Assignment 8

Inventor

Work Planes, Lofts, and Revolves

Objectives

When you complete this assignment you will:

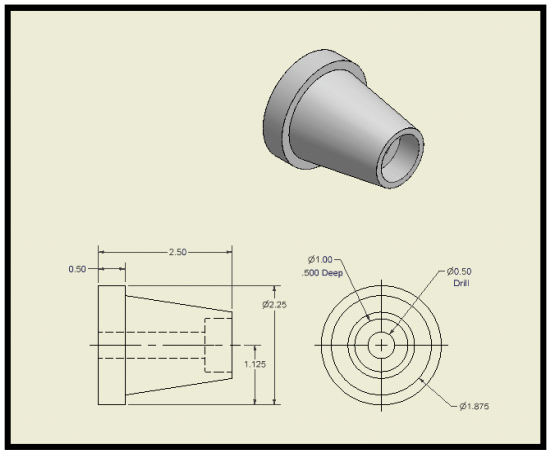
1. sketch and create models using new work planes and the loft command.
2. sketch and create models using the revolve command.
3. sketch and dimension problems.
3. extrude models to remove and add surfaces.
4. view and rotate the three-dimensional models.

Getting Started

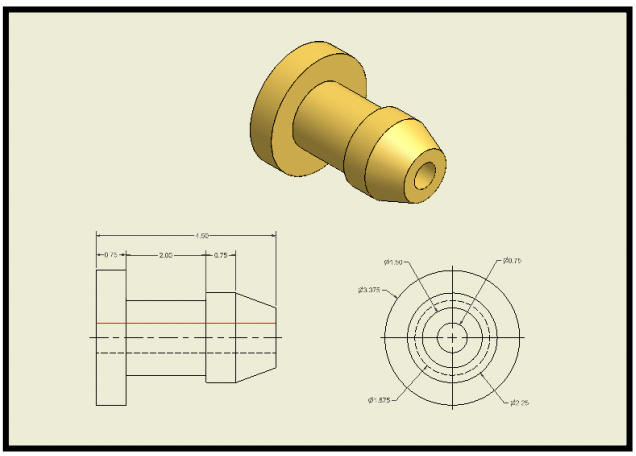
In this activity you will be introduced to three more tools that are useful in Inventor. There is usually more than one way to complete a drawing with this program so as you learn new commands think about when they might be helpful.

