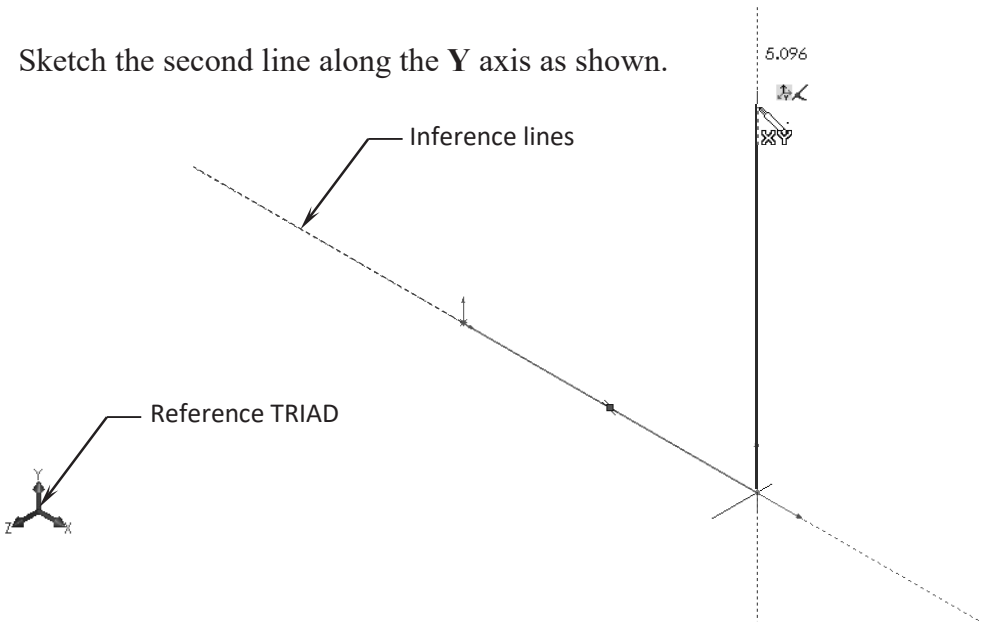
2. Creating a 3D Sketch:

Click or select **Insert / 3D Sketch** and change to **Isometric view** (Control+7).

Select the **Line** tool and sketch the first line along the **X** direction. A yellow symbol appears next to the mouse cursor when the line is drawn along the X axis; this indicates an **Along X** relation (horizontal) is being added to the line.



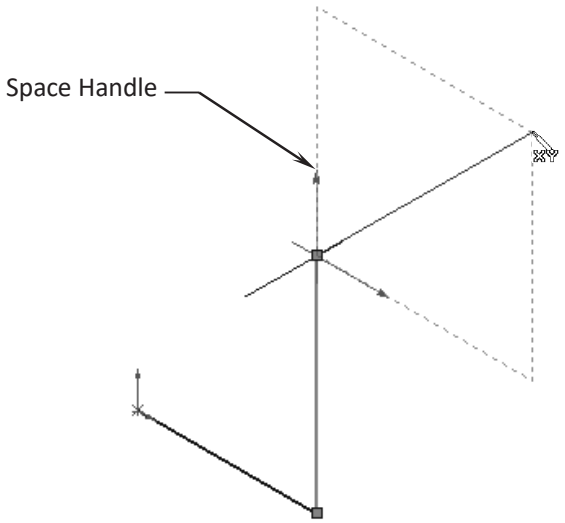
Sketch the second line along the **Y** axis as shown.



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.

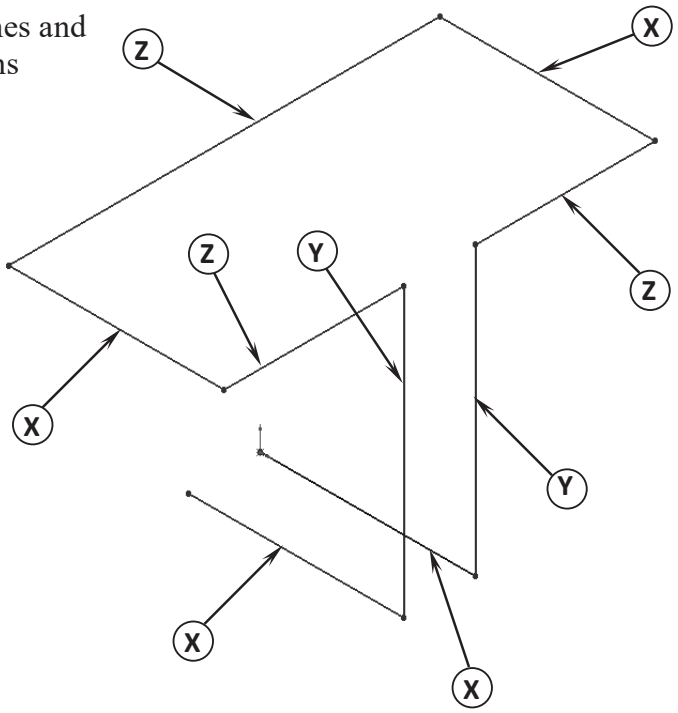


The TAB key

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

4. Completing the profile:

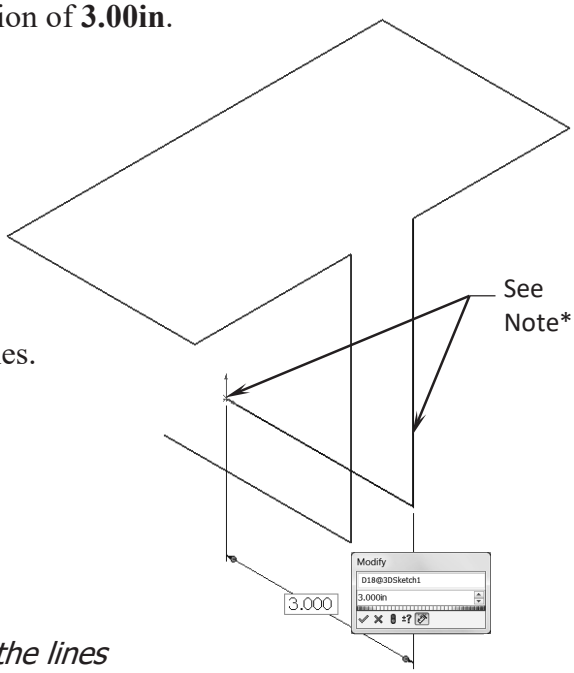
Sketch the other lines and follow the directions as labeled, press the **TAB** key if needed to change the direction.



5. Adding dimensions:

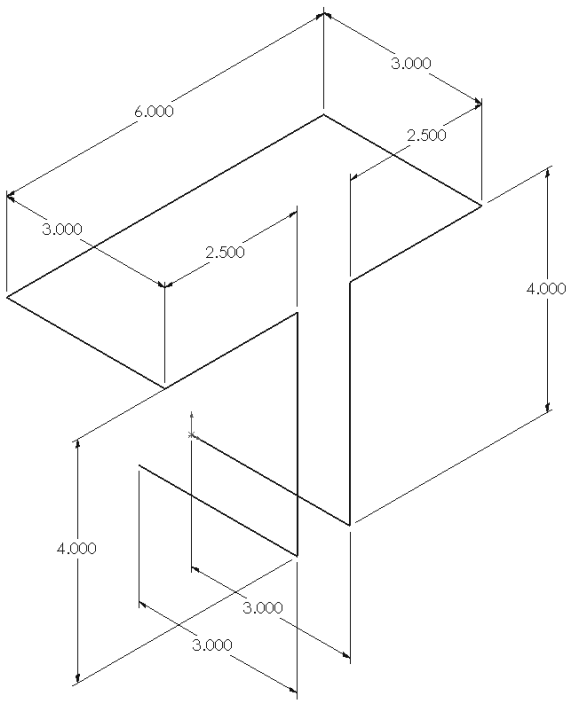
Click **Smart Dimension** or select **Tools / Dimensions / Smart Dimension**.

Click the first line and enter a dimension of **3.00in**.



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.

Note: *To make the dimensions parallel to the lines as shown, select the line and an endpoint instead of selecting just the endpoints.*



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

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

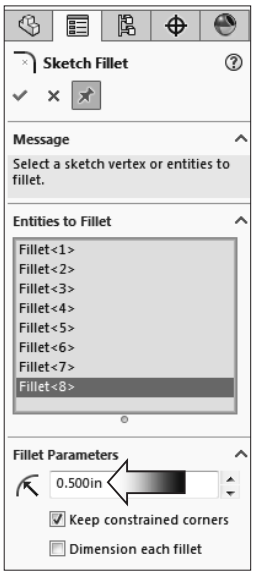
6. Adding the Sketch Fillets:

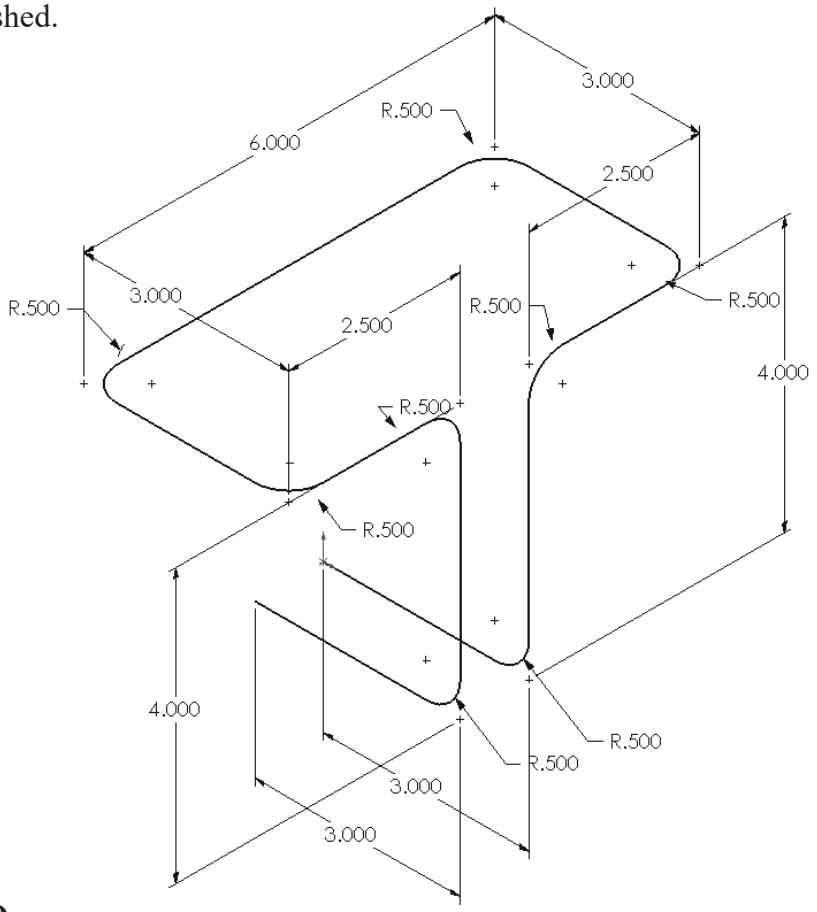
Click **Fillet** on the Features toolbar or select **Tools / Sketch Tools / Fillet**.

Add **.500”** fillets to <u>all</u> the intersections as indicated.

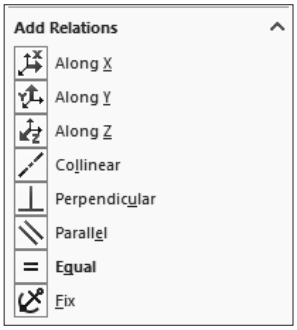
Enable the **Keep Constrained Corner** check box (to maintain the virtual intersection point if the vertex has dimensions or relations).

Click **OK** when finished.





<u>Exit</u> the 3D Sketch
or press **Control + Q**.



Geometric Relations

Geometric Relations such as Along X, Y, Z and Equal can also be used to replace some of the duplicate dimensions.

7. Creating the Swept feature:

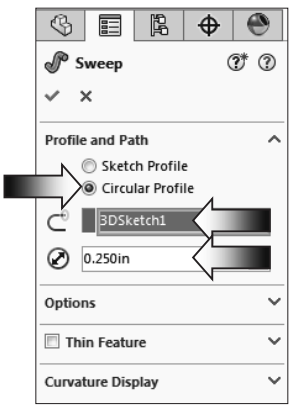
The Circular Profile option allows you to create a solid rod or hollow tube along a path, edge, or curve directly on a model without having to sketch the circular profile. This enhancement is available for Swept Boss/Base, Swept Cut, and Swept Surface features.

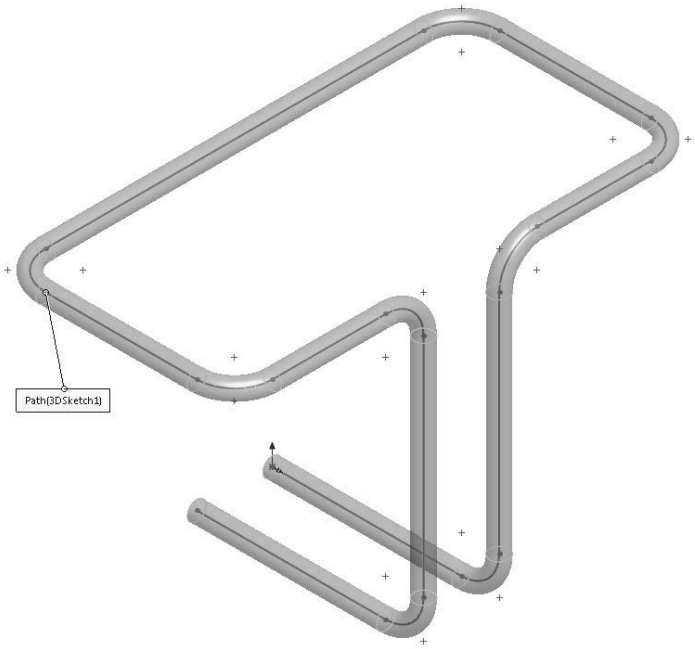
Click or select **Insert / Boss-Base / Sweep**.

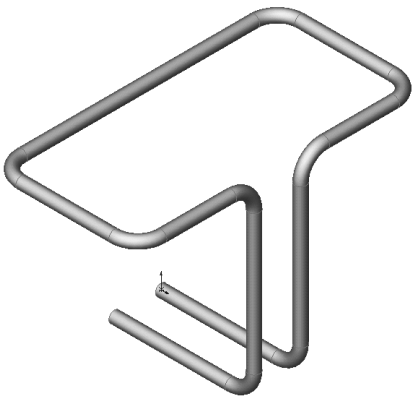
Select the **Circle Profile** option and enter **.250in** for the diameter of the profile .

Select the **3D Sketch** for Sweep Path (3Dsketch1).

Click **OK**.







8. Saving your work:

Select **File / Save As**.

Enter: **3D Sketch** for the file name.

Click **Save**.

Questions for Review

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 (or direction) 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

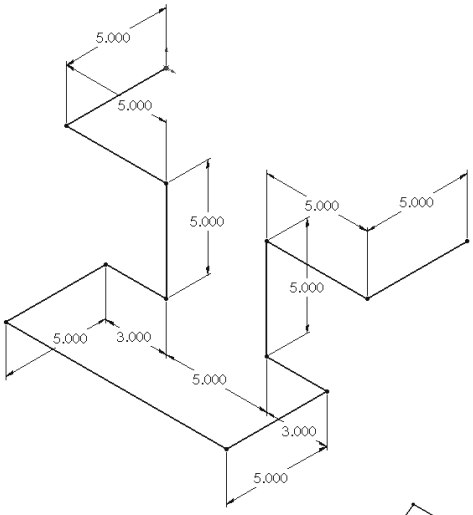
7. When adding sketch fillets, the option Keep Constrained Corner will create a virtual intersection point but will not create a radius dimension.
   a. True
   b. False

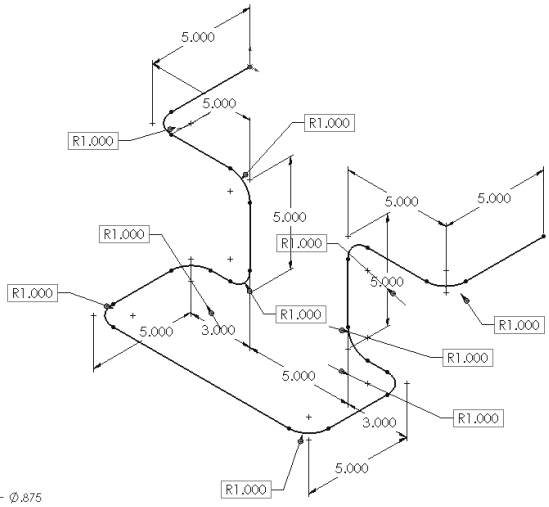
8. 3D Sketch entities can be used as a path in a swept feature.
   a. True
   b. False

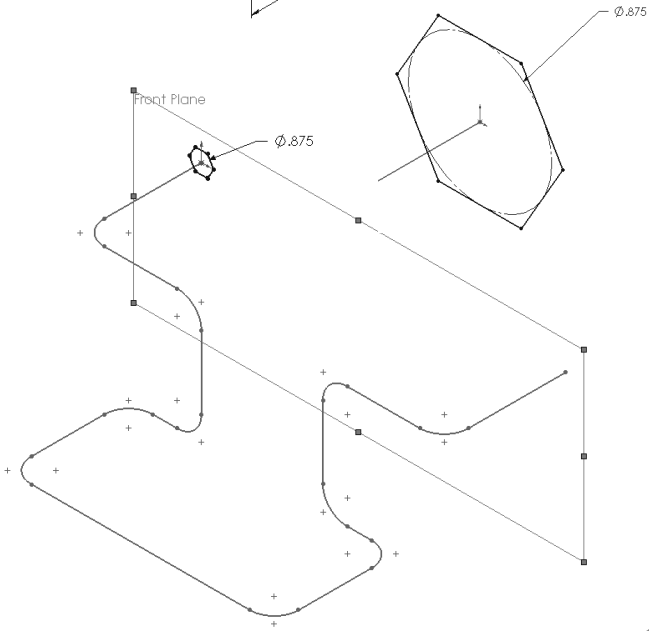
1. TRUE 2. TRUE
3. C 4. FALSE
5. FALSE 6. FALSE
7. FALSE 8. TRUE

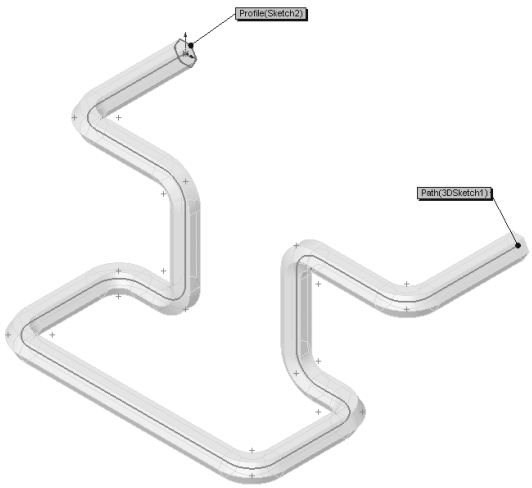
Exercise: Sweep with 3D Sketch

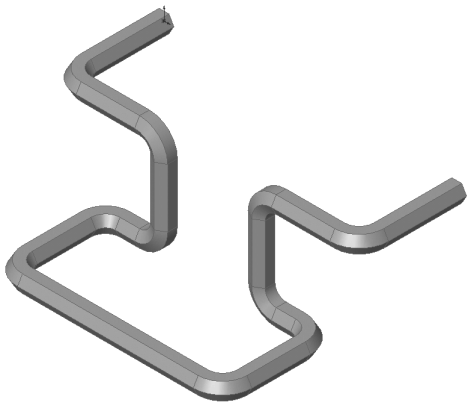
1. Create the part shown below using 3D Sketch.





