



Inventor® Self-paced Learning Modules  
**Autodesk Inventor Advanced**  
Module 13  
Loft Features

### Learning Outcomes:

When you have completed this module, you will be able to:

1. Describe a loft.
2. Apply the command LOFT to blend or transition a solid that connects two or more planer sections or part faces of different shapes.

### Loft Features

A *loft* feature, created with the LOFT command, is a blend or transition solid that connects two or more planer sections or part faces that usually have different shapes. The *planer sections* also called *profiles*. For example, you can create a solid that connects a circular and a rectangular shape as shown in Figure 13-1.

The LOFT command can create complex shapes like the ones used in commercial products. A good example of this would be a cellular phone. In this module, you will be learning how to create relative simple lofts. Complex lofts will be taught in the Inventor Expert Modules.

There is no limit to the number of sections that can be lofted in a single command.

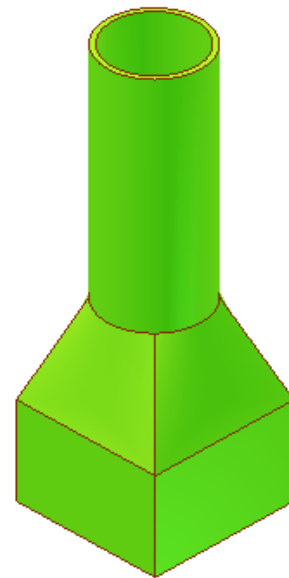
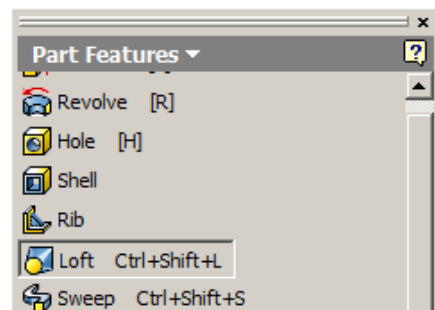


Figure 13-1  
A Loft

### Inventor Command: **LOFT**

The LOFT command is used to blend or transition a solid that connects two or more planer sections or part faces usually different shapes.

Shortcut: **Ctrl +Shift - L**



**WORK  
ALONG**

## Creating Loft Features

**Project:** Inventor Advanced

**Part Name:** Inventor Advanced Workalong 13-1

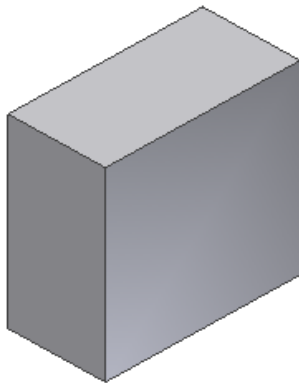
**Template:** English - Modules Part (in).ipt

**Color:** Galvanized (Texture)

**Material:** N/A

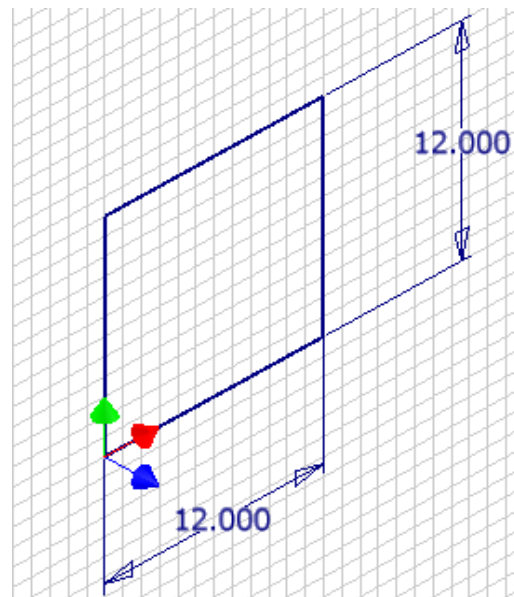
**Step 1** Start a new part using the English template. Save and name the part.

**Step 2** Start a new sketch on the Right Side view and on it draw a square with lines and dimension them as shown in Figure Step 2.



**Figure Step 3**

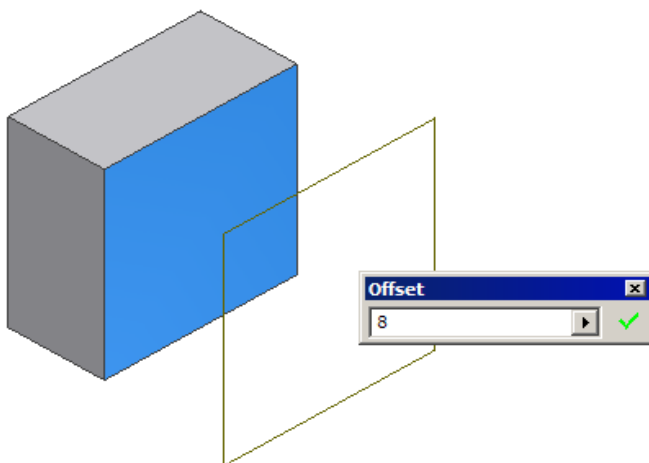
**Step 3** Extrude the sketch 6 inches in the positive Z direction as shown in Figure Step 3.



