

## Inventor Lecture #5

### Reading Assignment:

Read the following in Parametric Modeling with Autodesk Inventor 2009 by Randy Shih:  
Chapter 7 & 9 – Work Planes, Offset Work Planes  
Chapter 11 – Swept Features

### Lecture Outline:

#### Work Planes (Chapter 7)

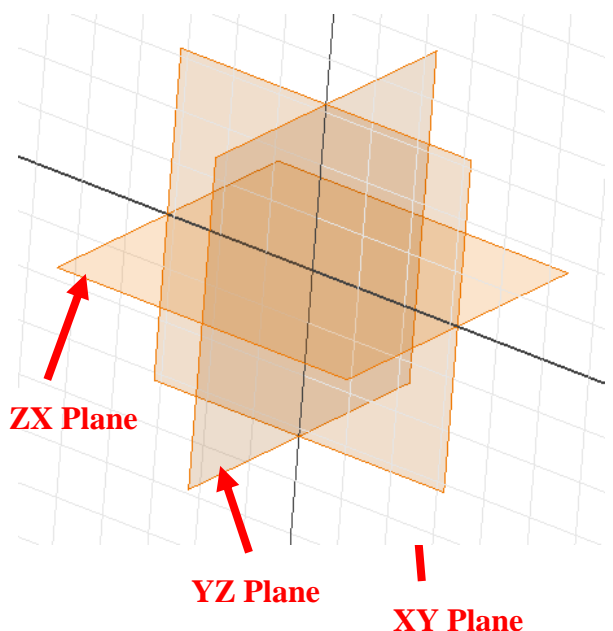
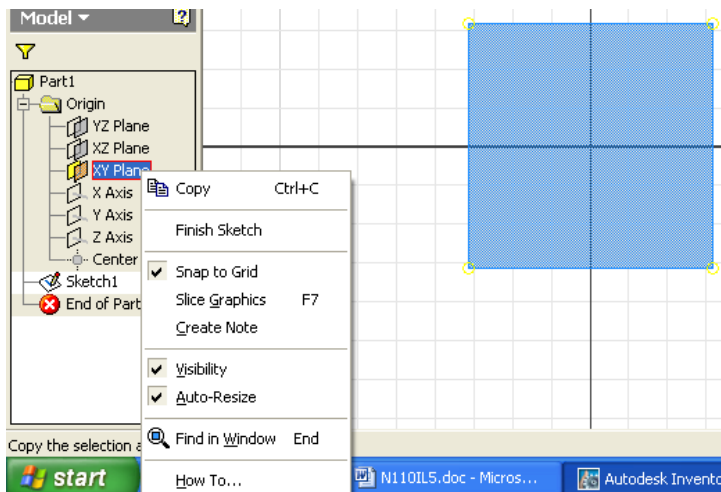
So far we have added **sketch planes** to certain part features that are located in the same plane as a new feature to be added. Sometimes a new feature will be located on a plane that is not available on the current part. In such a case we can add **work planes** where sketches will be formed in order to add new features. The simplest type of work planes use the standard orthographic planes. We will later see other types of work planes.

#### XY, YZ, and ZX Work Planes

Many features can be developed on standard planes in the world coordinate system.

- XY Plane – a plane parallel to the front view
- YZ Plane – a plane parallel to the right side view
- ZX Plane - a plane parallel to the top view

Click on the + symbol next to Origin in the Browser (shown below on the left) to list the available planes. However; note that these planes are not visible by default, so right-click on a plane and check the visibility option in order to see the plane. The image on the right below shows all three planes with visibility turned on from an isometric viewpoint.