1. Launch AutoDesk Inventor 2016 and launch the Standard.ipt template file.

2. In this activity you will be creating the following parts using circles and extrusions as well as a few new commands.



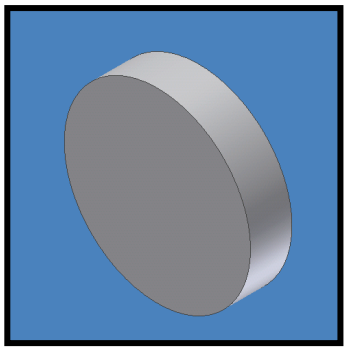
Work Planes & Lofts



Revolve

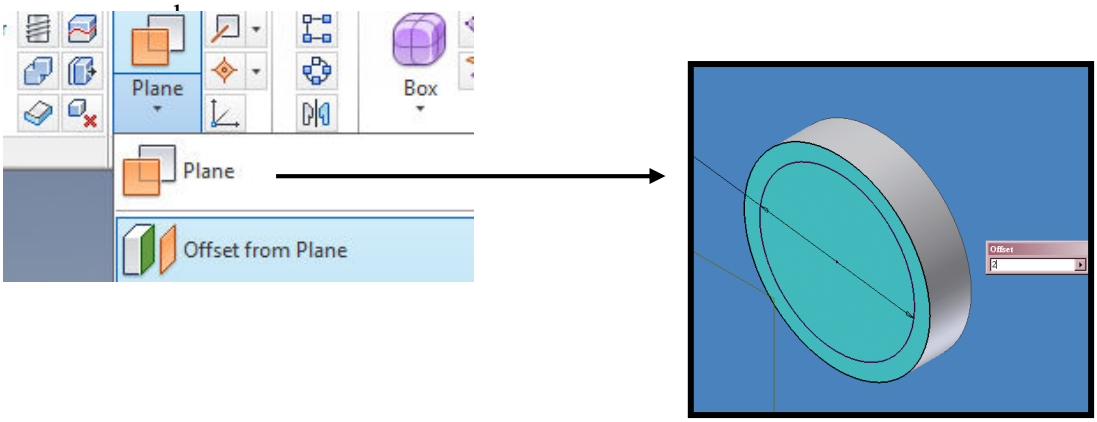
Creating a New Work Plane

1. Create the following part. The diameter of the cylinder should be set to 2.25 and the extrusion thickness is 0.5.

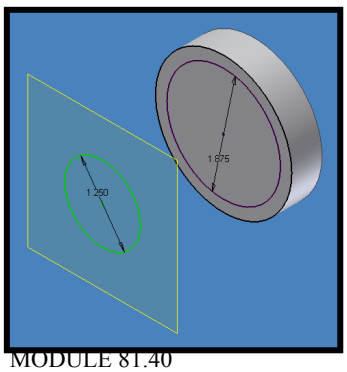


2. Create a new sketch plane on the **front face** of the cylinder and draw a circle with the same center point as the original part with a diameter of 1.875.

3. Finish the sketch and move into an isometric view. Select the **Offset from plan** tool from the 3D Model tab on the Ribbon tool bar and left-click the front face where you drew the last circle. Click the **square work plane** that was created and **drag away** from the face. You will see an **offset dialog box** appear. Enter 2" into the box and press enter or click the green check mark on the offset dialog

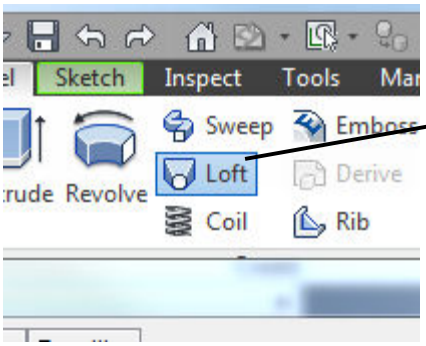


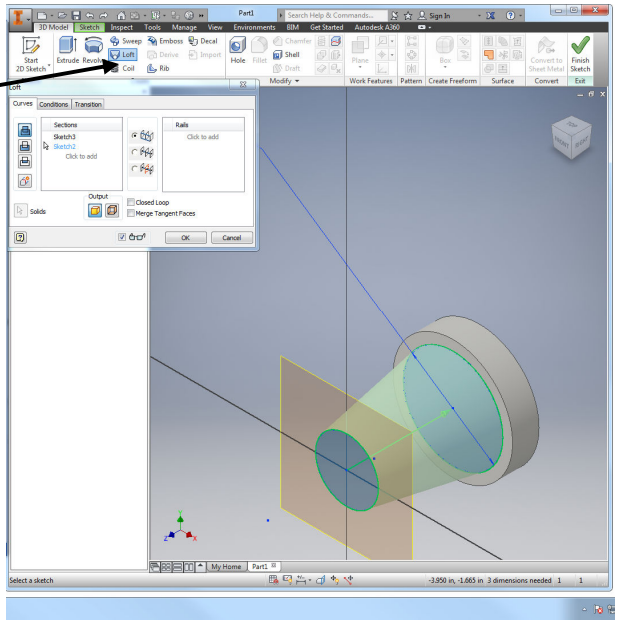
4. You just created a new work plane exactly 2" off the face of the original part. Now create a new sketch on that new work plane (you will have to right click on the outside lines of the plane to create the new sketch). Draw a circle with a diameter of 1.25.



Using the Loft Command

1. **Finish the sketch** of the circle you just created and select the **loft tool** from the Features toolbar. Select the circle on the **hovering plane**, then select the. 1.875 diameter circle located on the extruded base **(you may have to click twice inside the second circle)**.