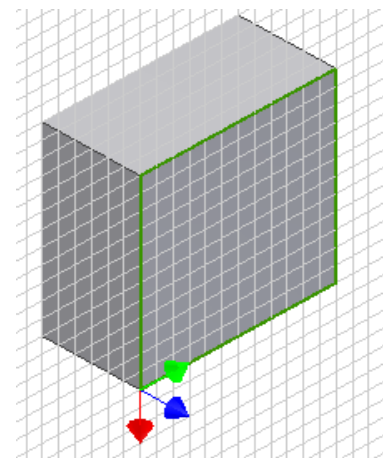
**Figure Step 2**

**Step 4** Start a new sketch in the right side face as shown in Figure Step 4. Return to model mode.

**Author's Comments:** The face of the part will be used so all you have to do is create a sketch for the LOFT command to use.



**Figure Step 5**



**Figure Step 4**

**Step 5** Create a work plane 8 inches from the right side face. See Figure Step 5.

...continued on page 13-3

### Creating Loft Features - Continued

**Step 6** Start a new sketch on the work plane as shown in Figure Step 6.

**Step 7** Use the LOOKAT command to look perpendicular at the work plane. Draw a construction line snapping from one corner to the opposite corner. See Figure Step 7.

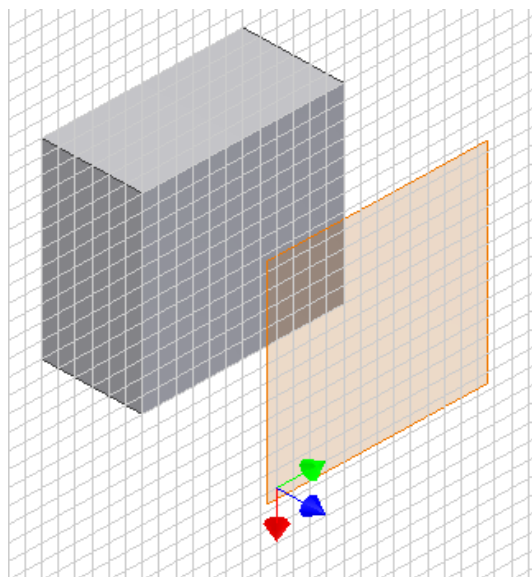


Figure Step 6

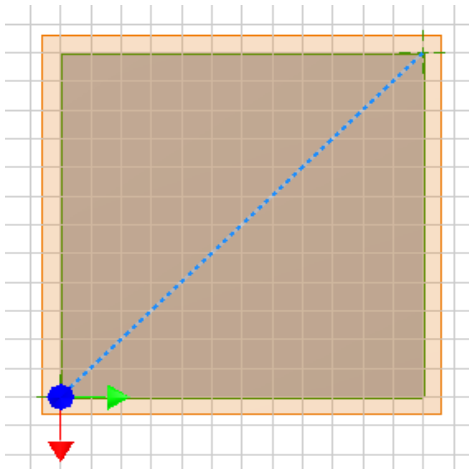


Figure Step 7

**Author's Comments:** By snapping a construction line from corner to corner and then using the midpoint as the center of the circle no location dimensions are required to constrain the circle.