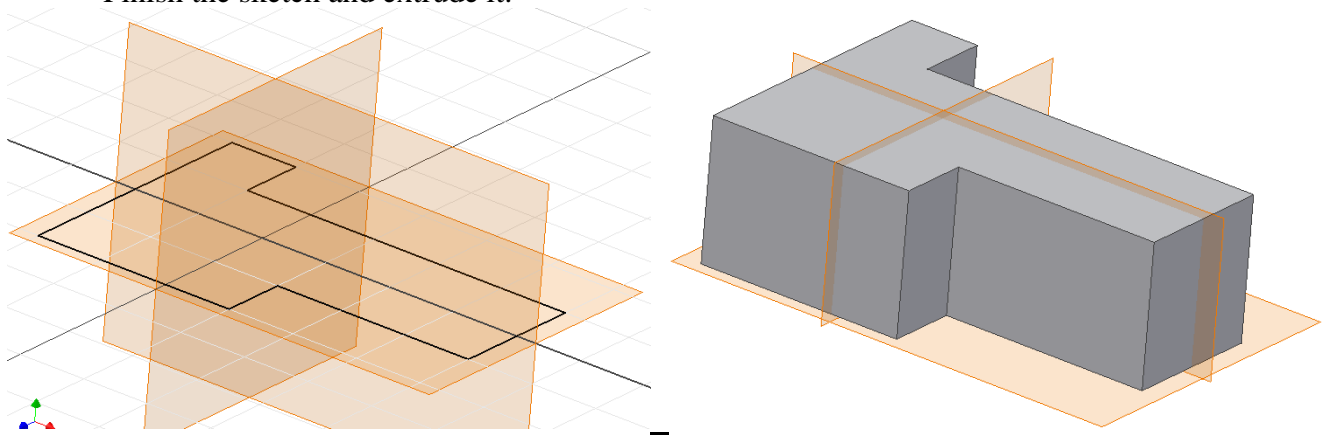
**Starting a drawing using a work plane**

Recall that by default Sketch 1 is in the XY plane when a new part is created (i.e., a front view).

However, we can begin by forming a sketch plane in the XZ work plane if we would prefer to begin sketching from a top view.

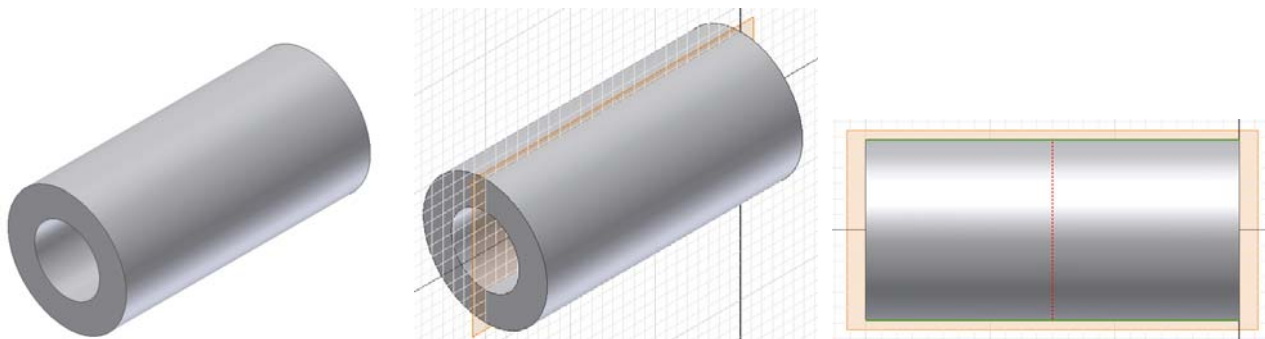
**Example:** Create an object by drawing the base in the XZ (top) plane.

- Make the XY, YZ, and XZ work planes visible
- Select Sketch from the standard toolbar – the status bar area at the bottom of the screen will state “*Select face, work plane, sketch or sketch geometry.*”
- Pick the XZ (top) work plane (the sketch grid should now appear on it).
- Draw a figure in this plane (or use Look At to look at the plane in true shape).
- Finish the sketch and extrude it.