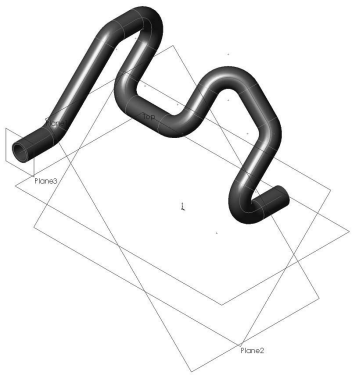






2. Save your work as:
   Sweep_3D Sketch_Exe.

Exercise: 3D Sketch & Planes



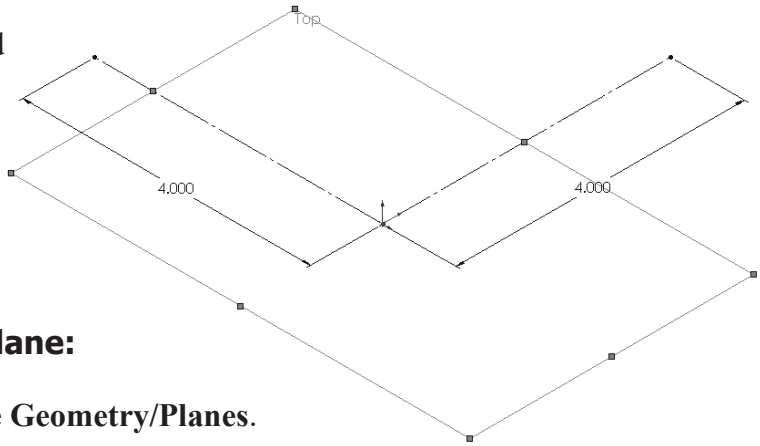
A 3D sketch normally consists of lines and arcs in series, and splines. You can use a 3D sketch as a sweep path, as a guide curve for a loft or sweep, a centerline for a loft, or as one of the key entities in a routing system.

The following exercise shows 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.

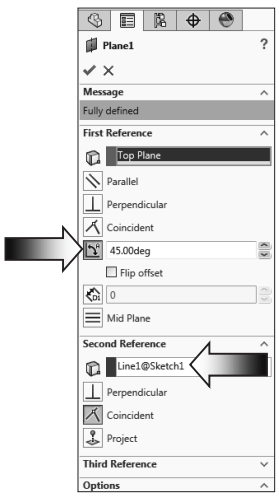


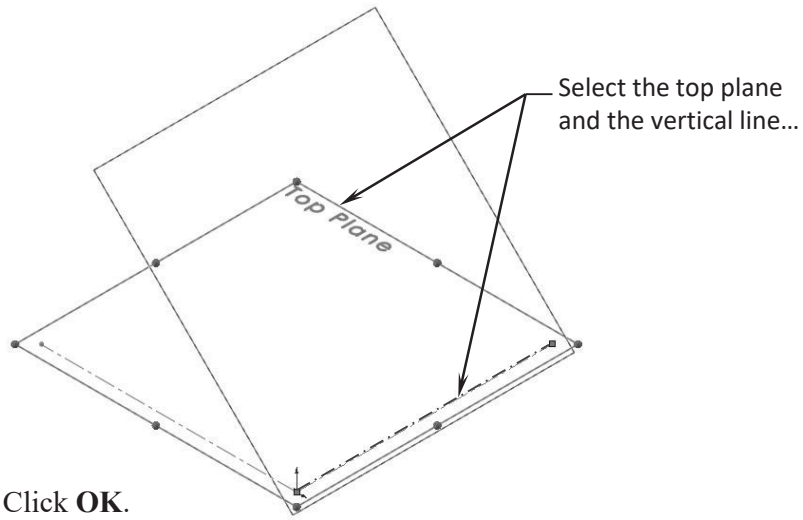
2. Creating the 1st 45º Plane:

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

Click the **At Angle** button and enter **45** for Angle (arrow).

Select the **Top** plane and the **Vertical line** as noted.





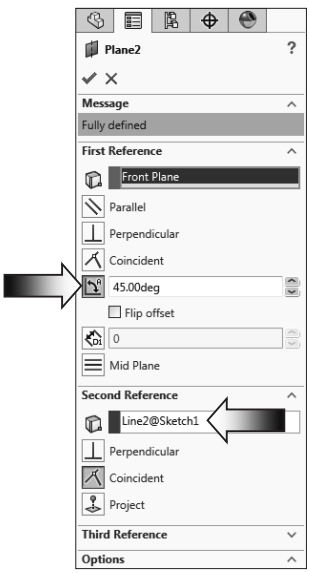
Click **OK**.

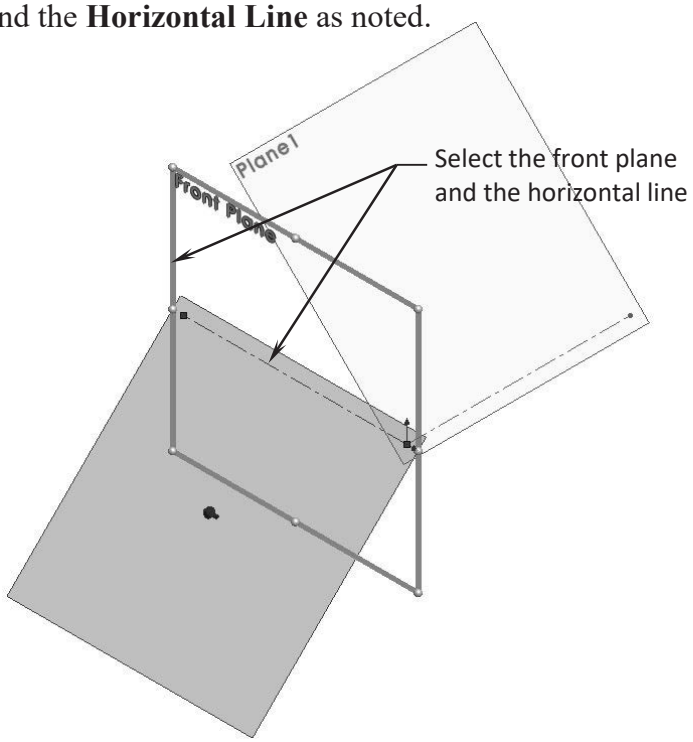
3. Creating the 2nd 45° Plane:

Click the **Plane** command or select **Insert/Reference Geometry/Planes** .

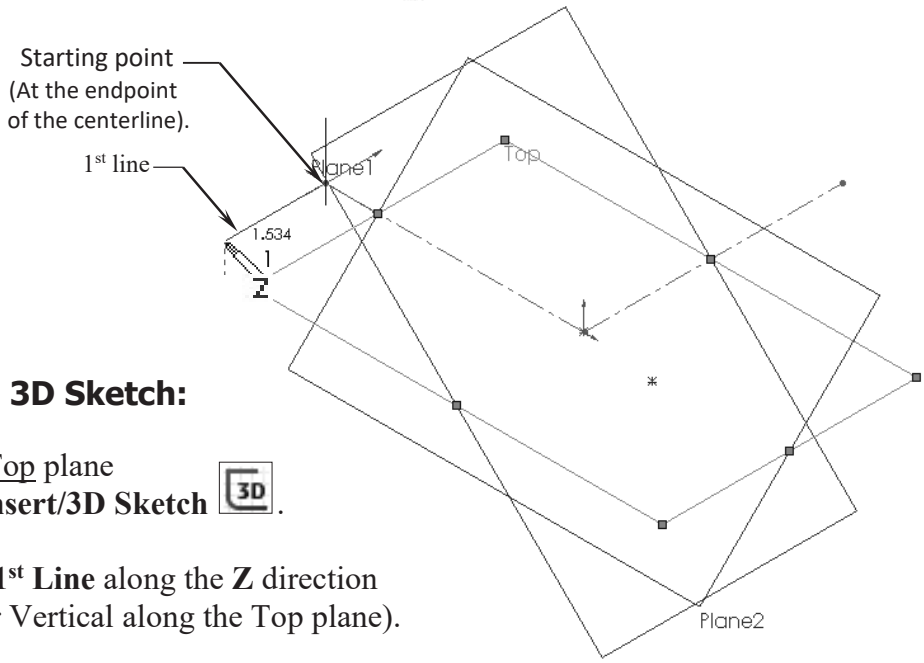
Click the **At Angle** option and enter **45** for Angle (arrow).

