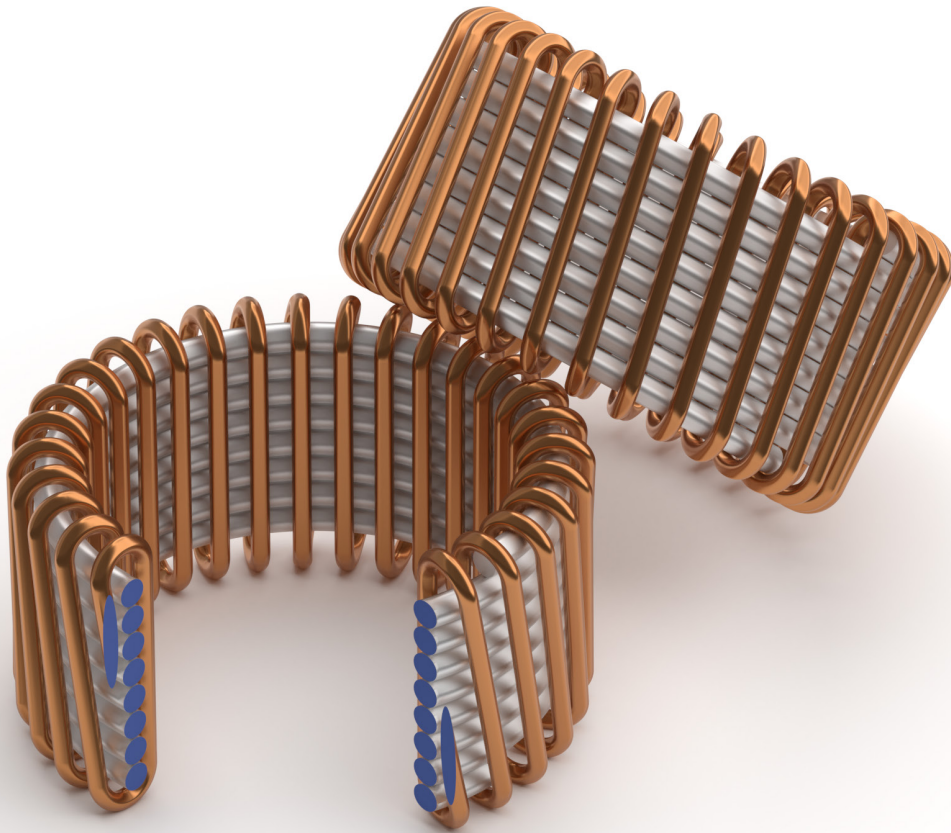


SOLIDWORKS® 2019 Advanced Techniques

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



Paul Tran CSWE, CSWI

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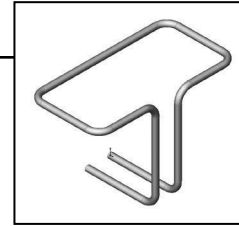
[amazon.com](https://www.amazon.com)

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

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

CHAPTER 1








Introduction To 3D Sketch

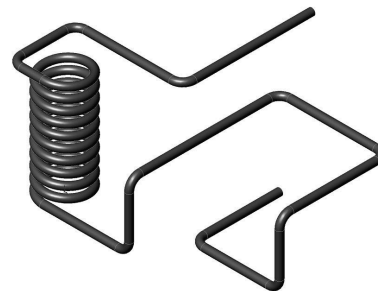
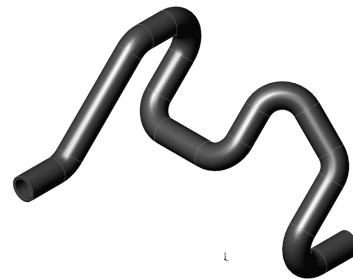


Introduction to 3D Sketch

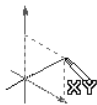
SOLIDWORKS allows you to create 3D sketches. A 3D sketch consists of lines and arcs in series and splines. You can use a 3D sketch as a sweep path, as a guide curve for a loft or sweep, a centerline for a loft, or as one of the key entities in a piping system. Geometric relations can also be added to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

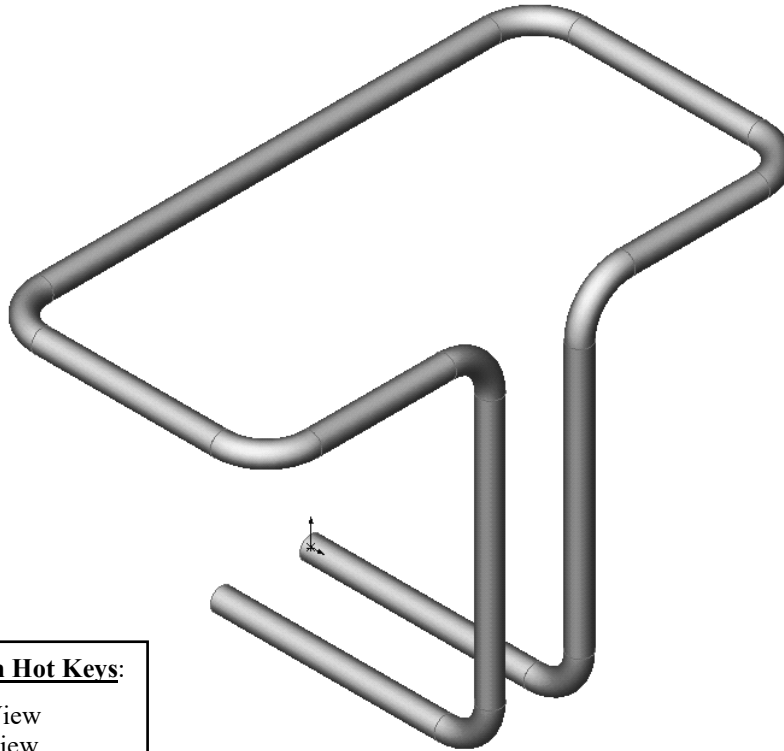


Space Handle



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



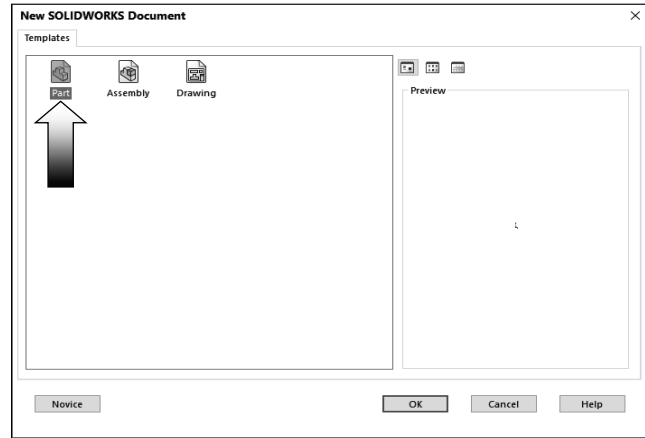
Tab Key






Base/ Boss Sweep

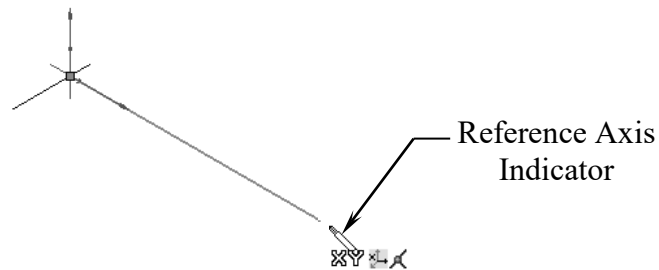
1. Starting a new part file:

- Click **File / New**.
- Select the **Part** template and click **OK**.

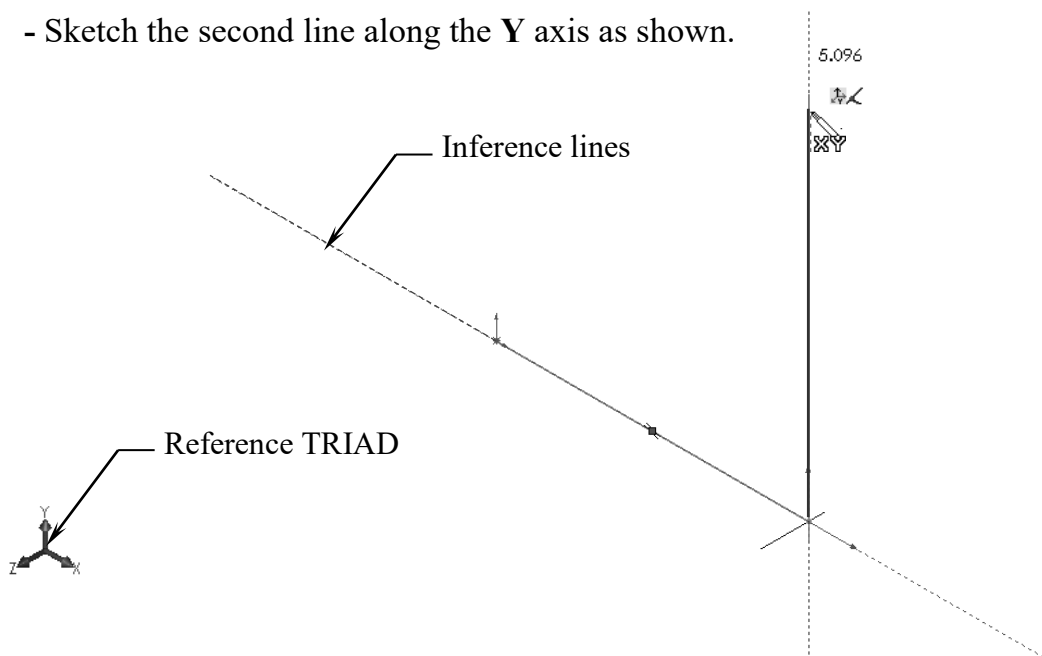


2. Creating a 3D Sketch:

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

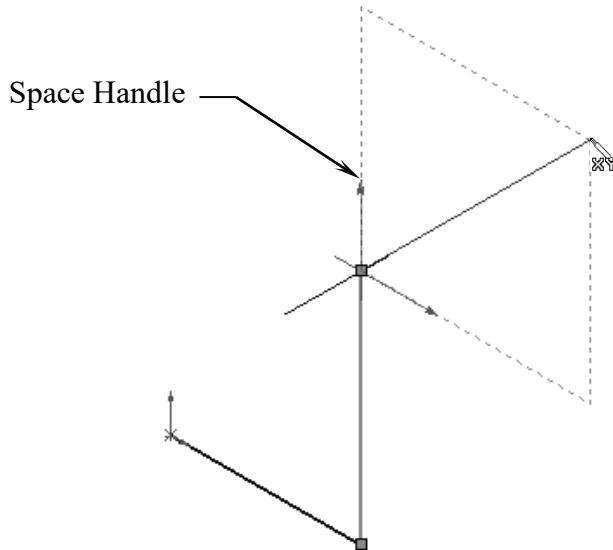


- Sketch the second line along the **Y** axis as shown.