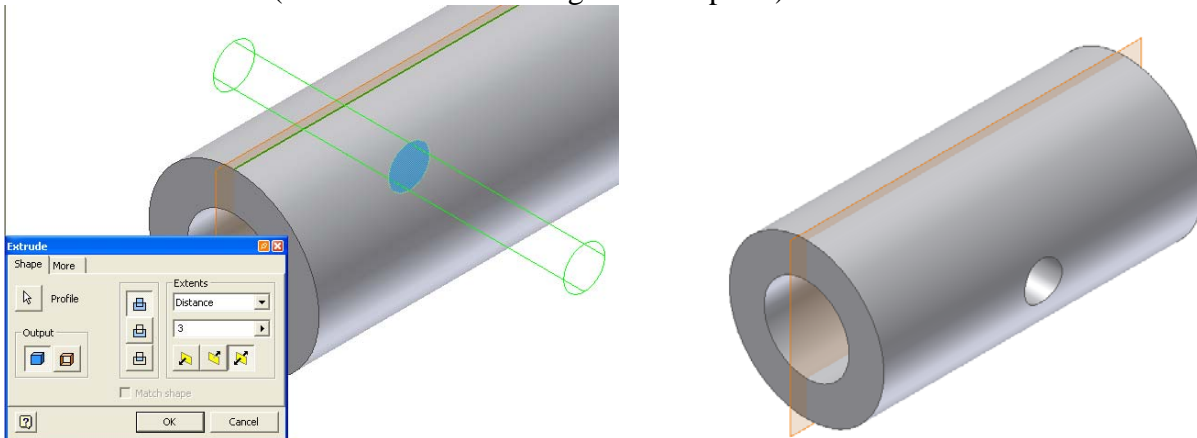


**Example** – Create a hollow cylinder (pipe) and then cut a hole through the side of it.

- Draw two concentric circles and extrude the area between them to form the pipe below (left).
- There are no convenient surfaces onto which we can attach a sketch plane, so use a work plane. Below (center) the YZ work plane has been made visible and is used as a sketch plane.
- Add a construction line (midpoint to midpoint) so that a circle can be extruded from the midpoint of the construction line. (You may need to use Project Geometry first to project the sides of the cylinder onto the sketch plane.)



- Extrude the hole (in both directions using the Cut option)



### **Work Planes**

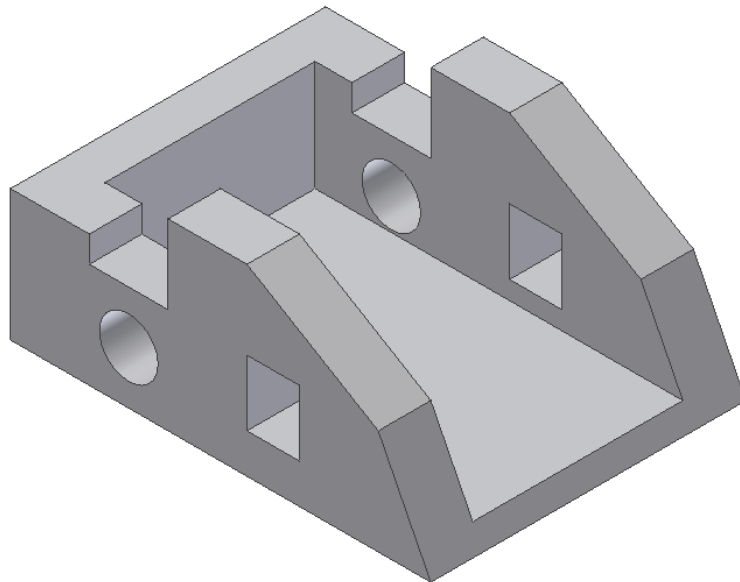
In addition to using the XY, YZ, and XZ planes as work planes, work planes can be created in many other ways. We will next consider the offset work plane.

### **Offset Work Planes** (Chapter 9)

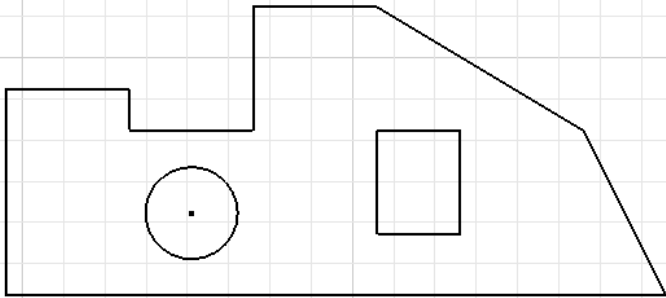
Work planes can be created which are offset from any desired surface. This is done by:

- 1) Selecting Work Plane from the Part Features panel
- 2) Selecting a plane or surface which is parallel to the desired work plane location
- 3) Dragging the work plane to one side or the other (a dialogue box will appear into which you can enter the desired offset distance).