Select the **Front** plane and the **Horizontal Line** as noted.





Click **OK**.



4. Creating the 3D Sketch:

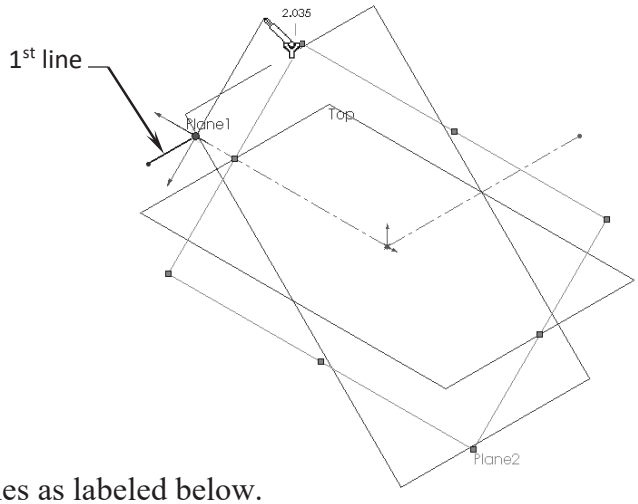
Select the Top plane
and click **Insert/3D Sketch** .

Sketch the **1st Line** along the **Z** direction
as noted (or Vertical along the Top plane).

Select **Plane2** (45 deg.) from the FeatureManager tree and Sketch the **2nd Line** along the **Y** direction (watch the cursor feedback symbol).

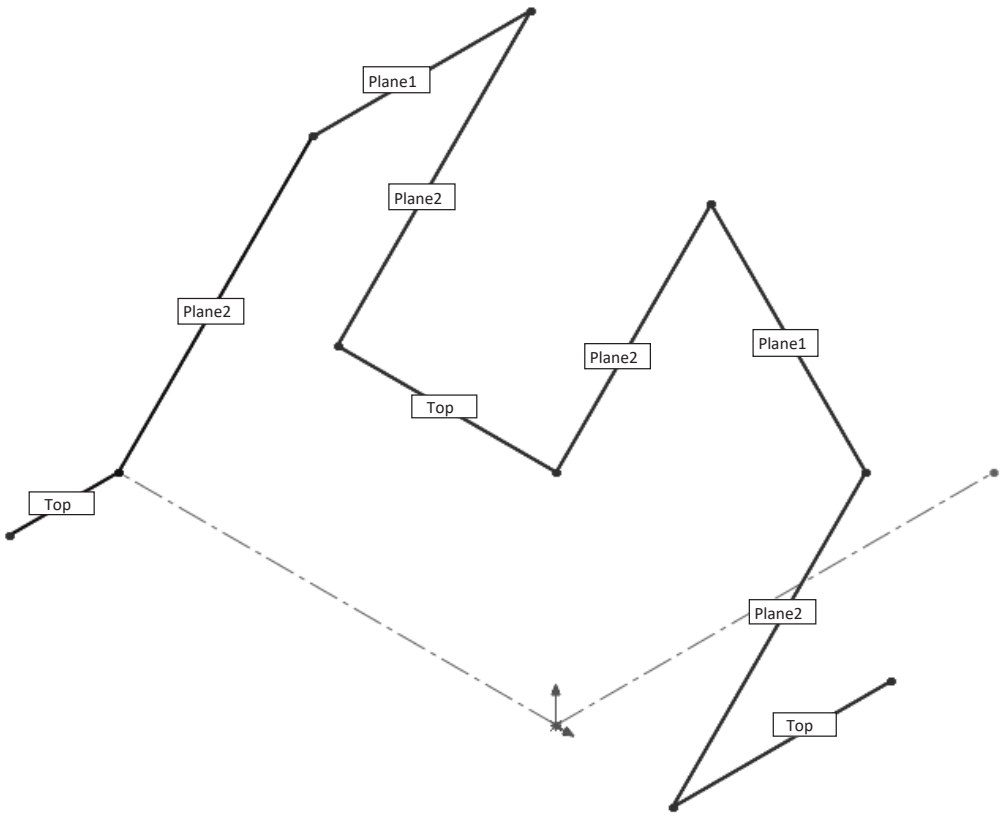
Switching Planes

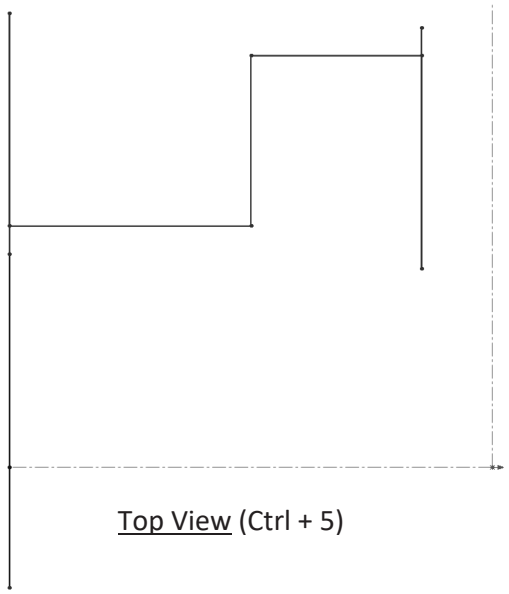
While sketching the lines, hold the **Control** key and click a plane to switch from one plane to another, or simply select them from the FeatureManager tree each time.



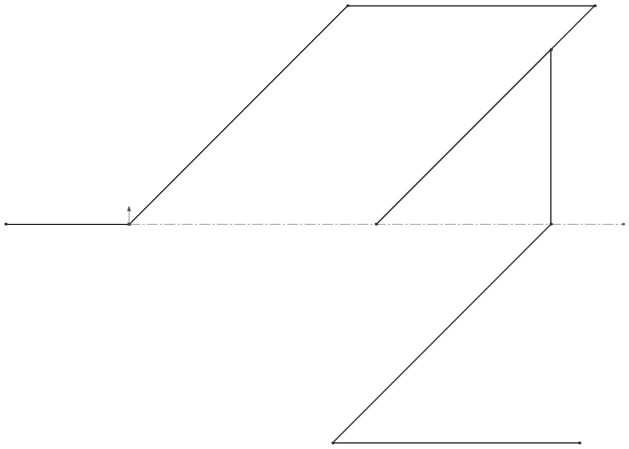
Sketch the rest of lines on the planes as labeled below.

For clarity, hide all the planes (select **View / Hide-Show** and click off **Planes**). We will select the planes from the FeatureManager tree when needed.



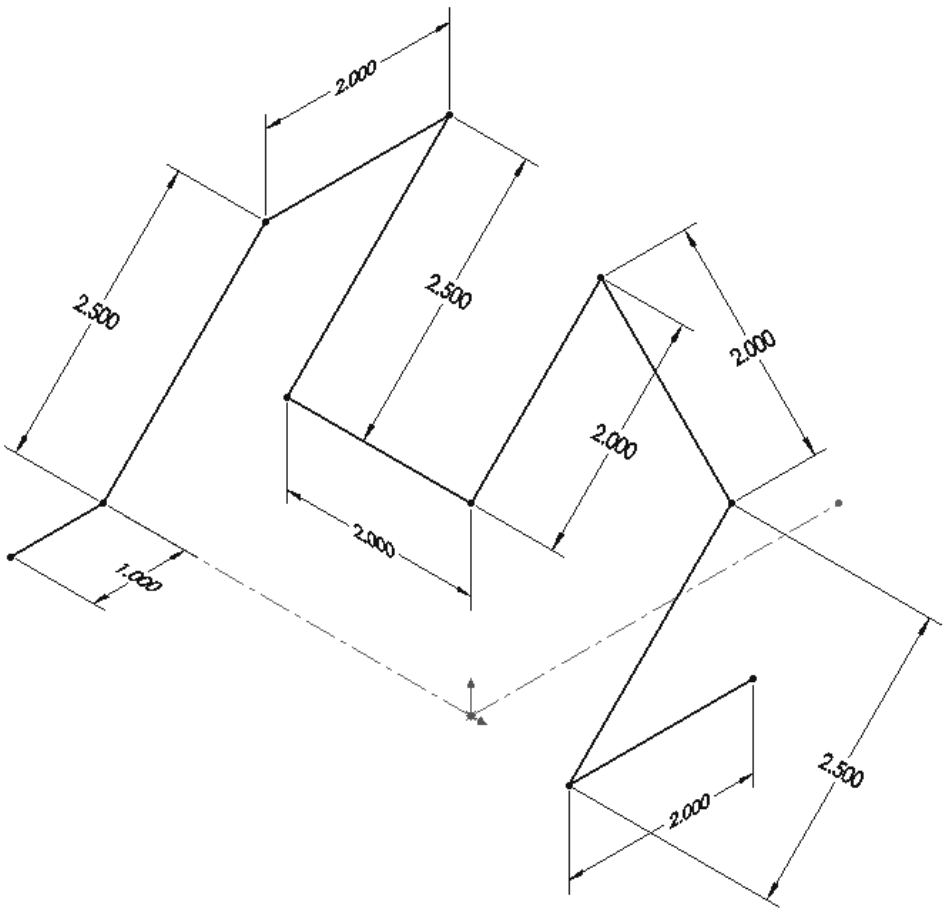


Top View (Ctrl + 5)