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.

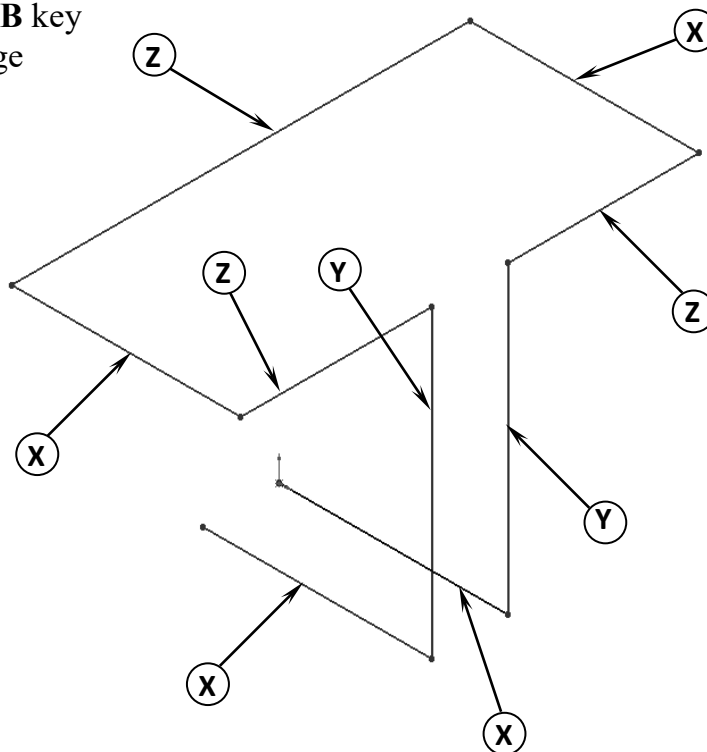


The TAB key


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

4. Completing the profile:

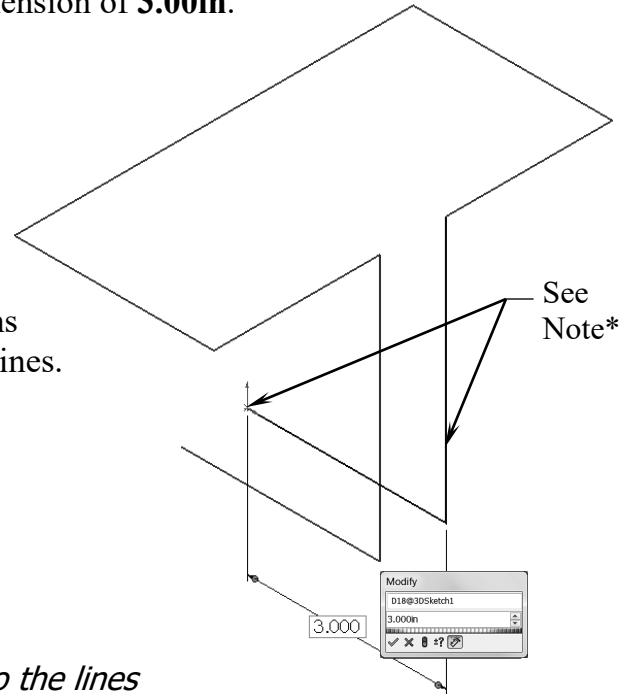
- Sketch the other lines and follow the axis as labeled; press **TAB** key if needed to change the direction.



5. Adding dimensions:

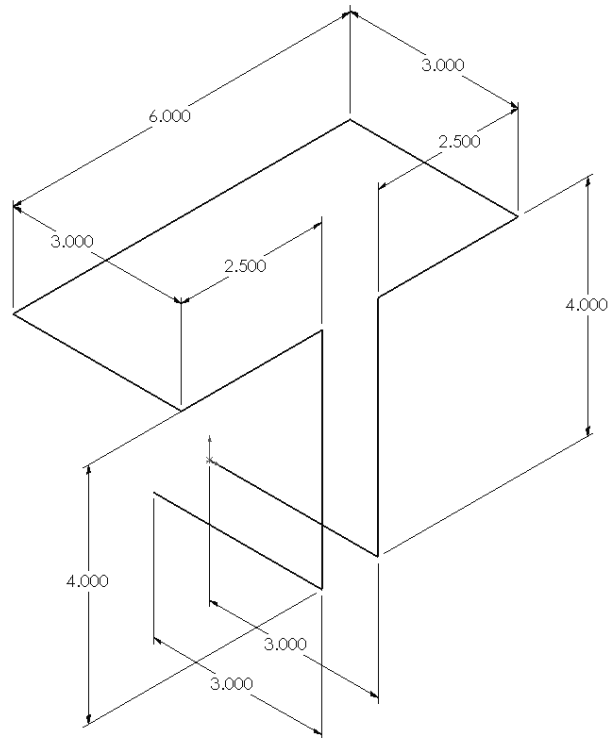
- Click  or select **Tools / Dimensions / Smart Dimension**.
- Click on the first line and add a dimension of **3.00in**.

- 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.

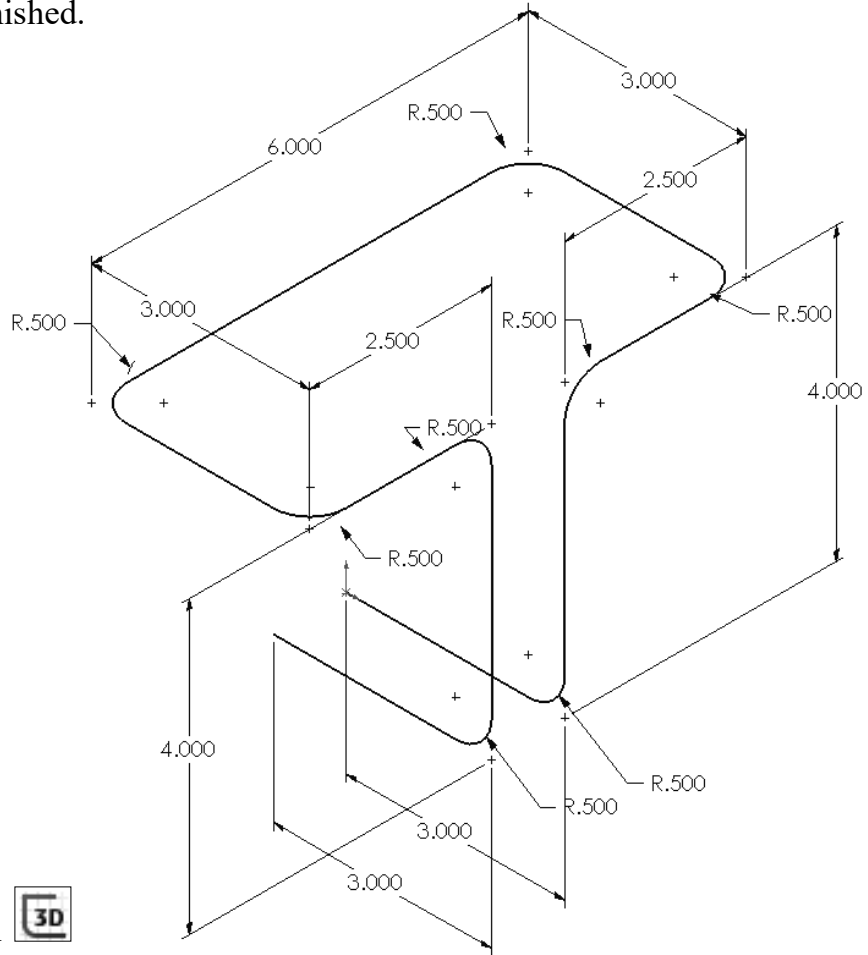
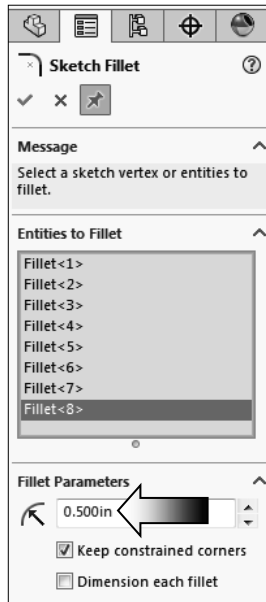
- Continue adding the dimensions to fully define the 3D sketch as shown.




- 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.

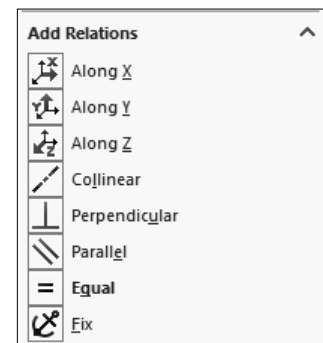


- **Exit** the 3D Sketch  or press **Control + Q**.