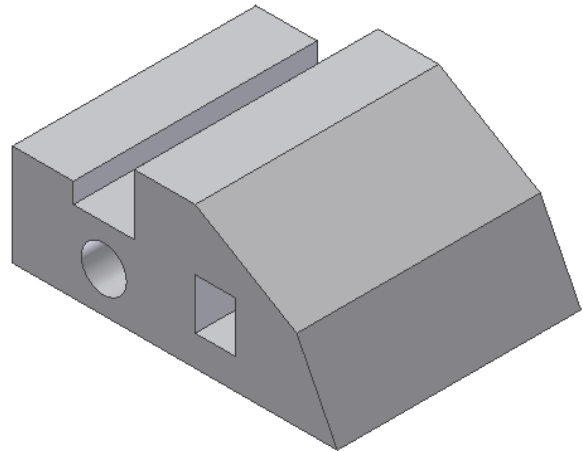
**Example:** Create the object shown below. Use one extrusion to form the main object and then use an offset work plane to create an extrusion that will cut out the center portion of the object.



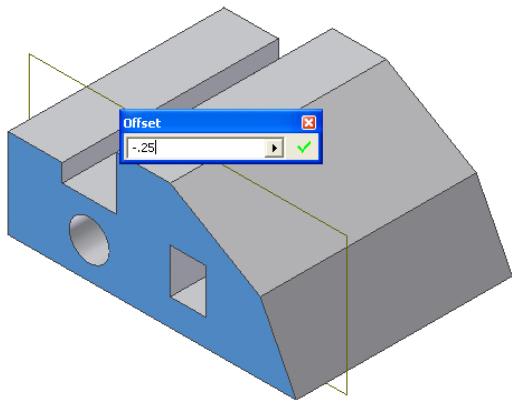
1) Form the sketch.



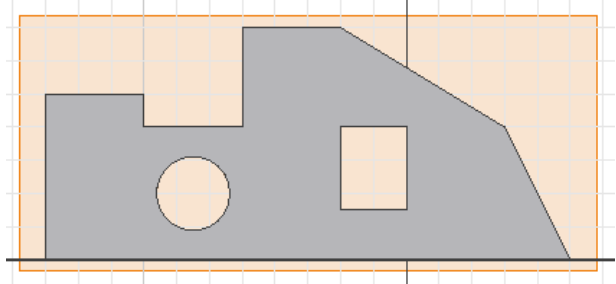
2) Extrude the sketch.



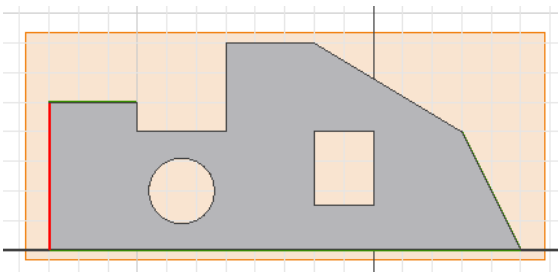
3) Select Work Plane from the Part Features panel, select the highlighted surface below, and drag the work plane toward the back of the object. Enter a value for the desired offset (thickness of the side wall).



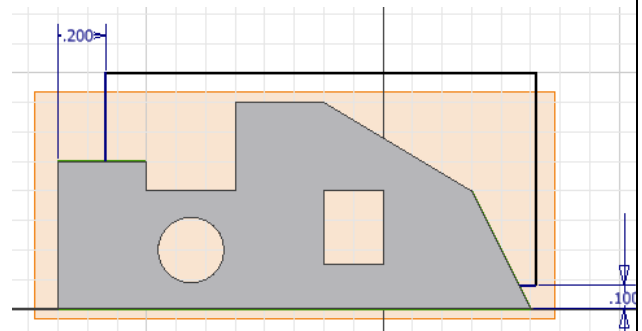
4) Add a sketch plane to the work plane and use Look At to view the side of the object in true shape.