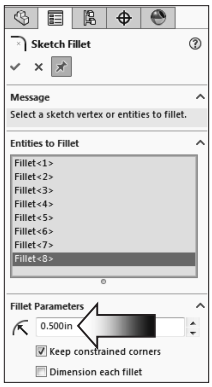


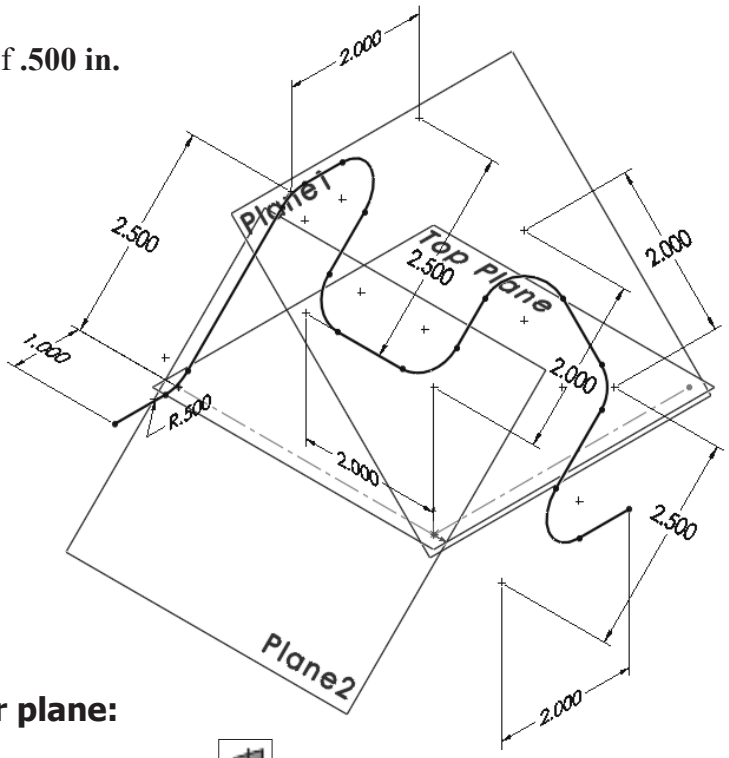
Right View (Ctrl + 4)

Add the dimensions below to fully define the sketch.



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





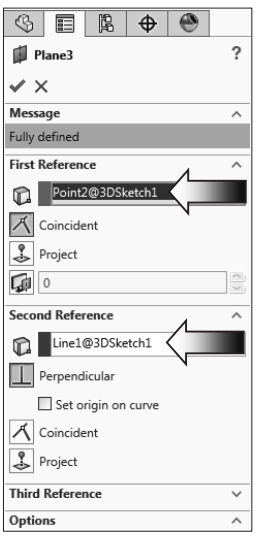
Exit the 3D Sketch or press **Ctrl+Q**.

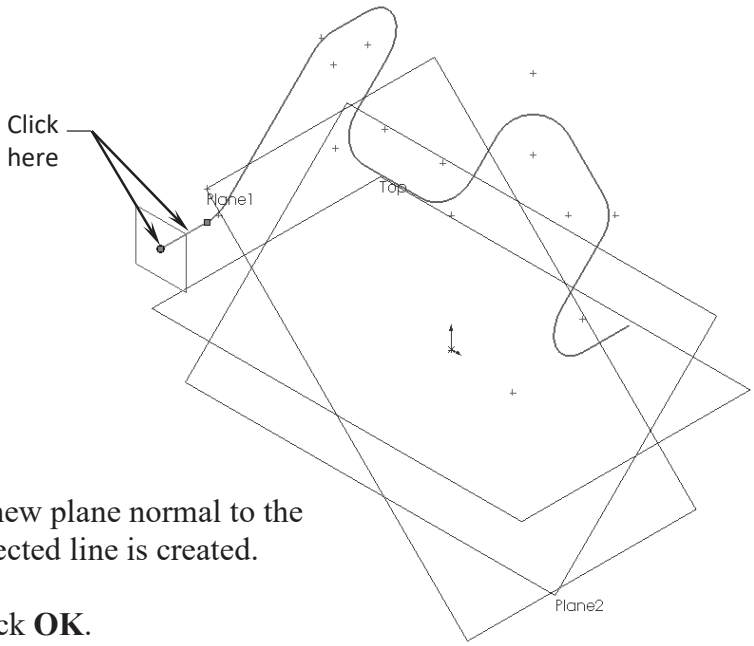
5. Creating a Perpendicular plane:

Select **Insert/Reference Geometry/Plane** .

Select the **line** and its **endpoint** approximately as shown.

The **Perpendicular** option should be selected by default.





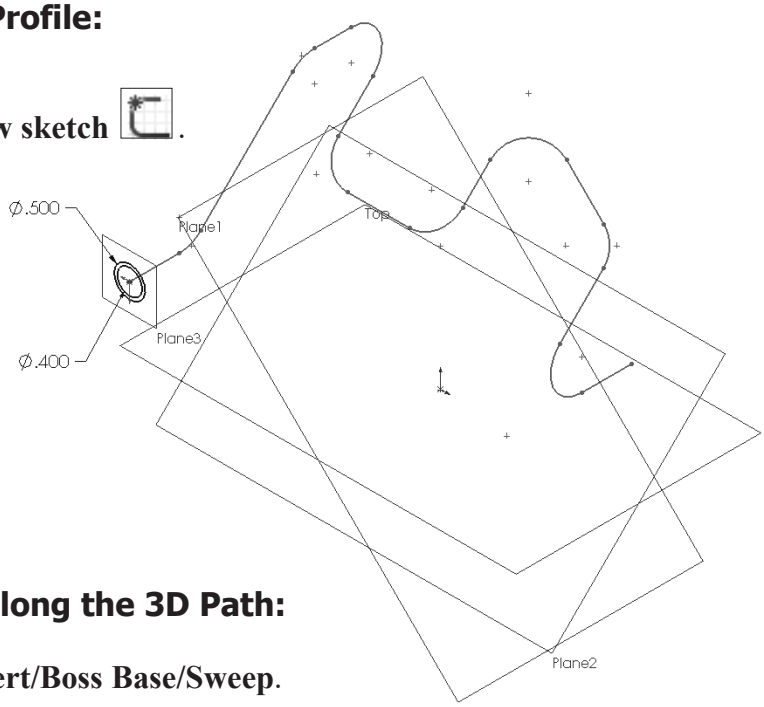
A new plane normal to the selected line is created.

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.

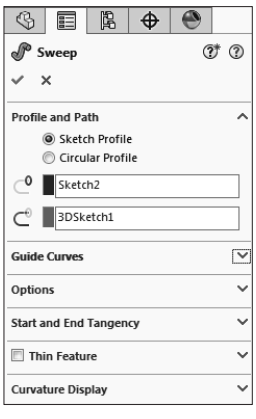


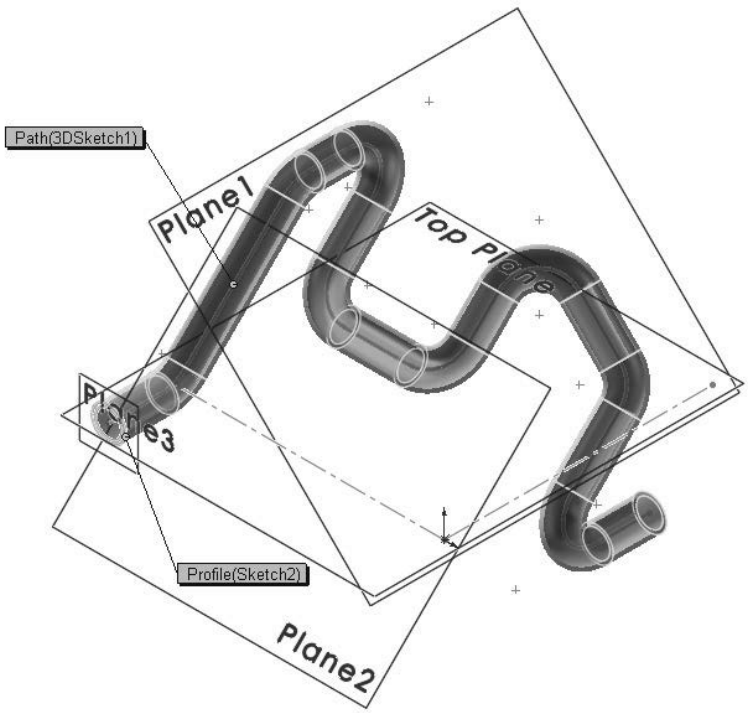
7. Sweeping the Profile along the 3D Path:

Click or Select **Insert/Boss Base/Sweep**.

Select the **Circles** as the Sweep Profile .

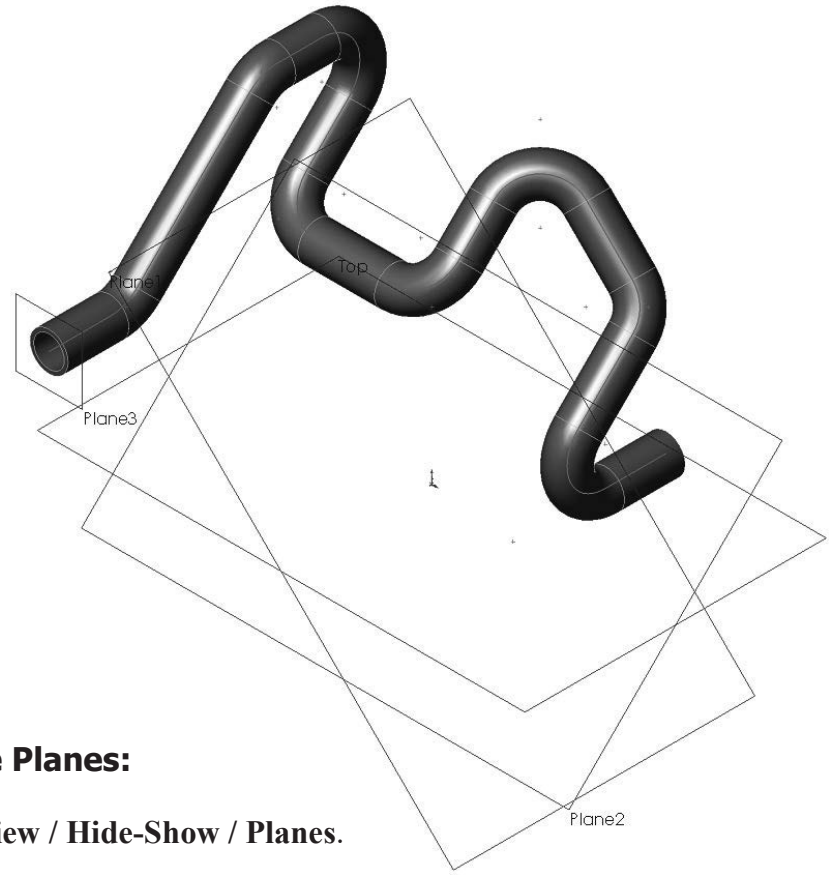
Select the **3D Sketch** as the Sweep Path .





Click **OK**.

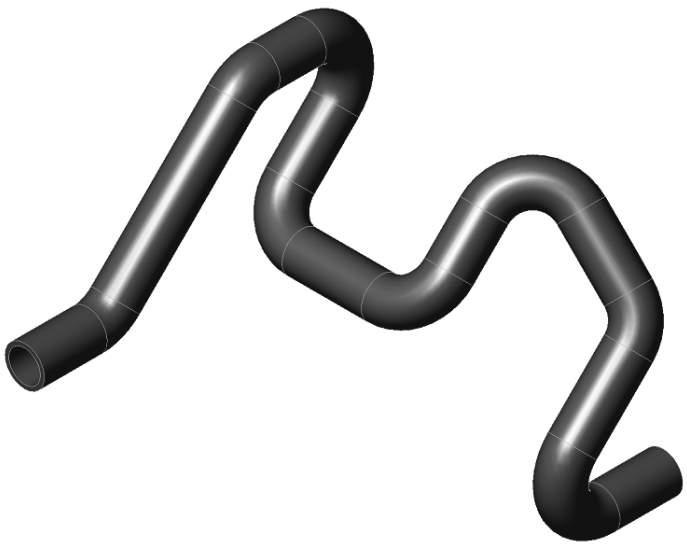
The resulting Swept feature.



8. Hiding the Planes:

Select **View / Hide-Show / Planes**.

The planes are temporarily put away from the scene.



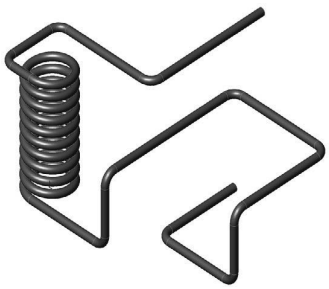
9. Saving your work:

Select **File / Save As**.

Enter **3D Sketch_Planes** for the name of the file.

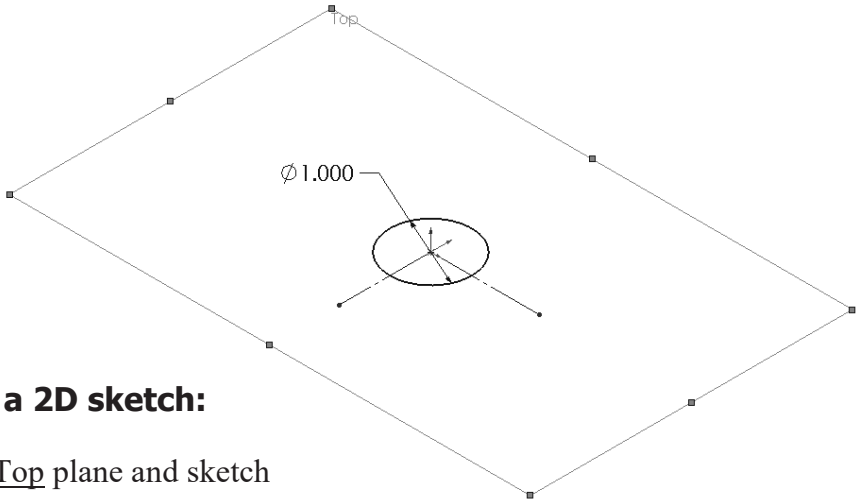
Click **Save**.

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 **1.00in** diameter **Circle**

and **2 Centerlines** .

2. Creating a Helix:

Select **Insert/Curve/**

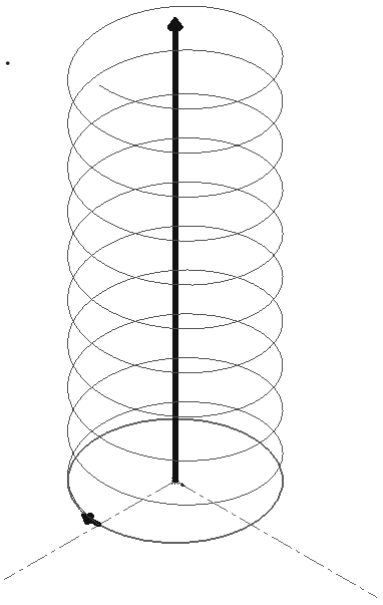
Helix-Spiral .

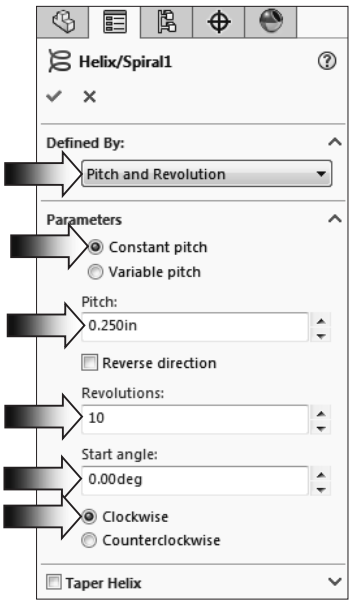
Pitch: **.250 in**.

Revolution: **10**.

Start Angle: **0 deg**.

Click **OK**.

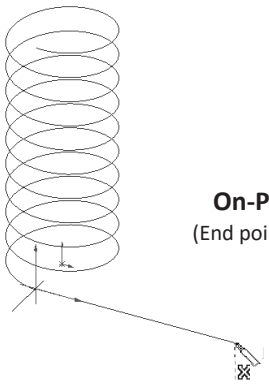




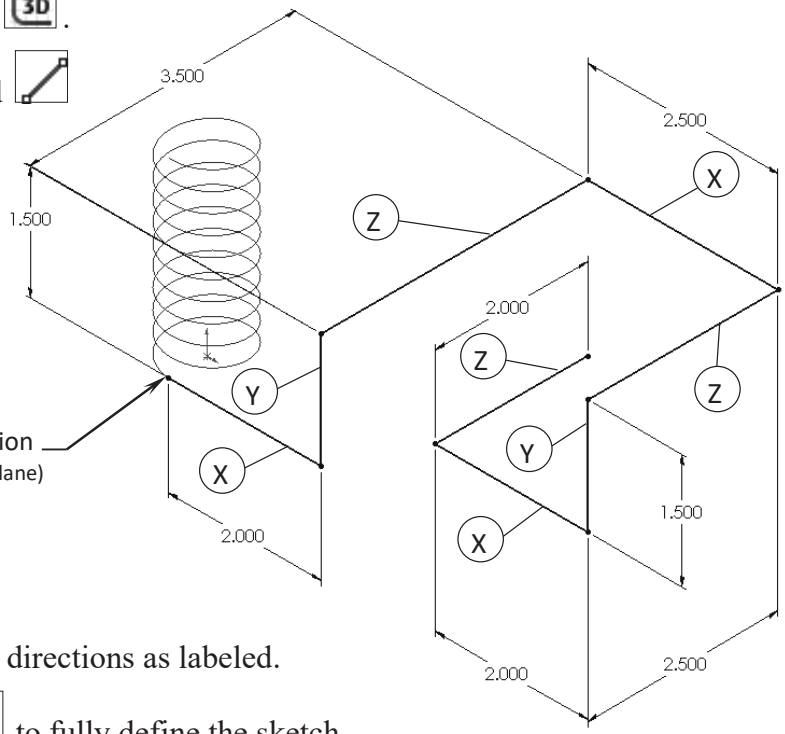
3. Creating the 1st 3D sketch:

Select **Insert/3D Sketch** .

Select the **Line** command and sketch the 1st line along the **X** direction.

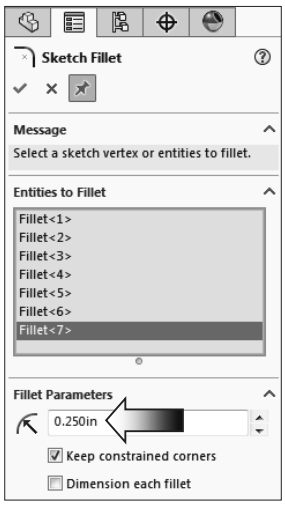


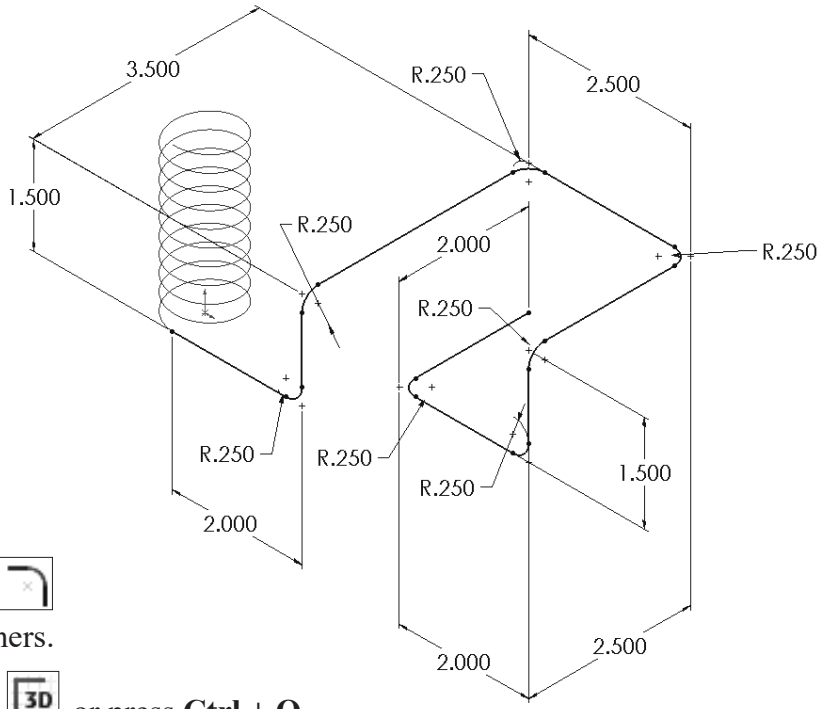
On-Plane relation
(End point & Right plane)



Add other lines in the directions as labeled.

Add Dimensions to fully define the sketch.





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

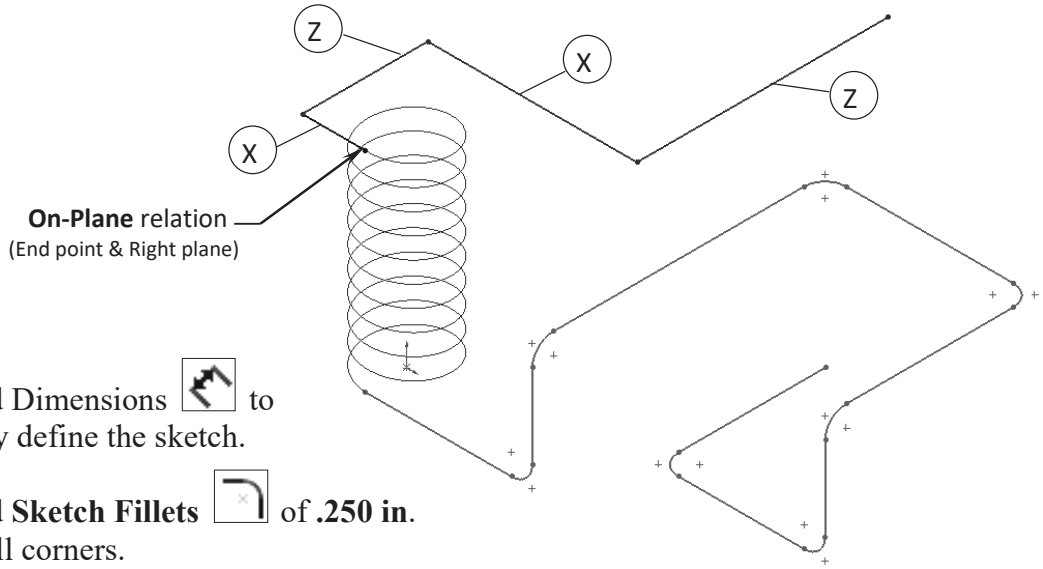
Exit the 3D Sketch or press **Ctrl + Q**.

4. Creating the 2nd 3D sketch:

Select **Insert/3D Sketch** .

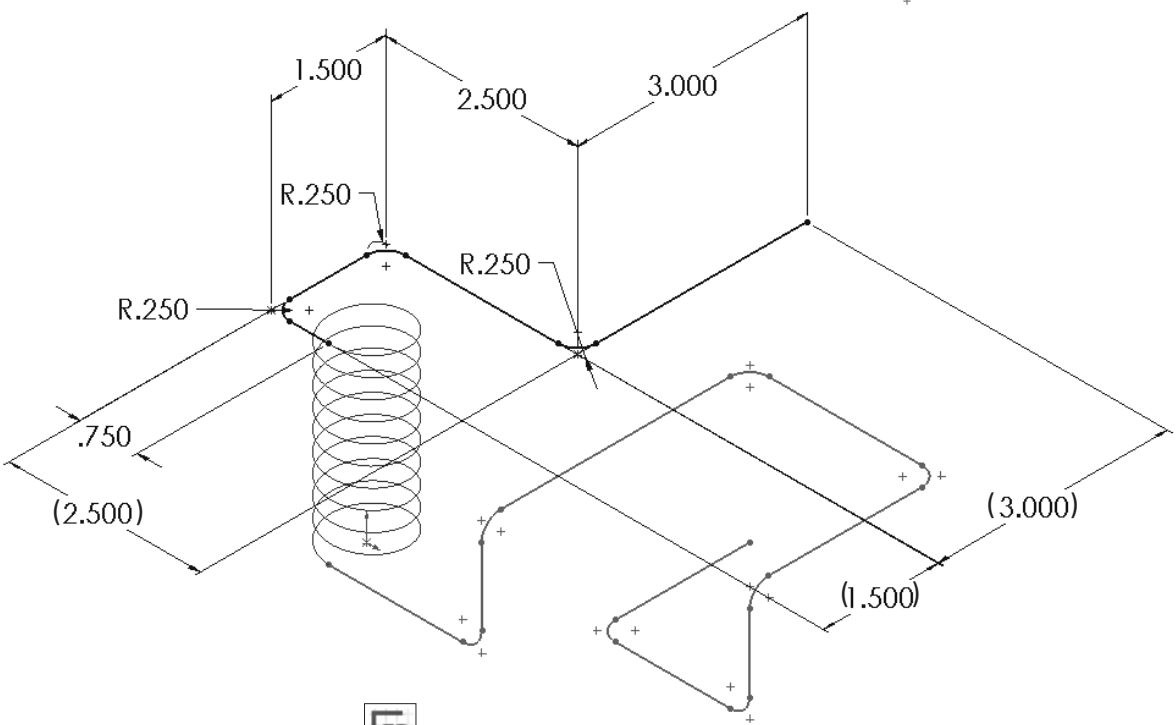
Select the **Line** command and sketch the 1st line along the X direction.

Sketch the rest of the lines following their direction shown below.



Add Dimensions to fully define the sketch.

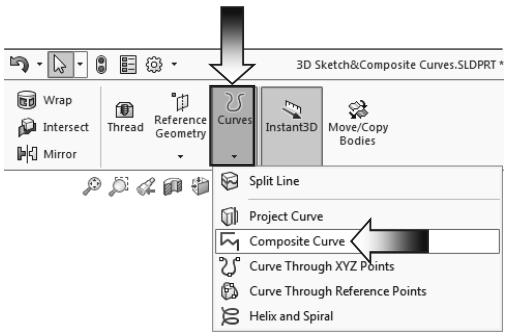
Add **Sketch Fillets** of **.250 in**. to all corners.

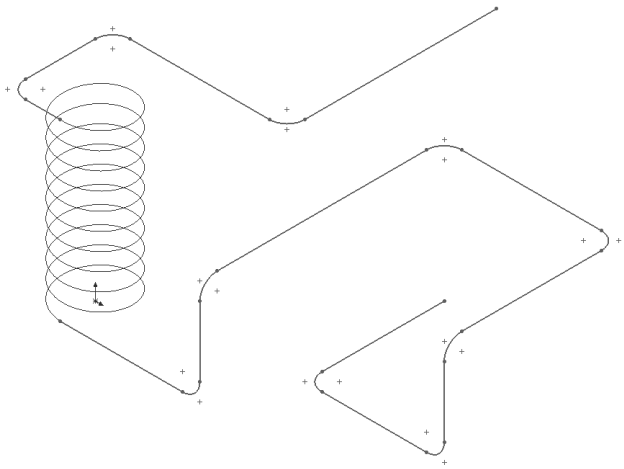


Exit the 3D Sketch or press **Ctrl+Q**.

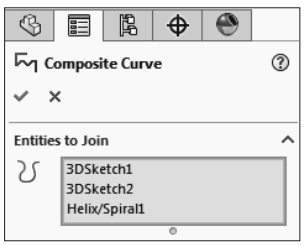
5. Combining the curves:

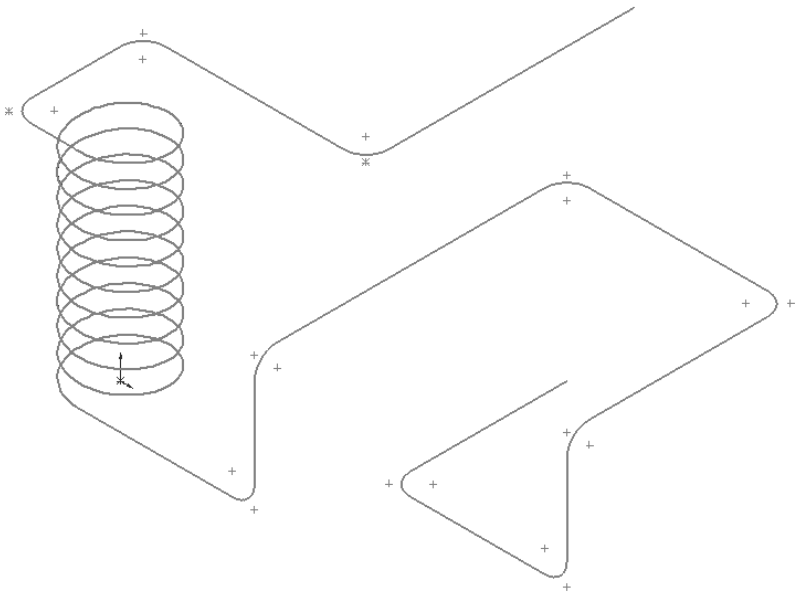
Select the **Composite Curve** command below the Curves button, or select: **Insert / Curve / Composite**.





Select the 3 Sketches either from the FeatureManager tree or directly from the graphics area.





Click **OK**.

The sketches are now
combined to 1 continuous curve.
We will use it as the sweep path in the next few steps.

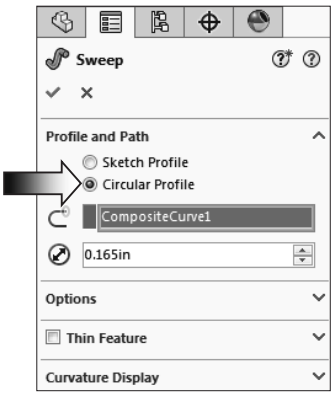
6. Creating a Sweep using Circular Profile:

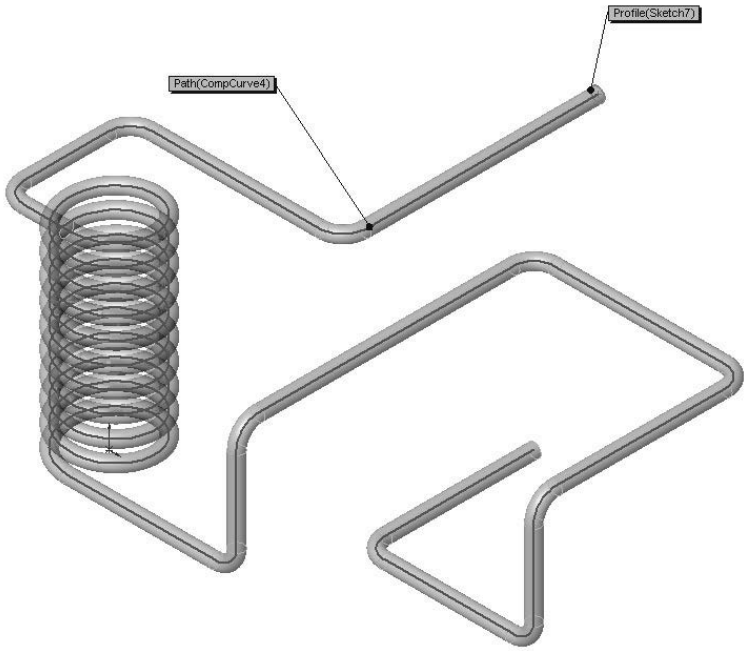
Select **Insert/Boss Base/ Sweep** .

Select the **Circle Profile** option (arrow).

For sweep profile, enter **.165 in** .

For Sweep Path, select the **Composite Curve** .

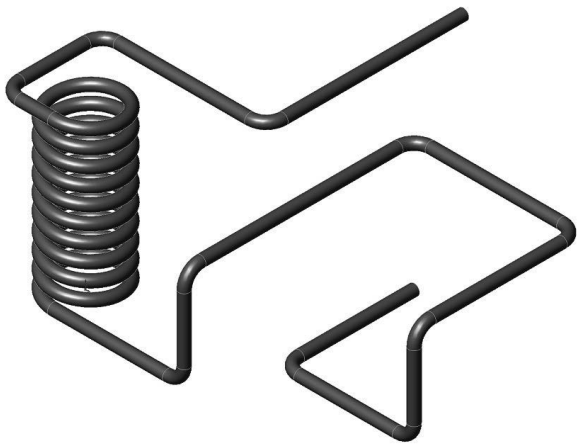
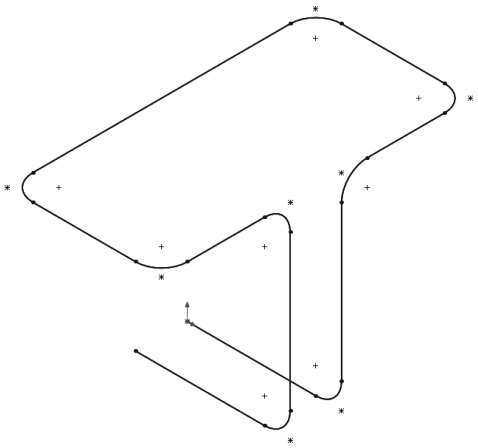
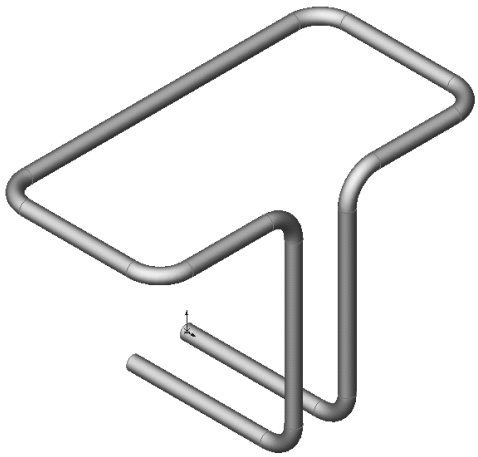
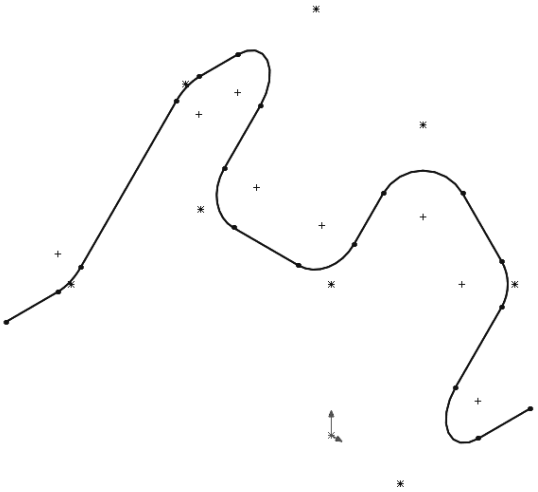
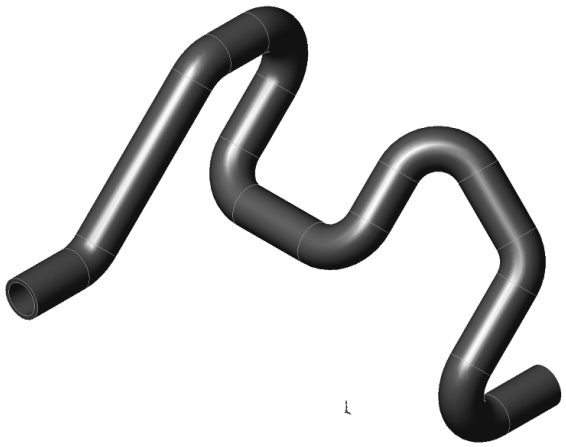
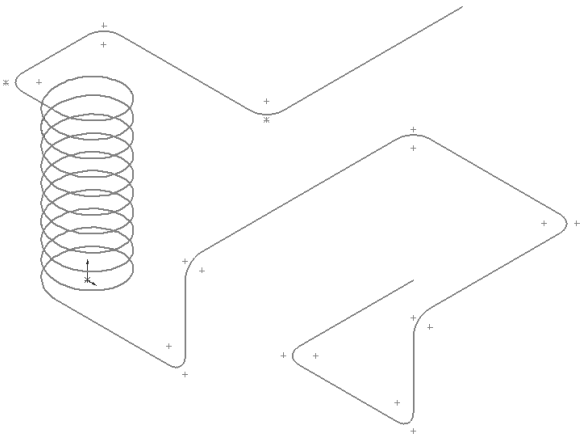
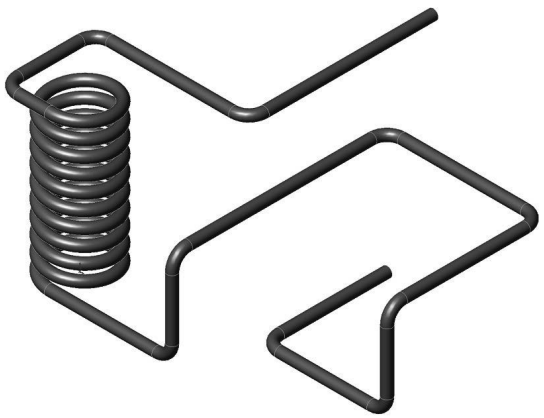




Click **OK**.

7. Saving your work:

Click **File/Save As**.

Enter **3D Sketch_ Composite Curve** for the name of the file.

Click **Save**.