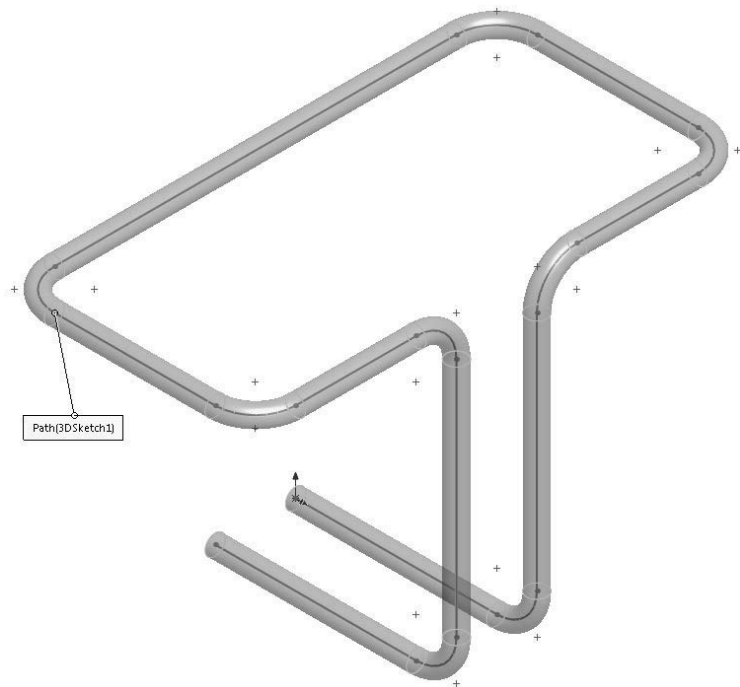
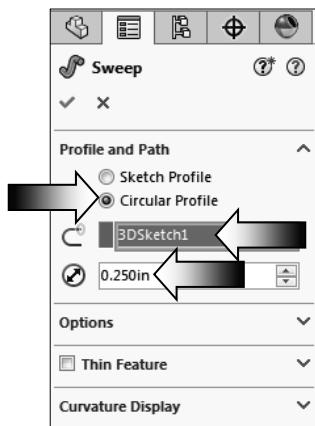
Geometric Relations

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



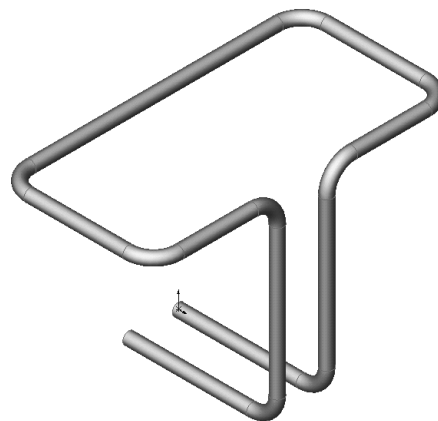
7. Creating the Swept feature:

- The new Circular Profile sweep 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

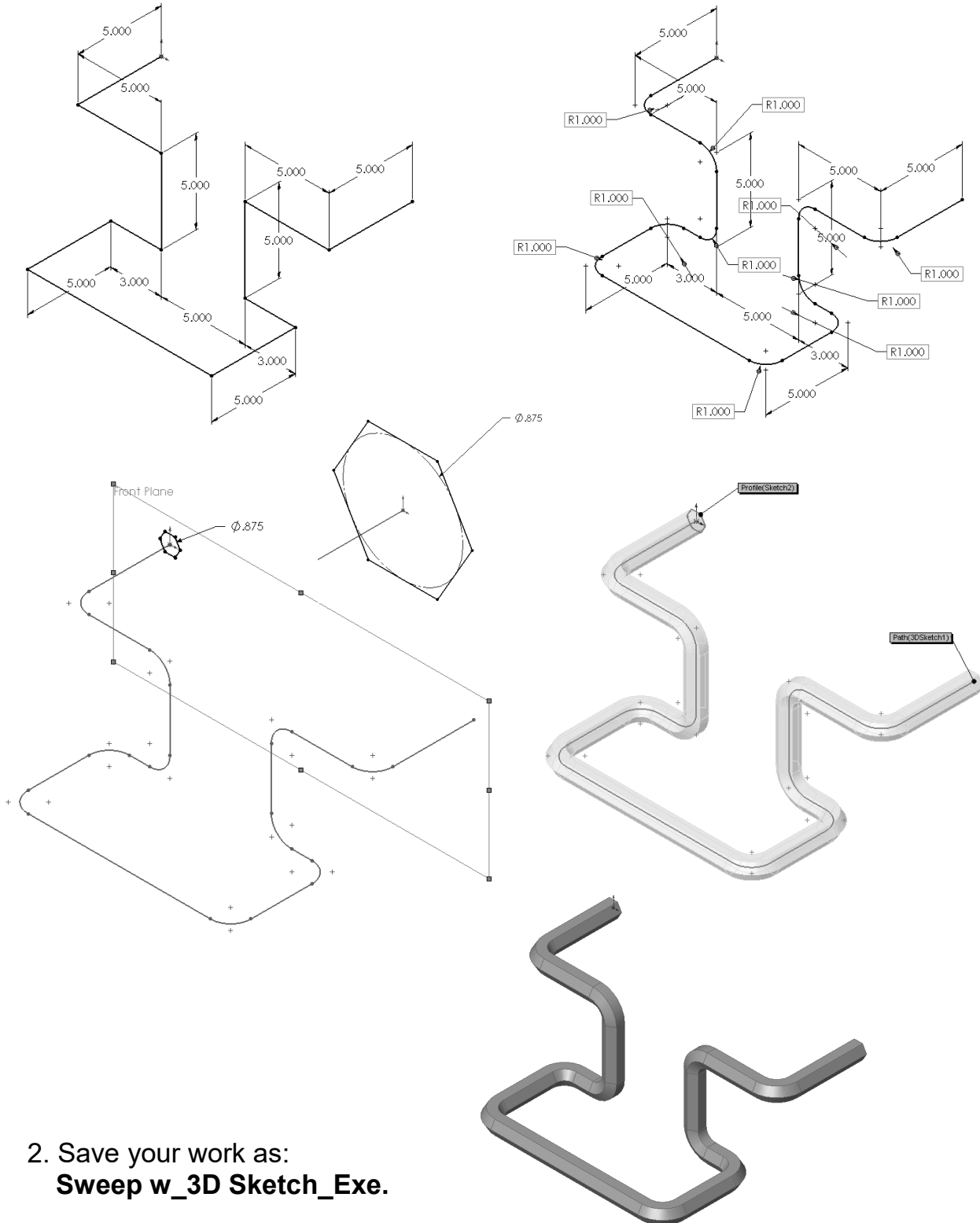
Introduction to 3D Sketch

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

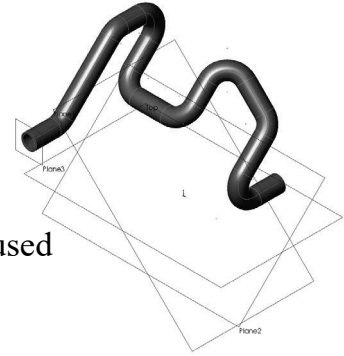
1. Create the part shown using 3D Sketch.



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.

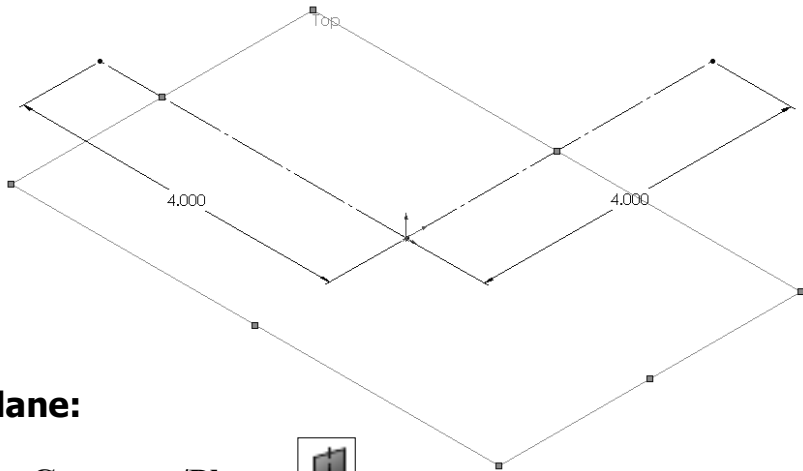
1. Sketching the reference Pivot lines:

- Select the Top plane and

open a new sketch .

- Sketch **2 Centerlines** .

and add Dimensions  as shown.

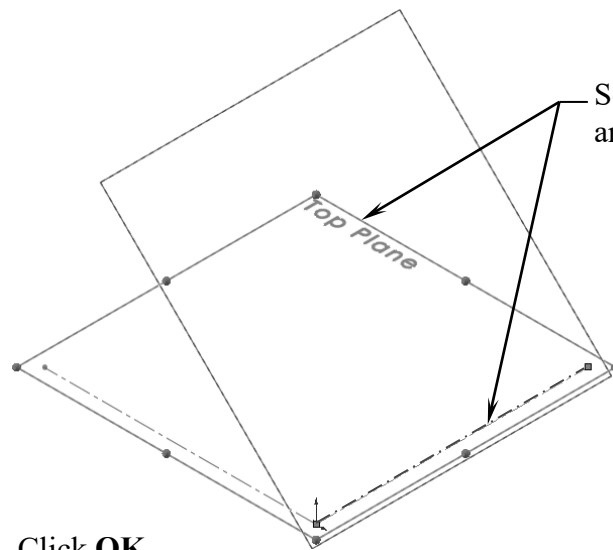
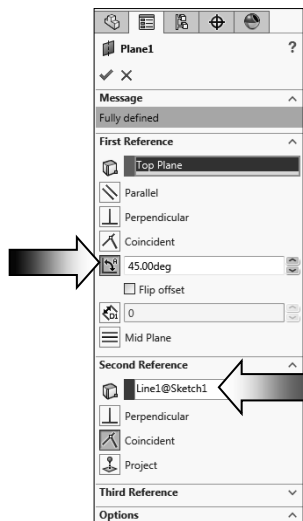