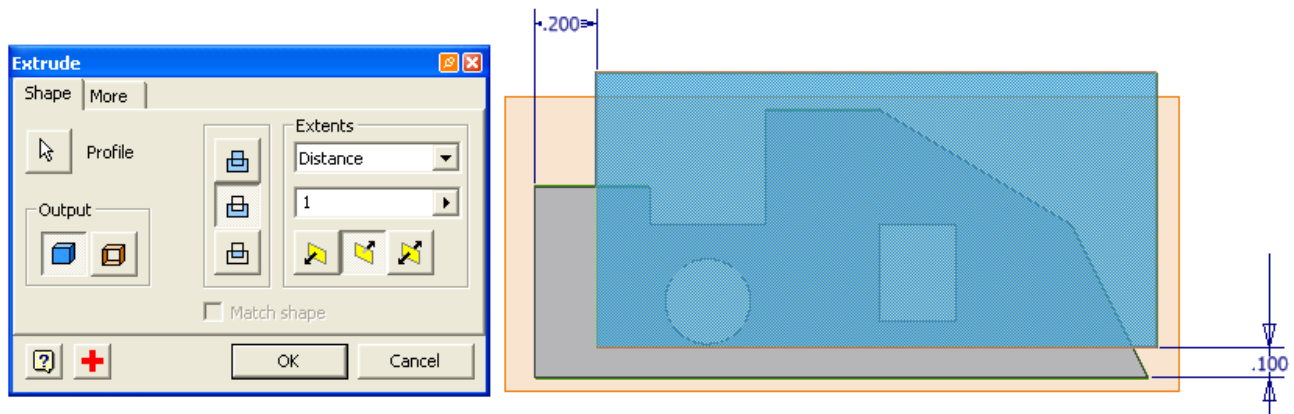
5) Select Project Geometry from the 2D Sketch panel and select lines on the object to be projected onto the sketch plane.



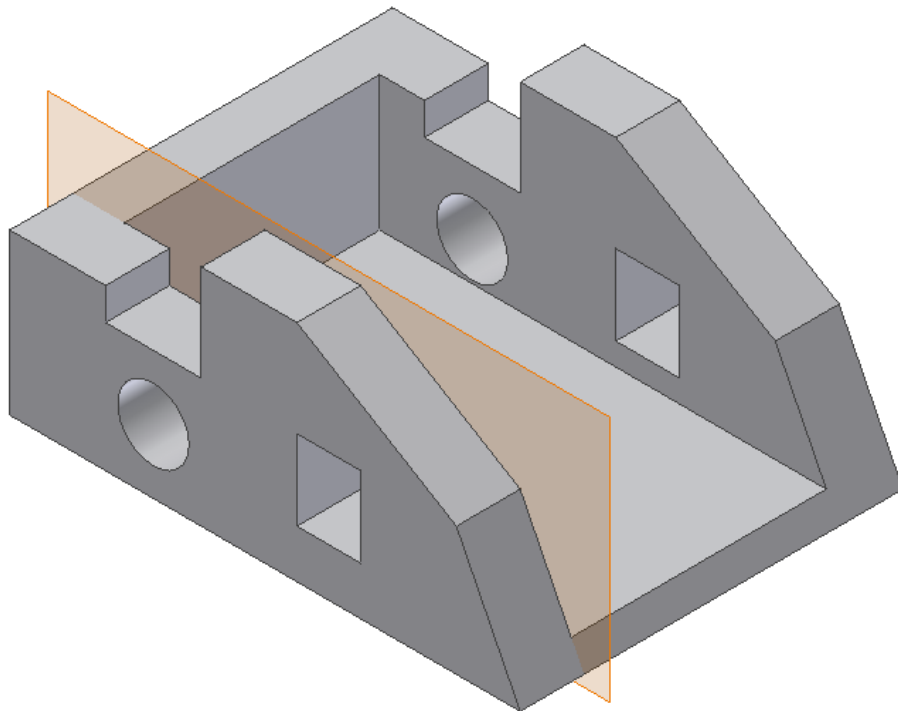
6) Add a rectangle (to be extruded) and use dimensions to control the wall thickness.



- 7) Extrude the rectangle. The depth of the extrusion is the depth of the original extrusion subtract the thickness of the two walls (i.e.,  $1.5'' - 2 * 0.25'' = 1.0''$ )



- 8) Switch to an isometric view. The visibility of the work plane may be turned off by right-clicking on the work plane in the browser and removing the check mark by *visibility*.



## **Features in Inventor**

Recall that Inventor includes two types of geometric features:

- 1) Sketched features (including extrusions, revolutions, and sweeps)
- 2) Placed features (including holes, fillets, chamfers, and shells)

We will next investigate another sketched feature – the sweep.