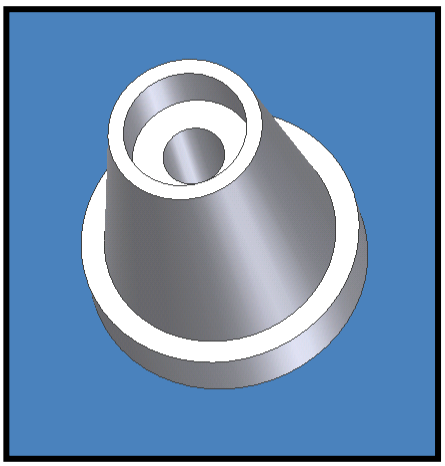
2. Click OK and notice you've just extended the part to the new work plane.
3. Right click the work plane and select visibility closed to remove the work plane.

Finishing the Loft Model

1. You will **finish** the model by **extruding** a .5" deep **bore** with a diameter of 1" and a **drill hole** with a diameter of .5 from the top center of the part. Refer to the engineering drawings on page 8-1 if you need a visual description of these holes.

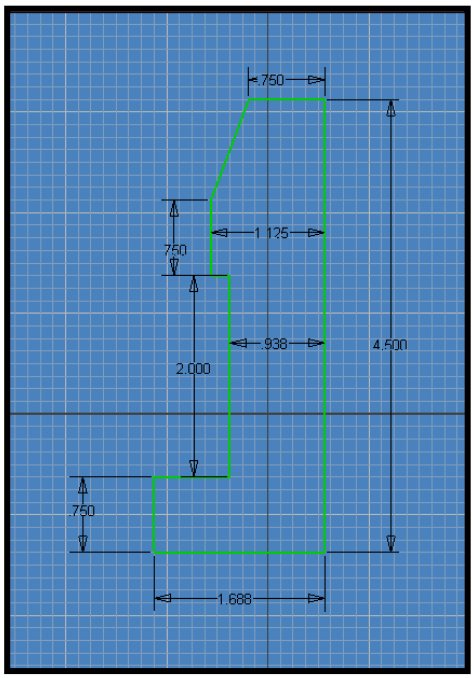
2. Your finished part should look like the one below. If the work plane you created is visible, right click on it a deselect 'Visibility.'

3. **Save the part in your server folder as IN-28**.