2. Creating the 1st 45° Plane:

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

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


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

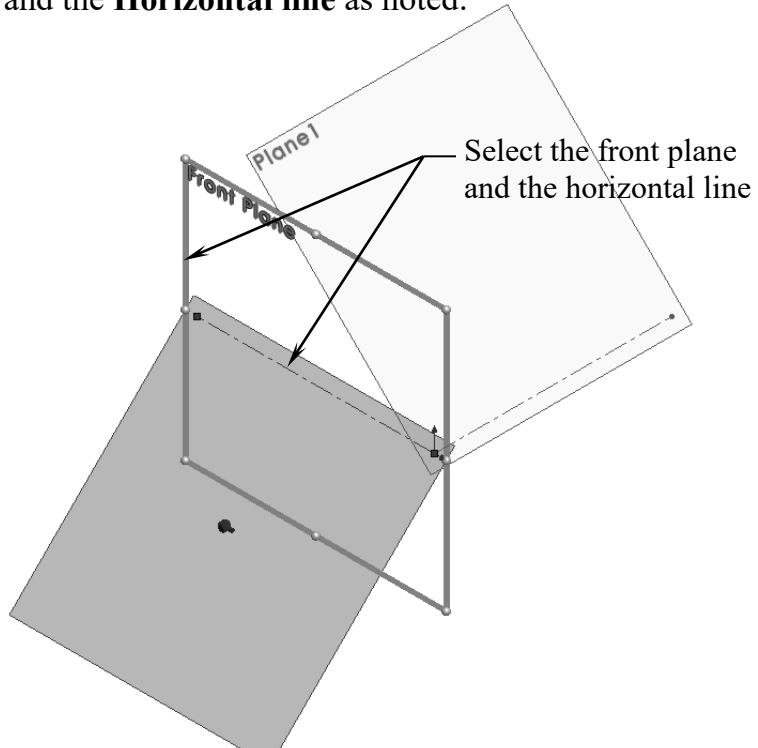
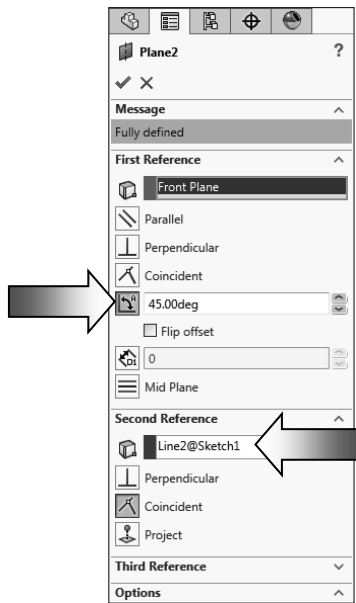


Select the top plane and the vertical line...

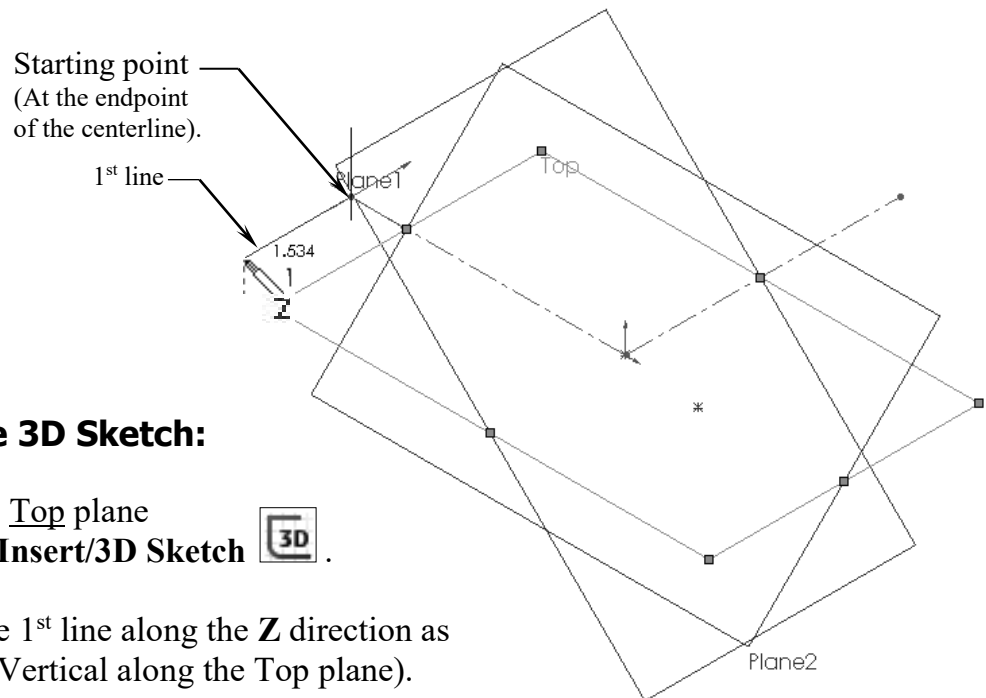
- Click **OK**.

3. Creating the 2nd 45° Plane:


- 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**.



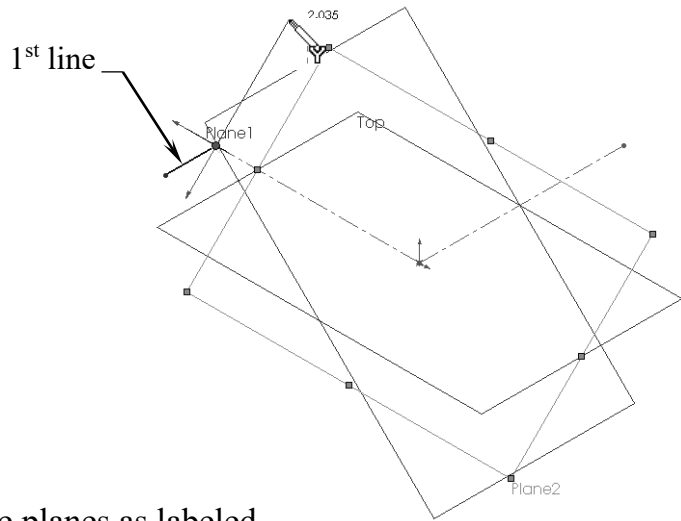
4. Creating the 3D Sketch:

- 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).

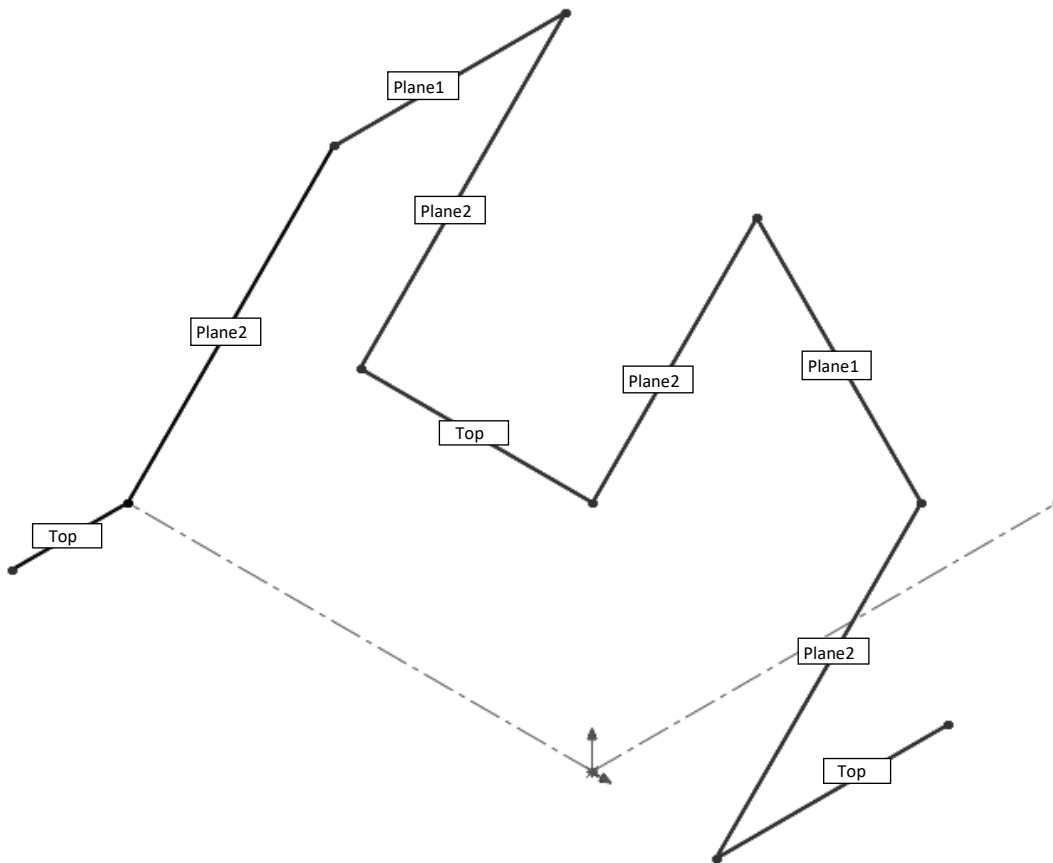
- Select the **Plane2** (45 deg.) from the Feature Manager tree and Sketch the 2nd line along the **Y** direction (watch the cursor feedback symbol).