**Step 8** Draw a 6 Diameter circle snapping the center to midpoint of the line. See Figure Step 8.

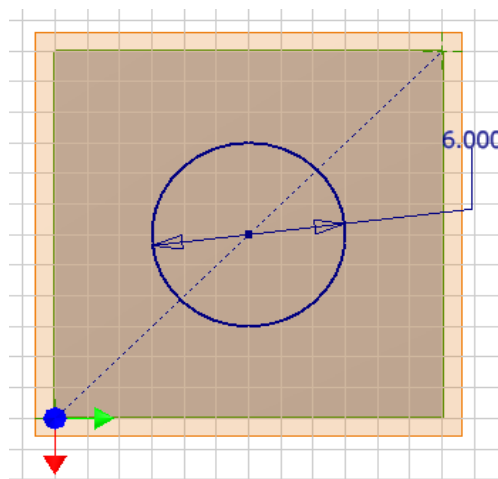


Figure Step 8

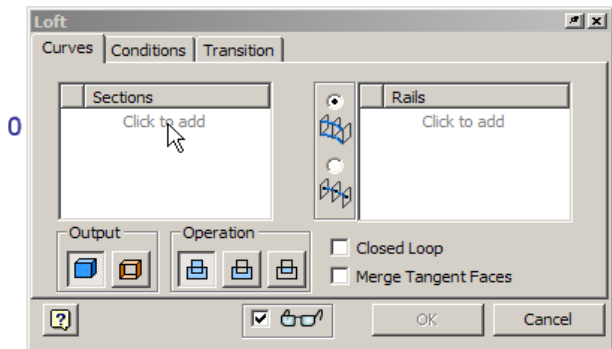


Figure Step 9

**Step 9** Enter the LOFT command. In the Loft dialogue box, click Click to add in the Sections area as shown in Figure Step 9.

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## Creating Loft Features - Continued

**Step 10** Select the edge of the face as first section. See Figure Step 10. Click Click to add again in the Sections area.

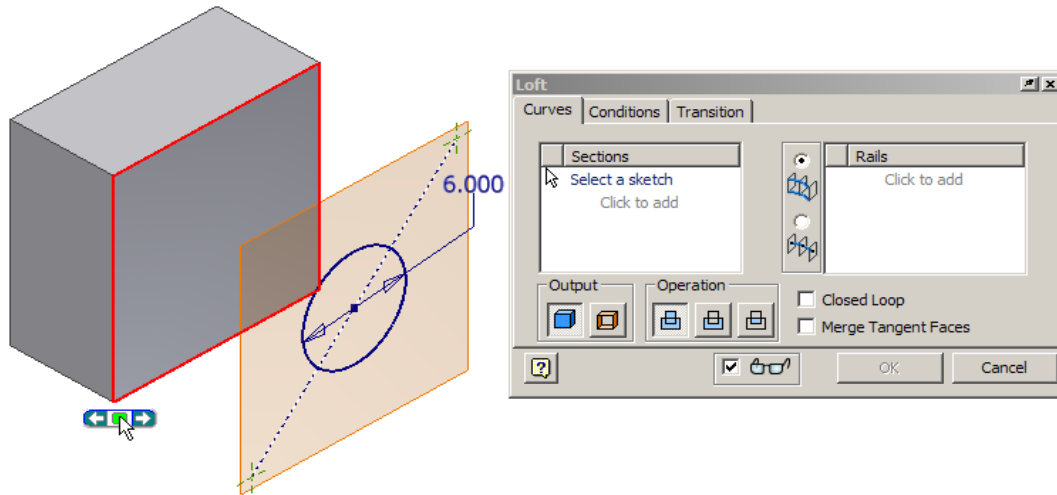


Figure Step 10

**Step 11** Select the circle as the second section. See Figure Step 11.

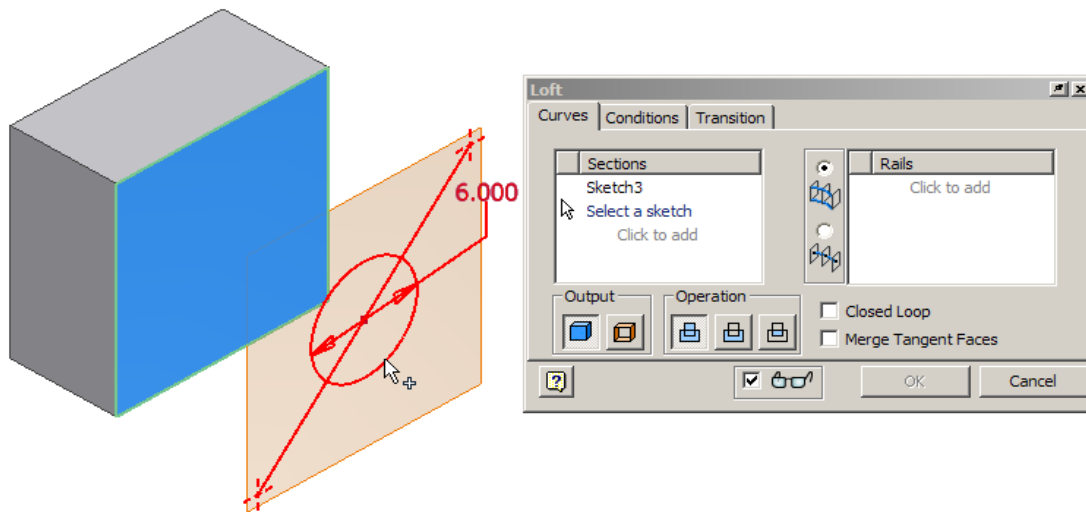
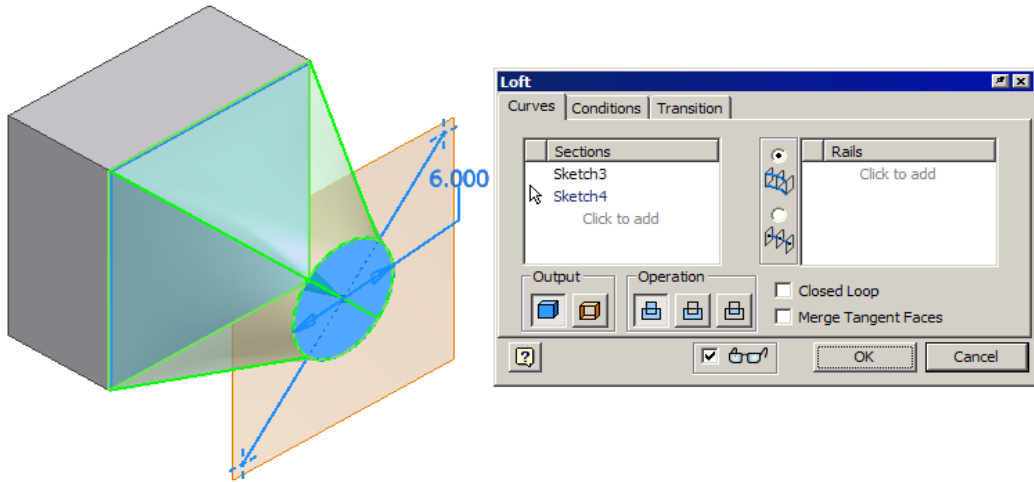