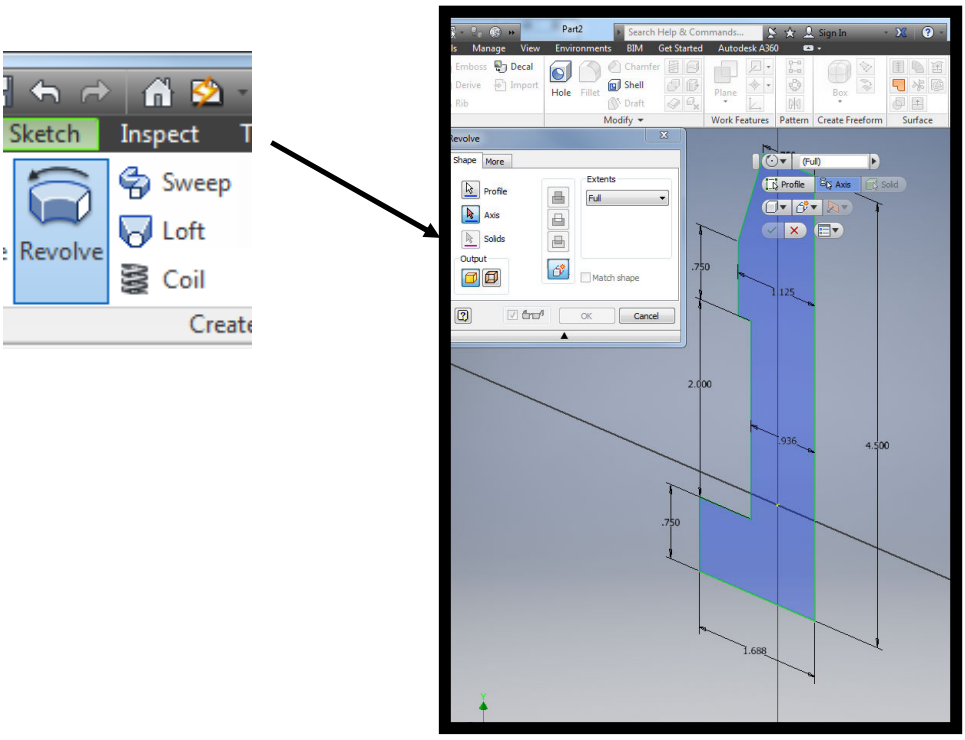


Using the Revolve Command

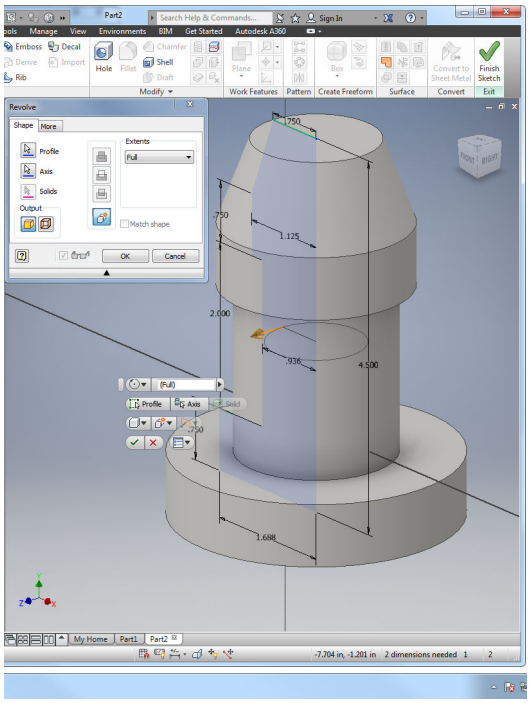
1. Open a new file and create the following drawing. Make sure you dimension the drawing correctly.



2. Finish the sketch and click on the **Revolve tool** on the 3D Model tab on the Ribbon bar. Select the 4.5" **vertical line as the axis**.



3. Click OK. You've just revolved a 2D drawing into a 3D model.

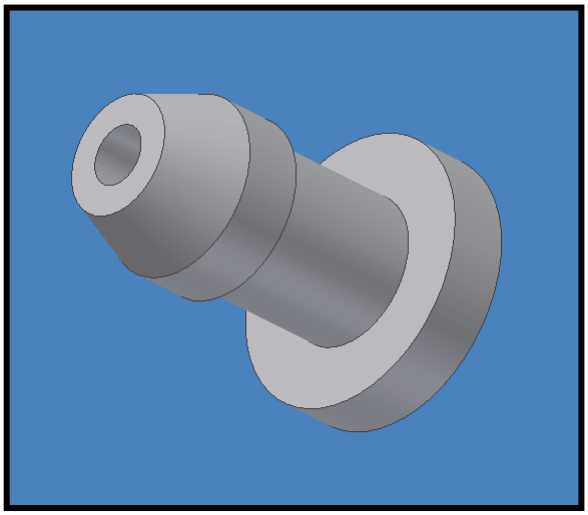


Finishing the Revolve Model

1. You will finish the model by extruding a .75" Drill hole through the center of the part.

2. Your finished part should look like the one below.

3. **Save the part in your server folder as IN-29**.



Finishing Up

1. Great! You've finished some more drawings. **Create a set of engineering drawings for the IN-29 part.**

2. Print and save the drawing into your server folder. Save the file as IN-29. The file extension will be different, so you're original drawing will remain intact and unharmed.

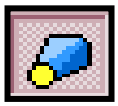
3. Finally, **Create the IN-30 Chess Piece model** from the schematic at the end of this assignment and save it in your server folder.

What Do You Know?

Load Assignment 8 of module 81.40 and answer the following questions:

1. Which of the following icons represents the Work Plane tool?

a. 

b. 

c. 

d. 

2. Which of the following icons represents the Revolve tool?

a. 

b. 

c. 

d. 

3. Which of the following icons represent the Loft tool?

a. 

b. 

c. 

d. 

4. When using the revolve tool, you must select a (an) _____.

a. axis

b. extrusion distance

c. loft angle

d. turning radius

5. True or False? The Work Plane tool allows you to project a new work plane a specific distance from a working model.