 **Switching Planes**

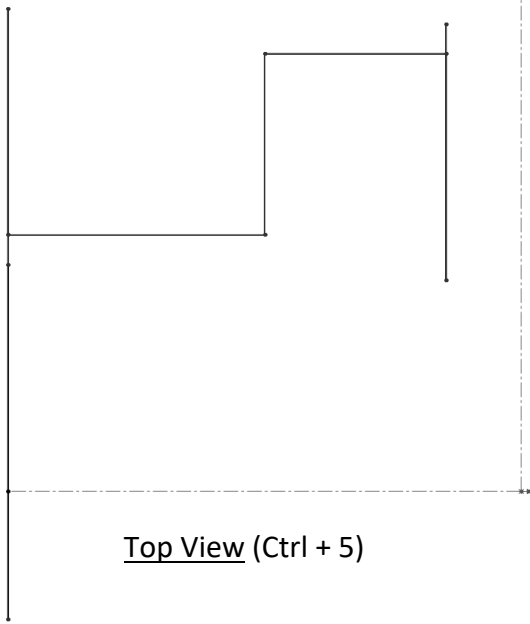
While sketching the lines, hold the **Control** key and click a plane to switch from one plane to another, or simply select them from the Feature tree each time.



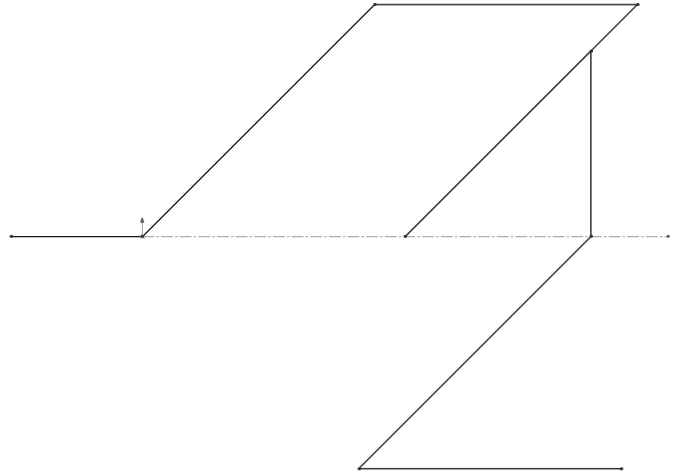
- Sketch the rest of the lines on the planes as labeled.
- For clarity, hide all the planes (select **View / Hide-Show** and click off **Planes**). We will select the planes from the FeatureManager tree when needed.




SOLIDWORKS 2019 | Advanced Techniques | Introduction to 3D Sketch

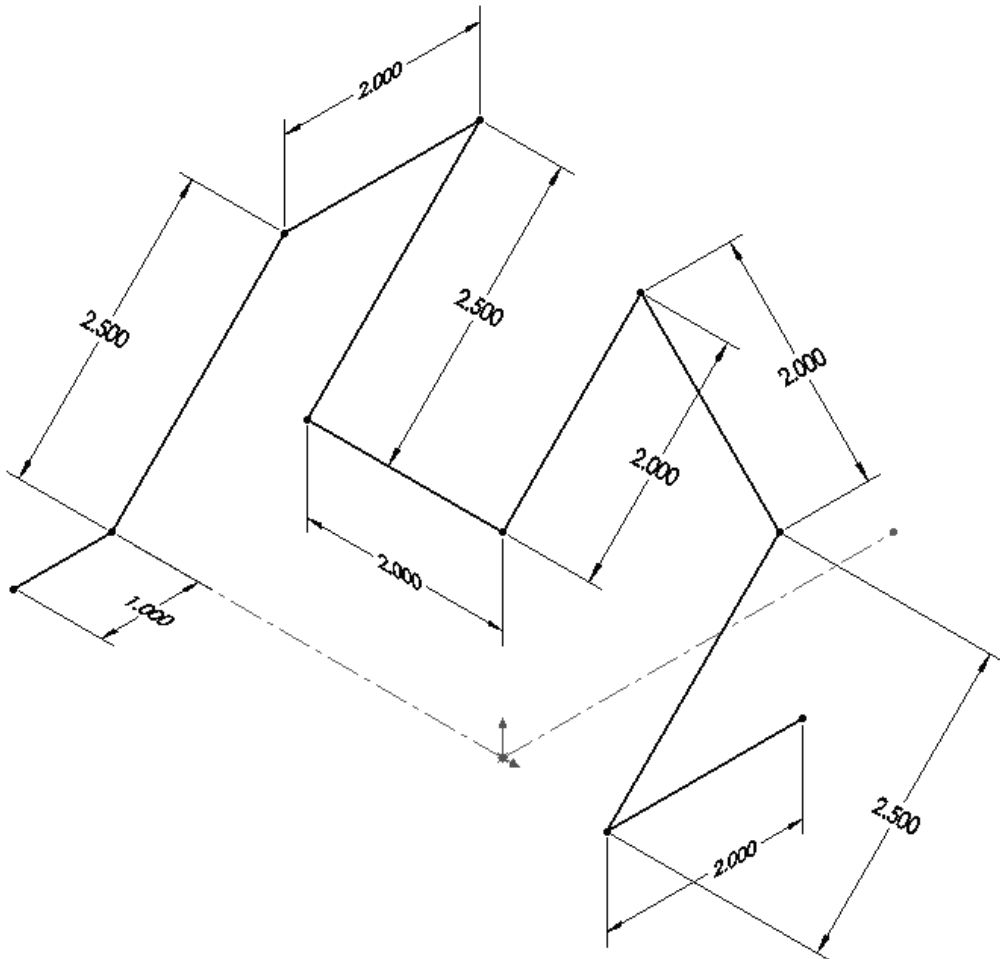


Top View (Ctrl + 5)

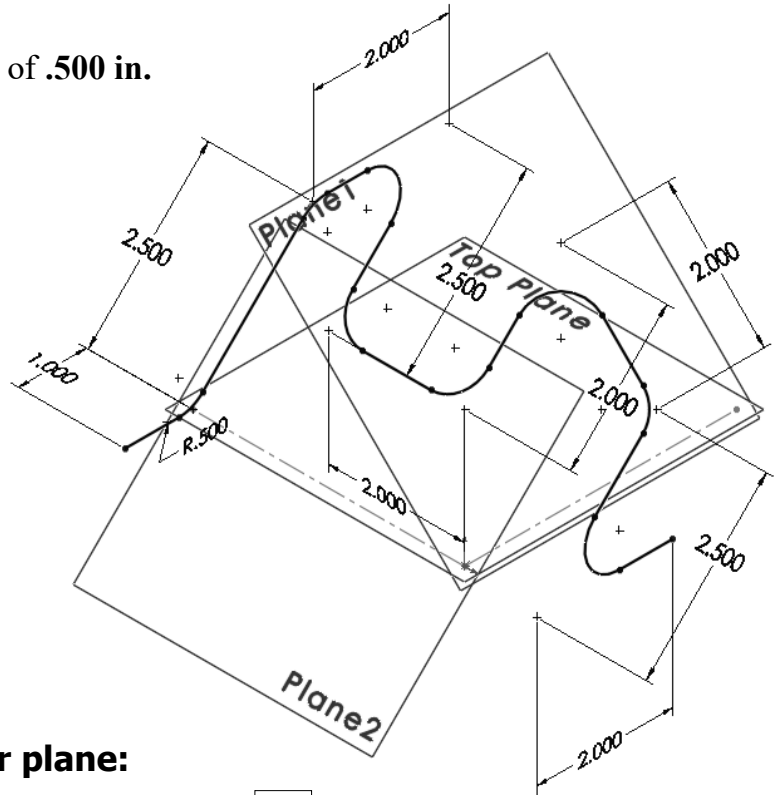
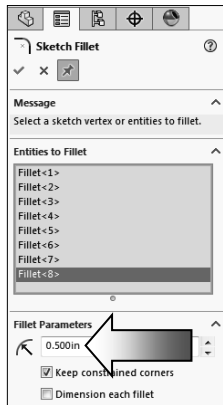


Right View (Ctrl + 4)

- Add Dimensions  to fully define the sketch.




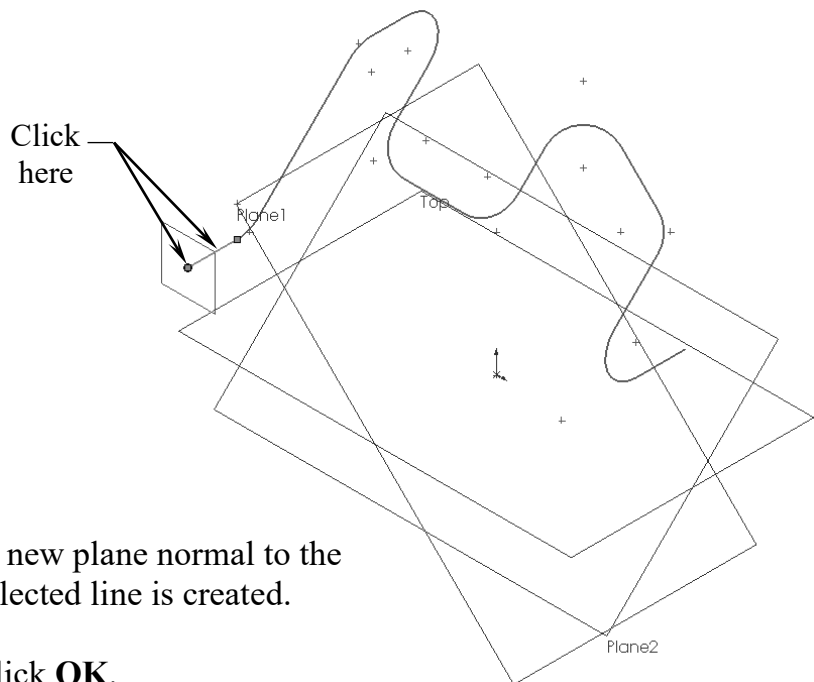
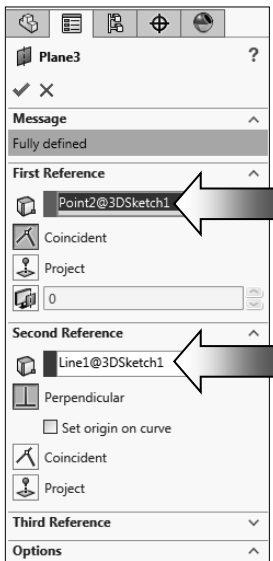
- Add Sketch Fillets  of .500 in. to all corners.



- **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.

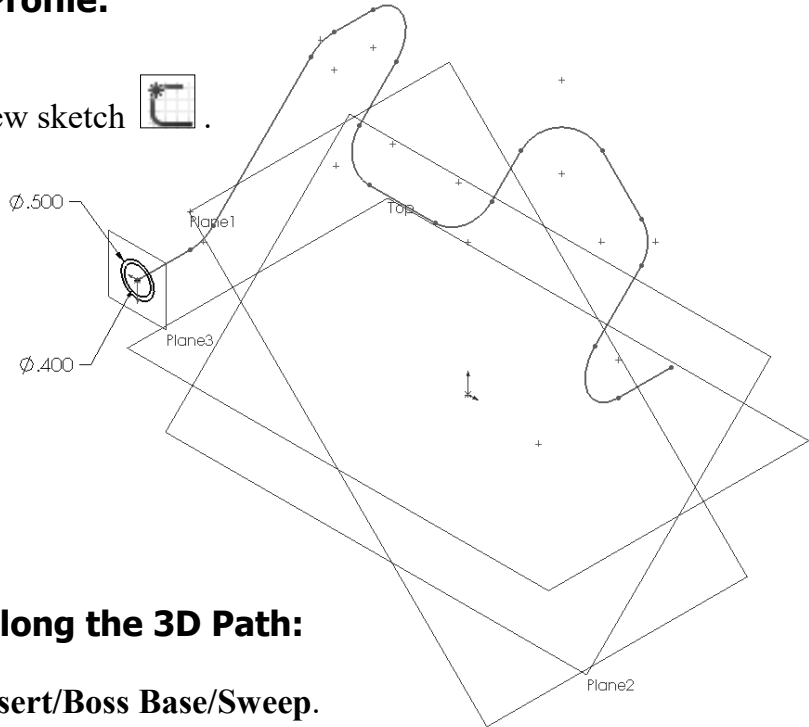


- A new plane normal to the selected line is created.
- Click **OK**.

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.

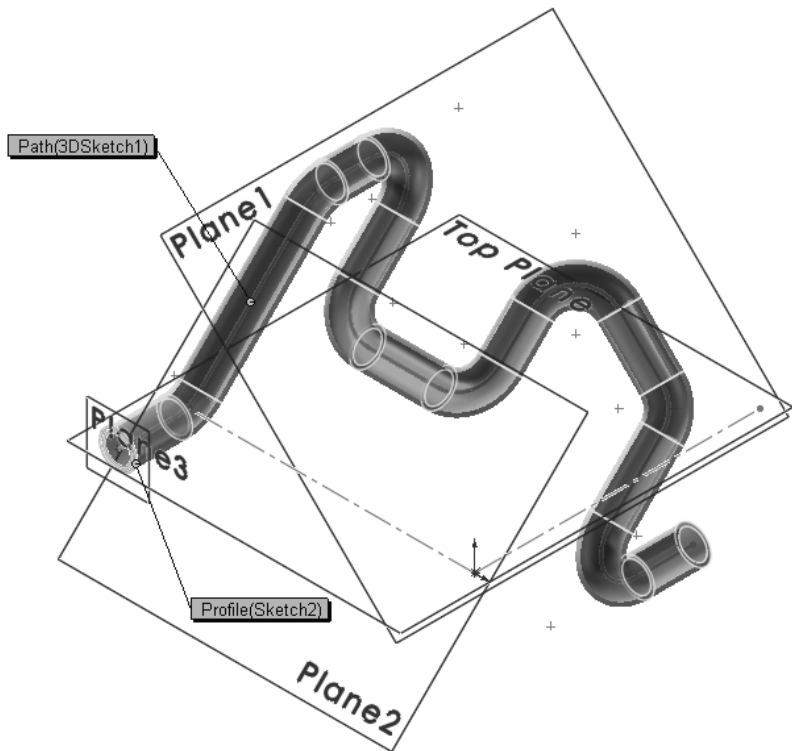
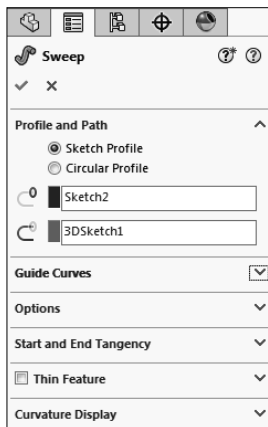


7. Sweeping the Profile along the 3D Path:

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

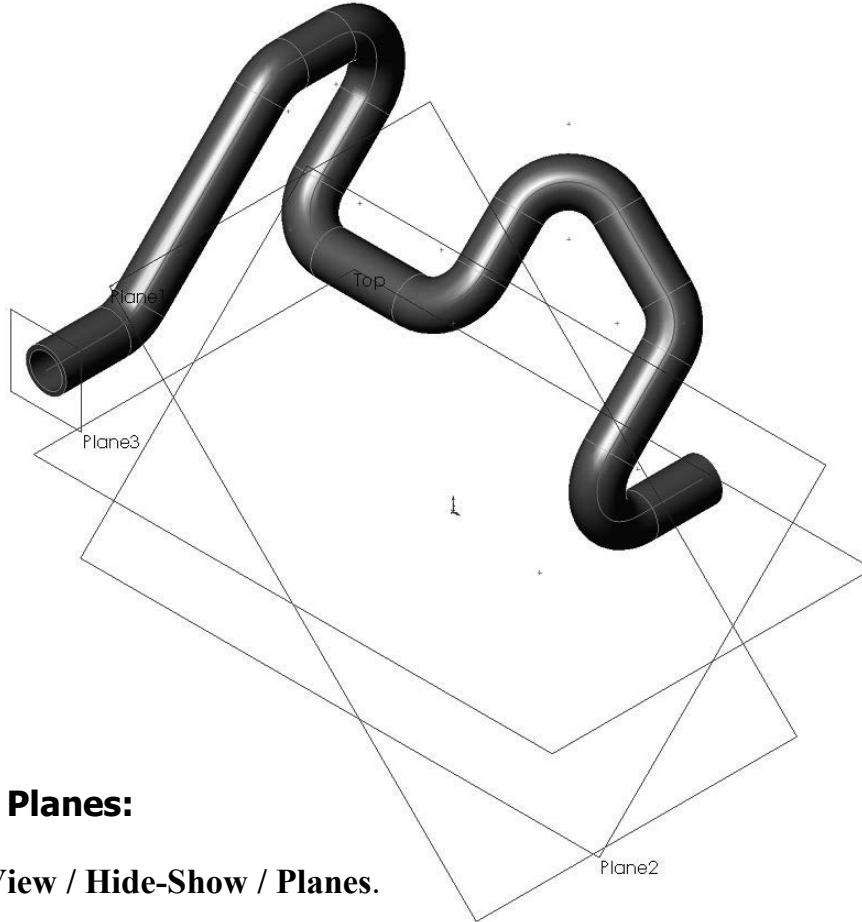
- Select the **Circles** as the Sweep Profile .

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



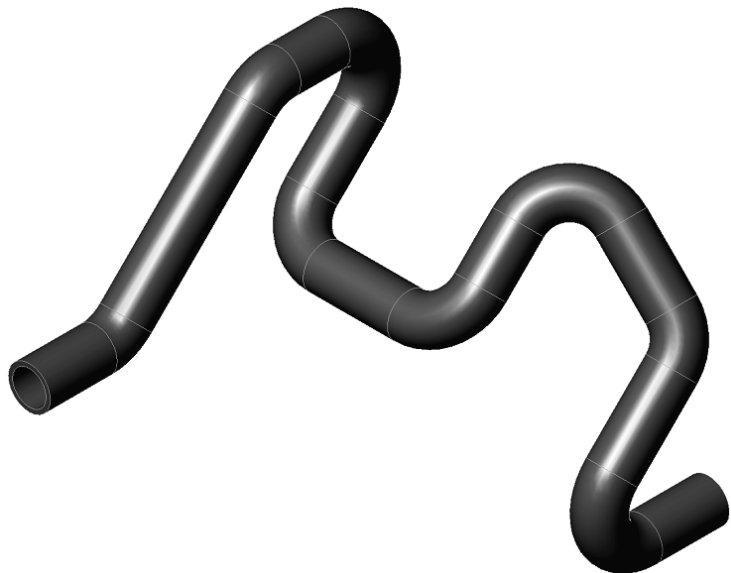
- Click **OK**.

- The resulting Swept feature.



8. Hiding the Planes:

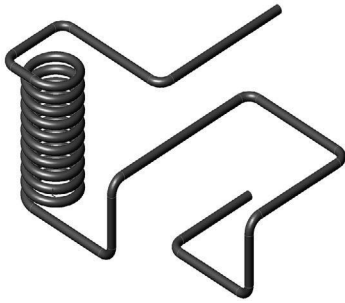
- Select **View / Hide-Show / Planes**.
- The planes are temporarily put away from the scene.



9. Saving your work:

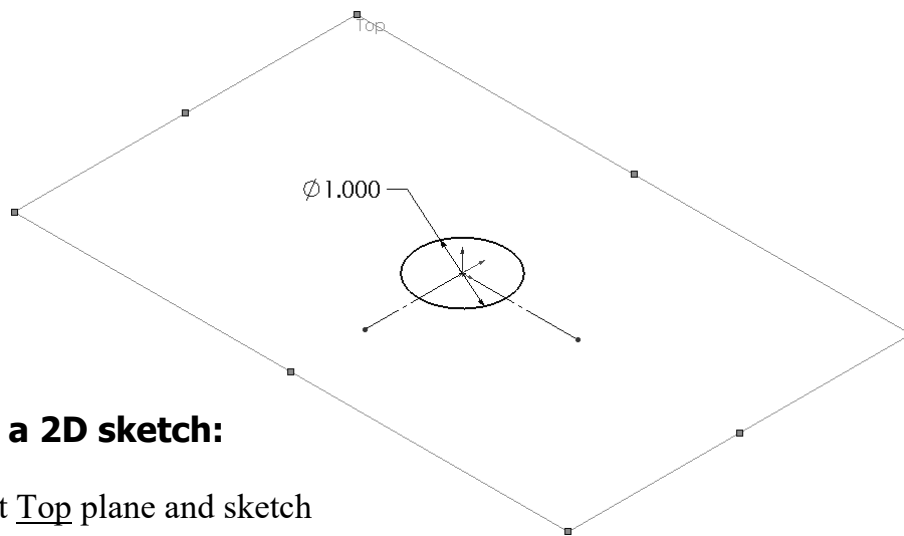
- Select **File / Save As**.
- Enter **3D Sketch_Planes** for the name of the file.
- Click **Save**.

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 **I**nsert/**C**urve/

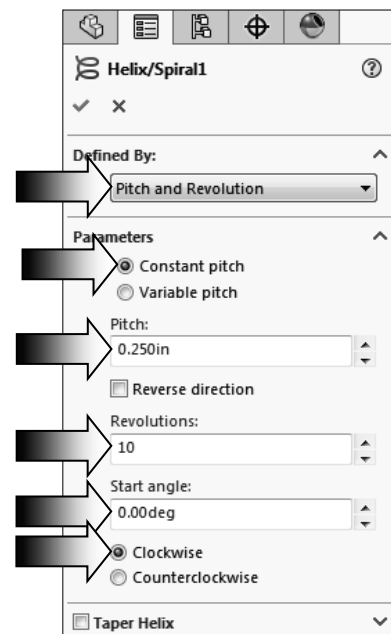
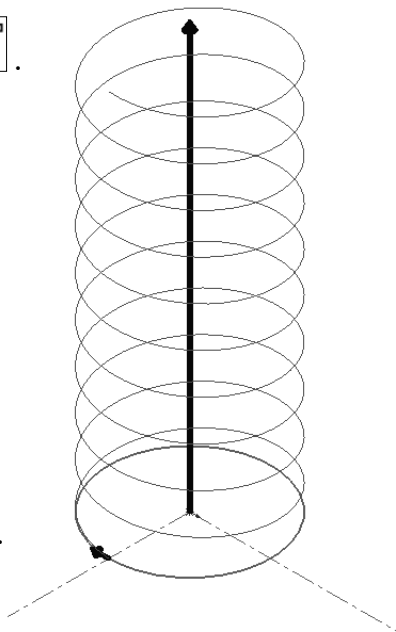
Helix-**S**piral .

- Pitch: .250 in.


- Revolution: 10.


- Starting Angle: 0 deg.

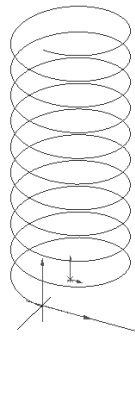
- Click **OK**.



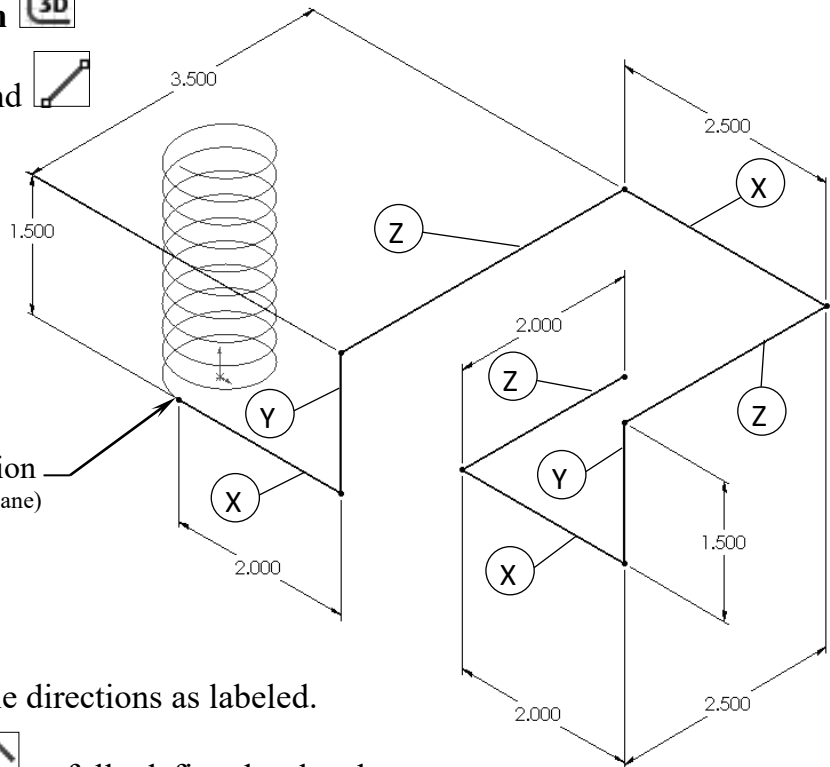
3. Creating the 1st 3D sketch:

- Select **Insert/3D Sketch** 


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

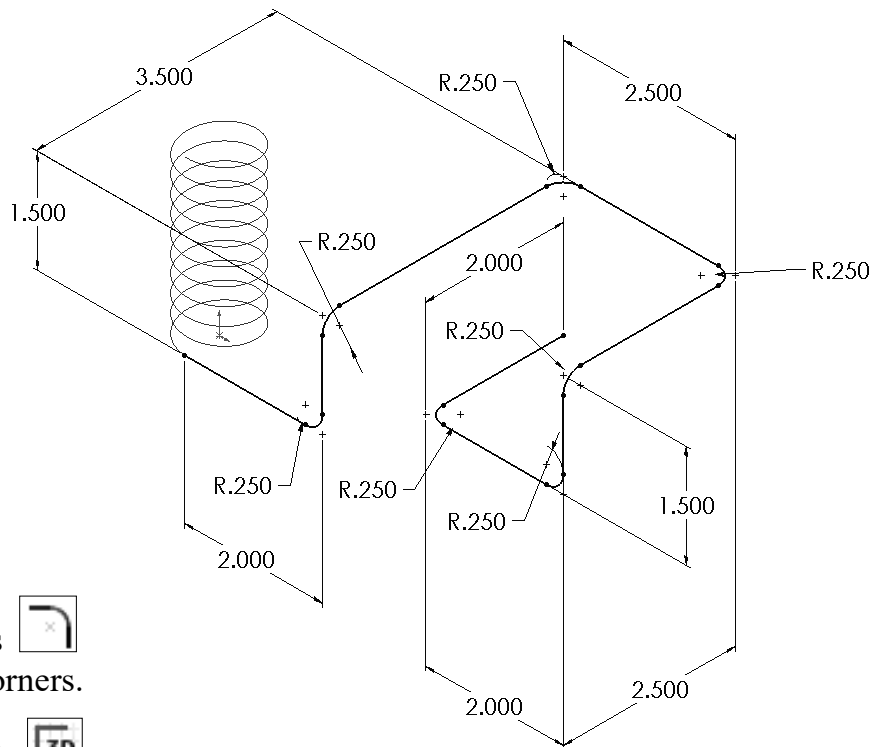



On-Plane relation
(End point & Right plane)




- Add other lines in the directions as labeled.



- Add Dimensions  to fully define the sketch.

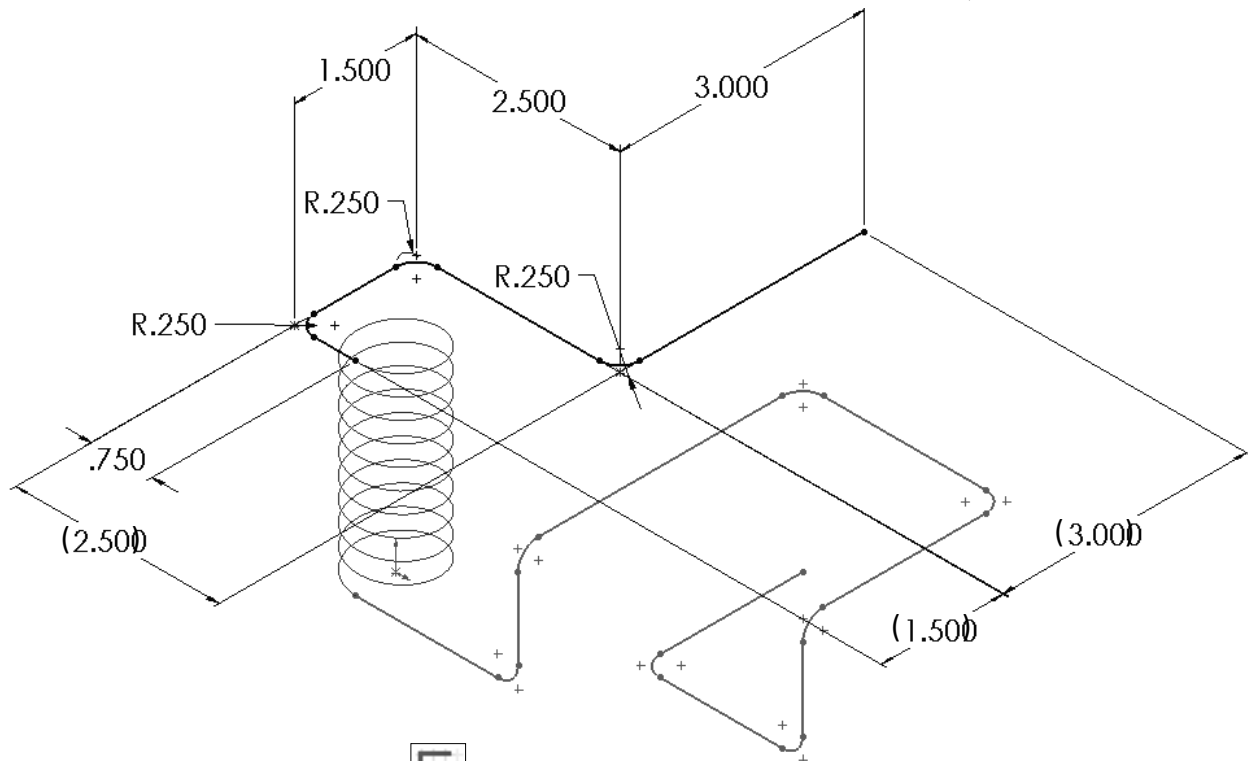
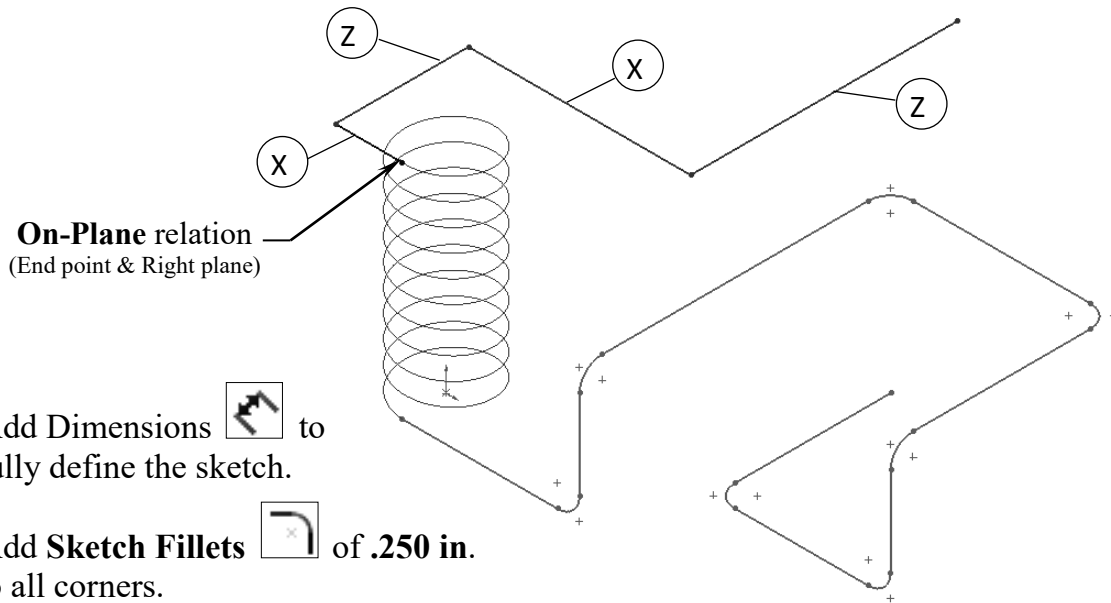