Figure Step 11

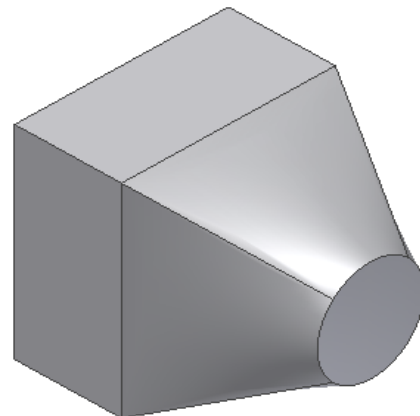
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### Creating Loft Features - Continued

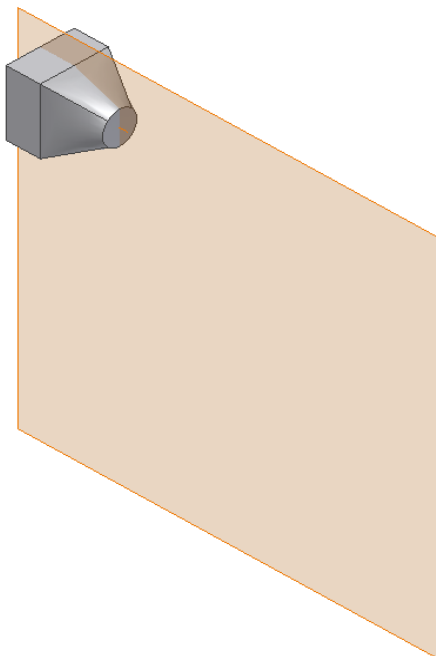
**Step 12** The LOFT command will display the lofted solid that will be created as shown in Figure Step 12A. Click **OK** to accept it. The completed loft should appear as shown in Figure Step 12B.