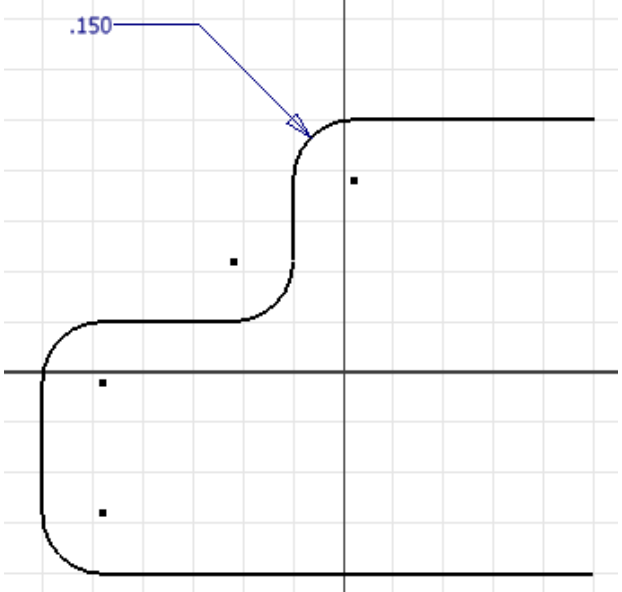
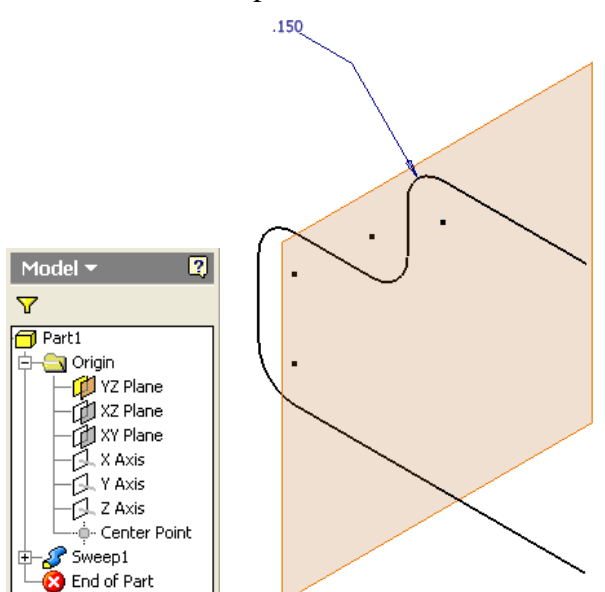
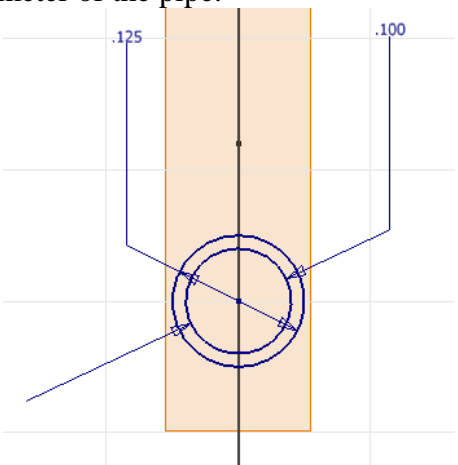
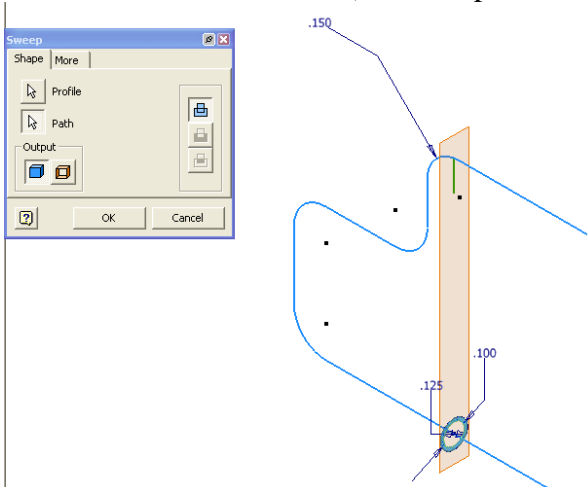
**Swept Features** (Chapter 11)

An extrusion takes any profile and projects it (or sweeps it) along a linear path. The sweep feature in Inventor will sweep a profile along any path. This is very convenient for creating wires, tubing, etc., that follows some path.