- Add Sketch Fillets  of **.250 in.** to all corners.

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


4. Creating the 2nd 3D sketch:

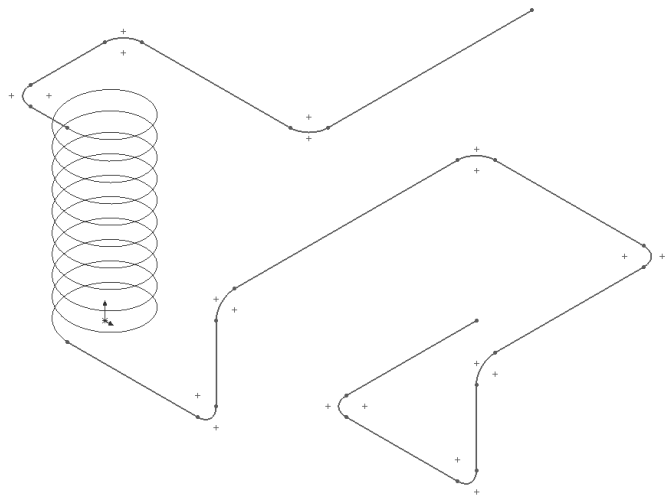
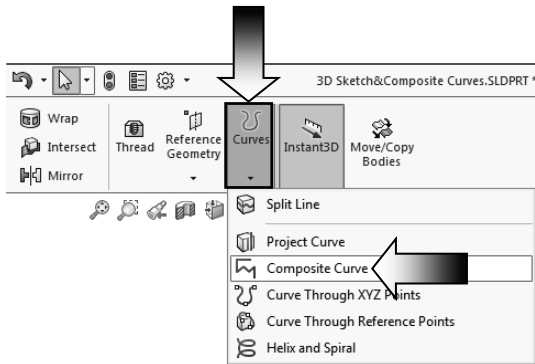
- Select **Insert/3D Sketch** .
- Select the **Line** command  and sketch the 1st line along the X direction.
- Sketch the rest of the lines following their direction shown below.



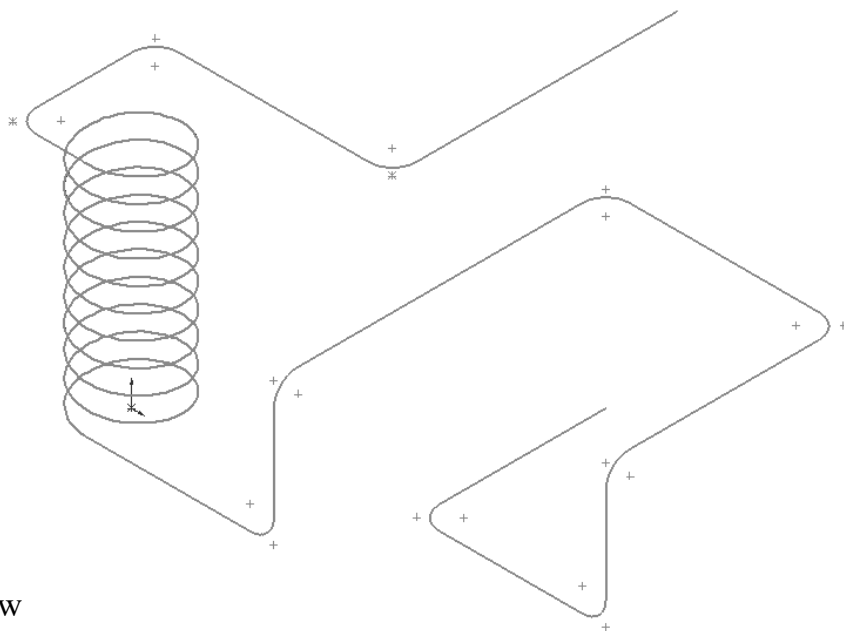
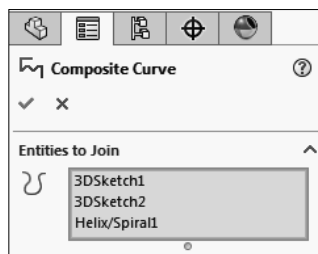
- **Exit** the 3D Sketch  or press **Ctrl+Q**.

5. Combining the curves:

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






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

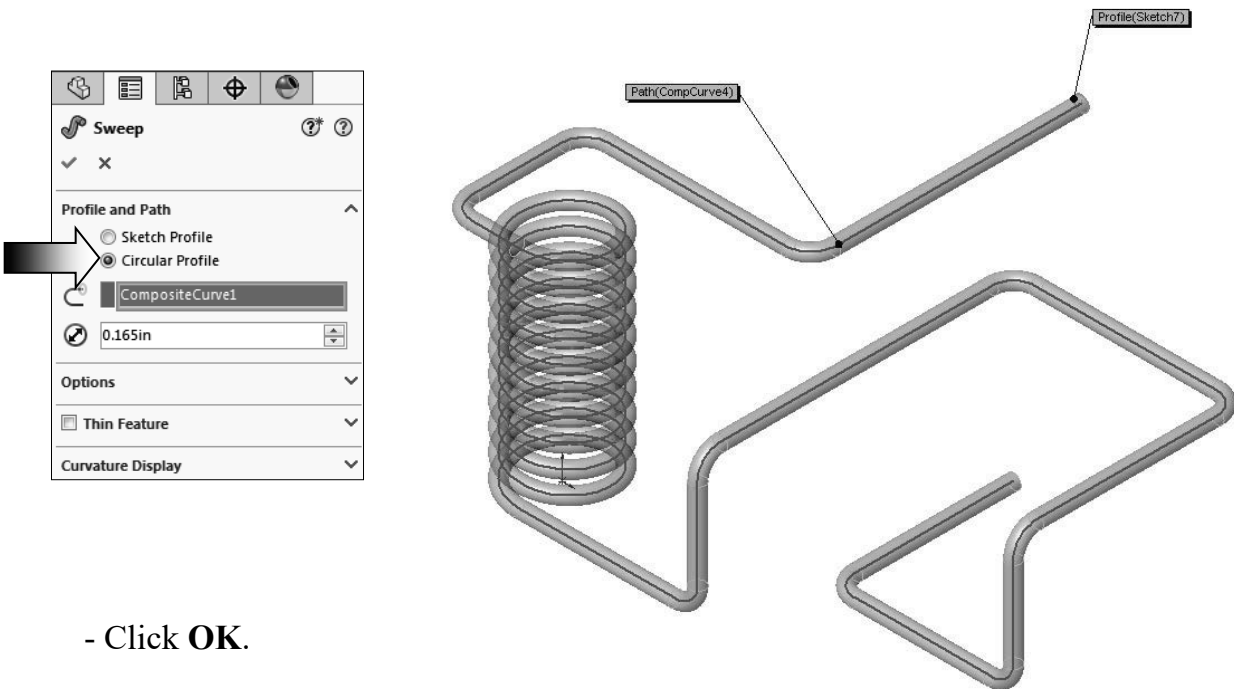


- Click **OK**.

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

6. Creating a Sweep using Circular Profile:

- Select **Insert/Boss Base/ Sweep** .
- Select the **Circle Profile** option (arrow).
- Enter **.165 in** for the diameter of the sweep profile .
- Select the **Composite Curve** as the Sweep Path .



- Click **OK**.

7. Saving your work:

- Click **File/Save As**.
- Enter **3D Sketch_ Composite Curve** for the name of the file.
- Click **Save**.