**Figure Step 12A**



**Figure Step 12B**



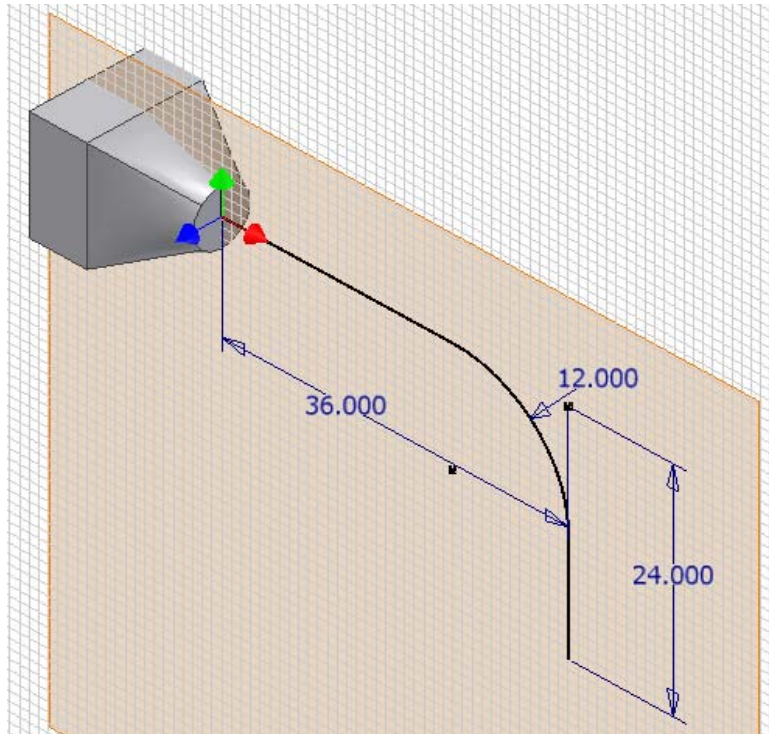
**Figure Step 13**

**Step 13** On the Front (ZX) plane, create a work plane though the center of the part and then enlarge it as shown in Figure Step 13.

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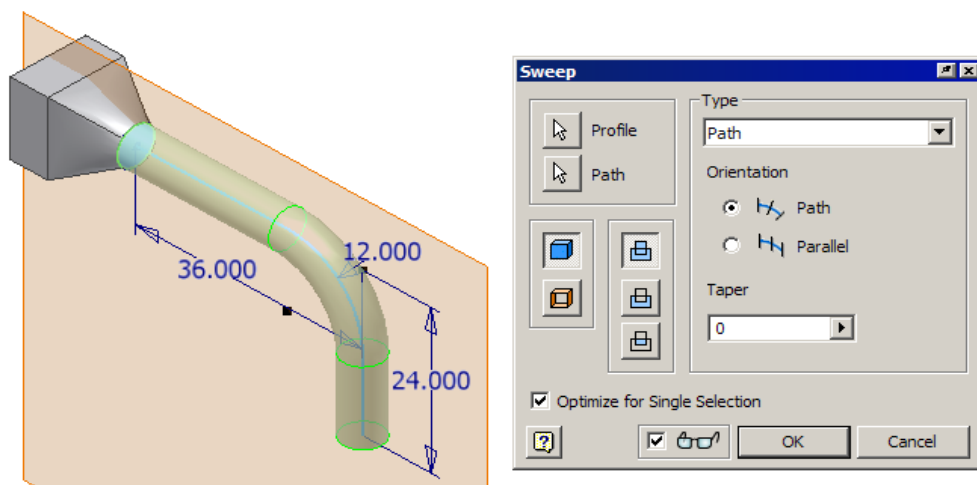
## Creating Loft Features - Continued

**Step 14** Create a sketch on the work plane and draw the lines, add the fillets and dimensions as shown in Figure Step 14.



**Figure Step 14**

**Step 15** Create a loft of the circular pipe along the path you just created. See Figure Step 15



**Figure Step 15**

...continued on page 13-7

## Creating Loft Features - Continued

**Step 16** Start a new sketch at the end of the cylinder you just created as shown in Figure Step 16. Finish the sketch.

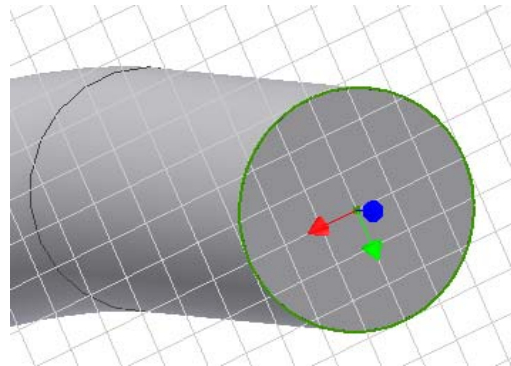


Figure Step 16

**Step 17** Create a work plane offset 10 inches from the end of cylinder. See Figure Step 17.

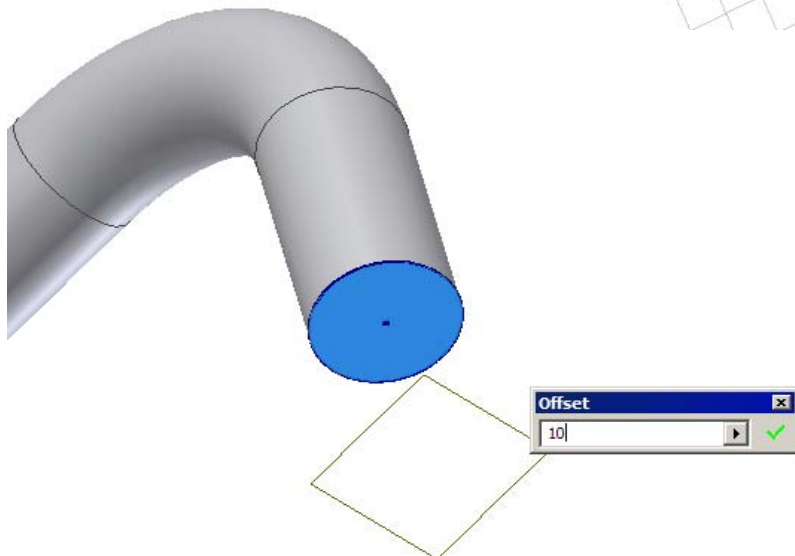


Figure Step 17

**Step 18** Create a sketch on the work plane and change the view to Isometric. See Figure Step 18.

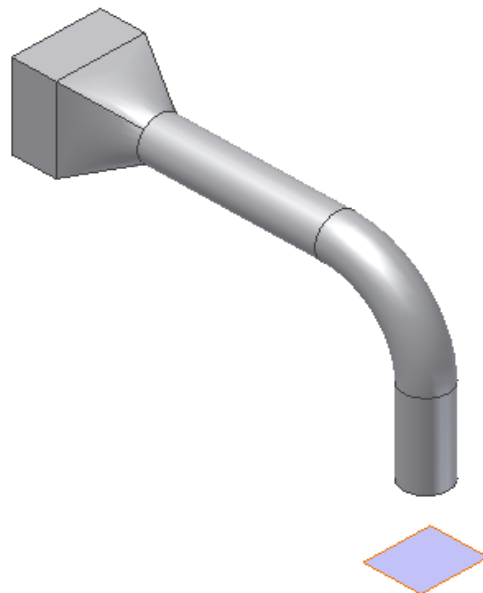


Figure Step 18

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## Creating Loft Features - Continued

**Step 19** On the sketch draw a 8 in by 3 in rectangle as shown in Figure Step 19. Ensure you constrain the sketch.

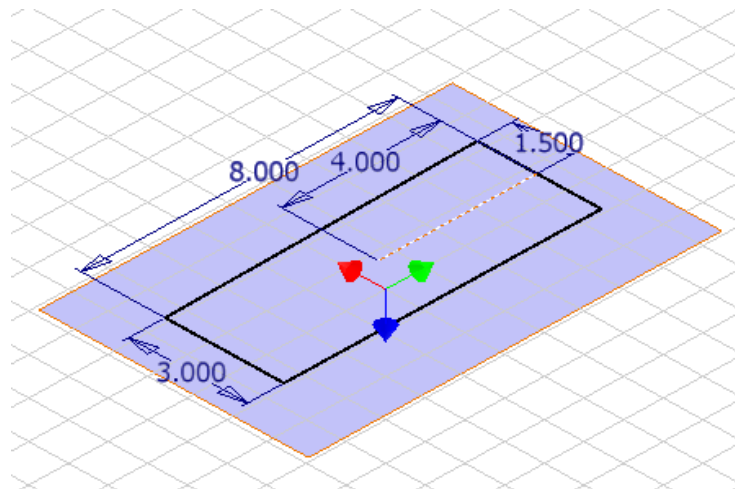


Figure Step 19

**Step 20** Create a loft between the cylinder face and the rectangle as shown in Figure Step 20.

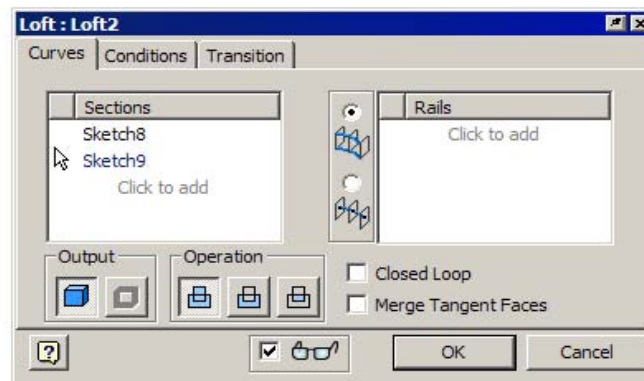
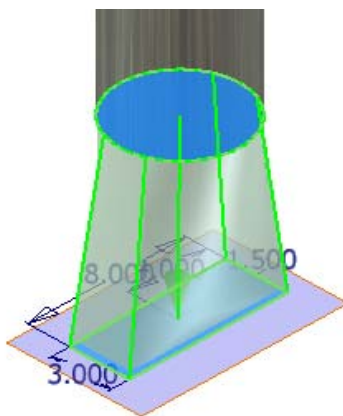


Figure Step 20

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## Creating Loft Features - Continued

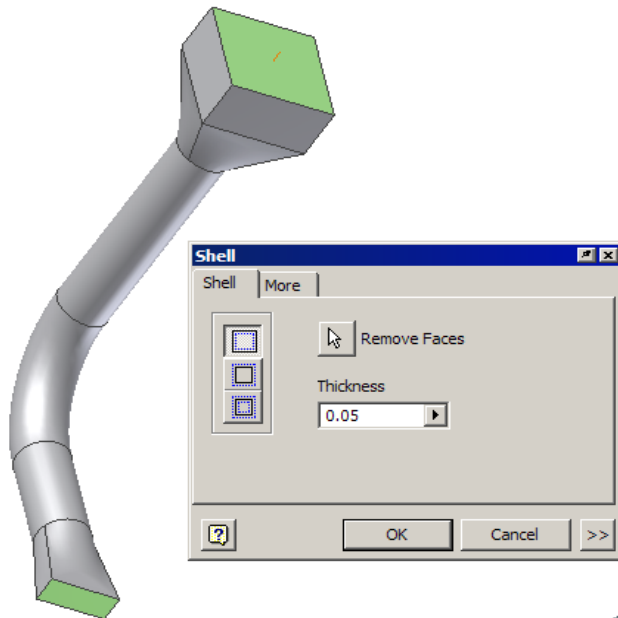


Figure Step 21

**Step 21** Shell the part with a width of 0.05 inches and remove both end as shown in Figure Step 21.

**Step 22** Change the view to Home or Isometric and the color to Aluminum (Polished) as shown in Figure step 22. Save and close the part.



Figure Step 22

### The Key Principles in Module 13

1. A loft feature is a blend or transition solid that connects two or more planer sections or part faces that usually have different shapes. The planer sections also called profiles.

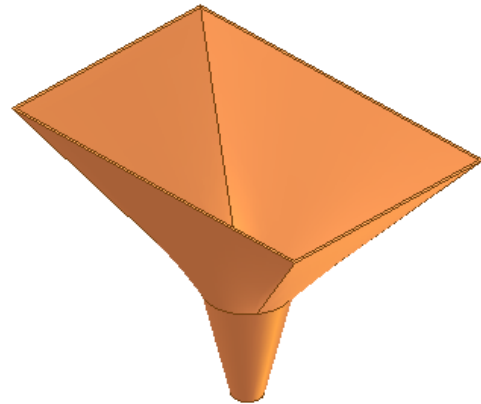
**Lab Exercise 13-1****Time Allowed: 30 Min.**

<b>Name:</b> Inventor Advanced Lab 13-1	<b>Project:</b> Inventor Advanced	<b>Units:</b> mm
<b>Template:</b> Metric - Modules Part (mm).ipt	<b>Color:</b> Orange	<b>Material:</b> N/A

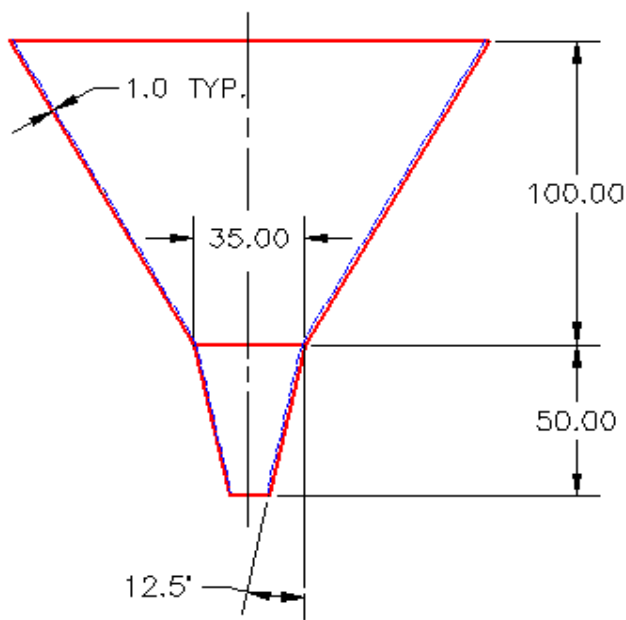
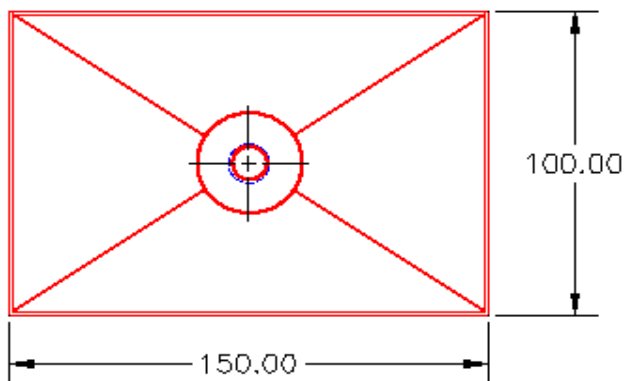
**Instructions:**

- Using the multiview drawing shown below, create the part using lofts. Draw it as a solid and when complete, create the shell. Apply all of the necessary geometrical and dimensional constraints to maintain the objects shape.

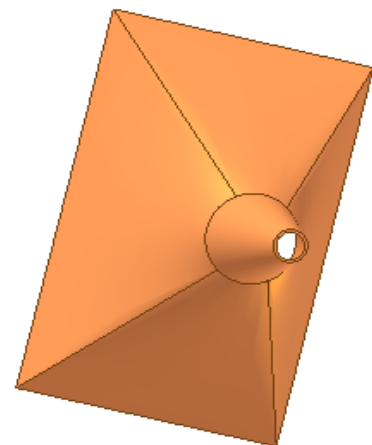
Hint. The bottom taper is easiest to create using a tapered extrusion.



Completed Part -  
Home or Isometric View



Multiview Drawing



Completed Part -  
Rotated View

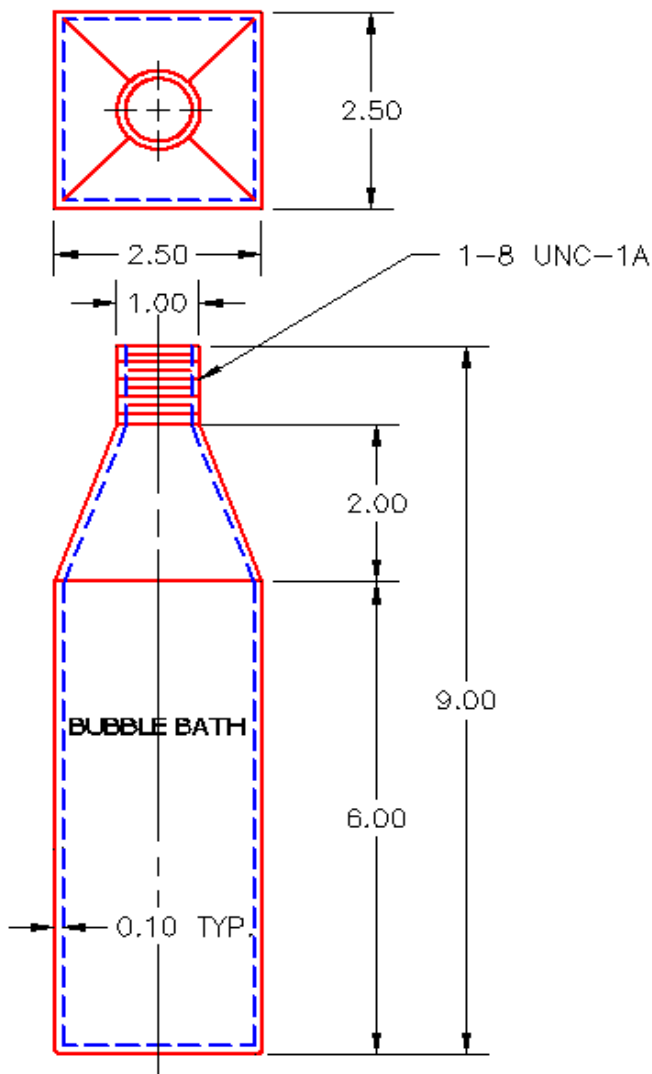
<b>Lab Exercise 13-2</b>		<b>Time Allowed: 40 Min.</b>
<b>Name:</b> Inventor Advanced Lab 13-2	<b>Project:</b> Inventor Advanced	<b>Units:</b> Inches
<b>Template:</b> English-Modules Part (in).ipt	<b>Color:</b> Green (Clear/Polished)	<b>Material:</b> N/A

**Instructions:**

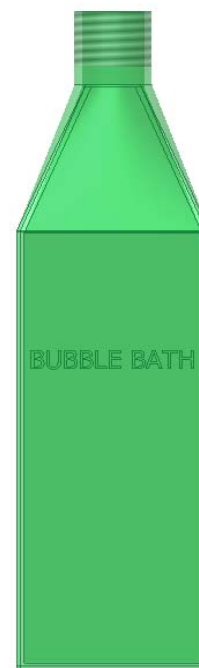
- Using the multiview drawing shown below, create the part using a loft. Draw it as a solid and when complete, create the shell. Draw the thread using the THREAD command. Apply all of the necessary geometrical and dimensional constraints to maintain the objects shape.

Text Specifications:  
 Font: Tahoma  
 Height: 0.250 inches  
 Engraved

Fillet all edges sides and bottom when complete (except for top) 0.05 inch radius.



Completed Part - Home or Isometric View



Completed Part - Front View