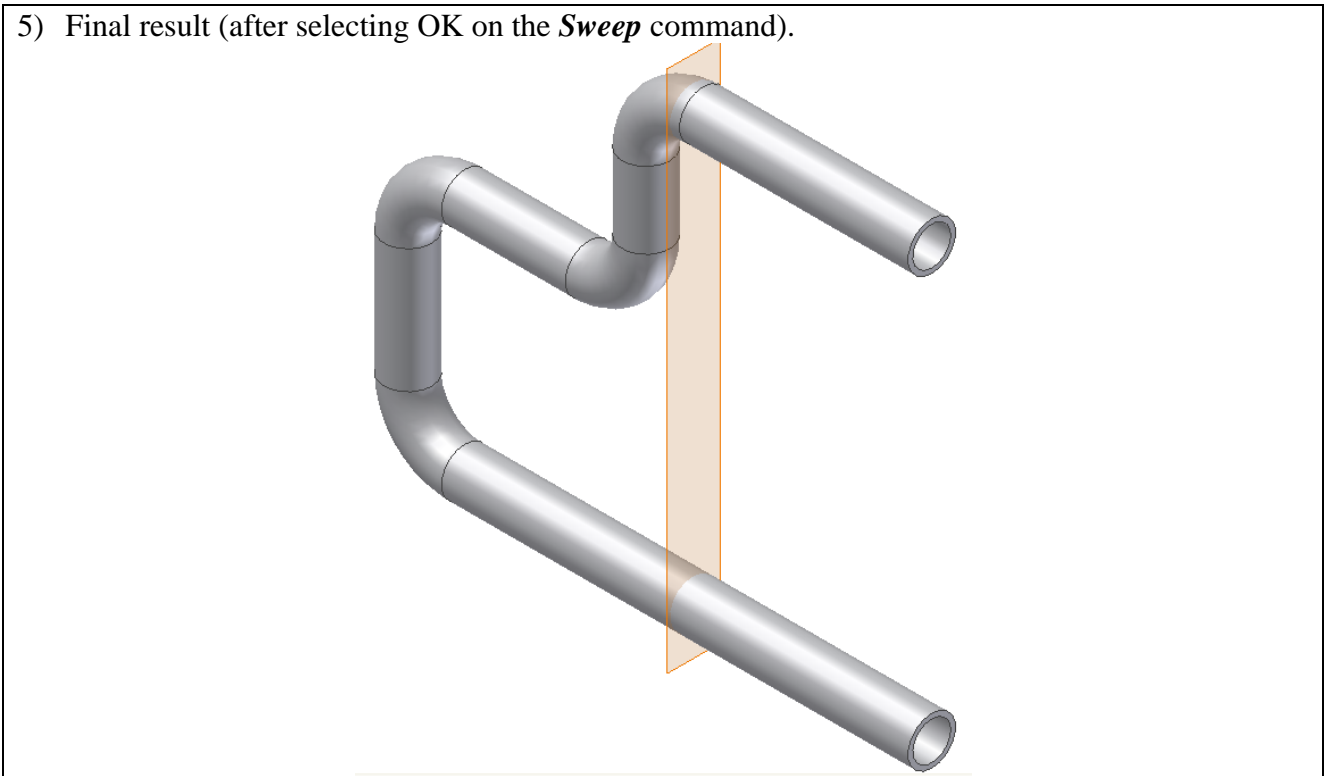
A swept feature in Inventor requires two sketches:

- 2D path (along which the profile will be swept)
- Profile (created in a plane that is perpendicular to the plane containing the 2D path)

**Example:** Create a section of tubing that follows a curved path.

<p>1) Create the following sketch (the path):</p> 	<p>2) Add a YZ work plane and turn on its visibility. Use <b>Project Geometry</b> to project portions of the path onto the sketch plane.</p> 
<p>3) Add circles representing the inner and outer diameter of the pipe.</p> 	<p>4) Finish the sketch and select <b>Sweep</b> from the Part Features panel. Select the profile (area between the two circles) and the path.</p> 

5) Final result (after selecting OK on the *Sweep* command).



Try some other examples using the *Sweep* command:

