

Lesson 14

Lofting

Learning Objectives

When you have completed this lesson, you will have:

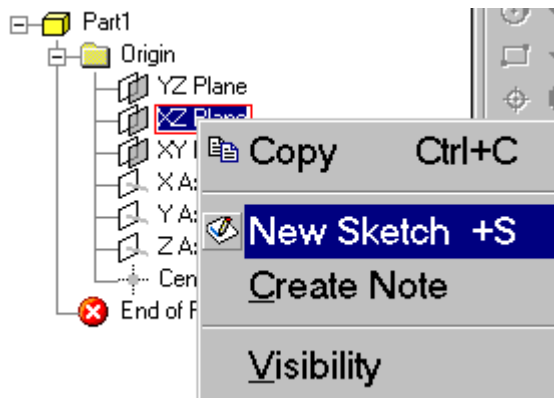
- ◆ Used the precise input toolbar to sketch profiles
- ◆ Created an Offset Work Plane
- ◆ Created a Simple Lofted Model

If you have a part that blends from one shape to another and it cannot be created by extruding or revolving, lofting is the best answer. A lofted solid blends two or more distinct shapes. Before creating a lofted solid, we must define the profiles or sketches to be blended.

Lofts follow these rules:

- ◆ A minimum of two profiles must be specified
- ◆ Each profile must be on it's own work plane
- ◆ Work planes may be offset or angled.
- ◆ Perpendicular Work Planes can not be used
- ◆ An existing planar face may be used for the start or end of a loft

We'll start a new file with standard part using inches.



Exit out of the sketch that Inventor starts us in.

Delete that sketch.

We will create our loft using the Top or XZ plane.

Select the XZ plane in the browser.

Right click and select New Sketch.

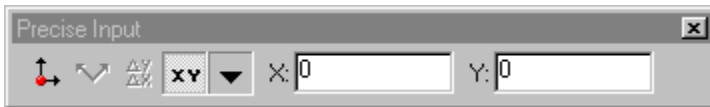
The bottom of our part will be a circle with a diameter of 2 inches. We need to be able to line up our profiles. We can use the Precise Input toolbar to assist us.



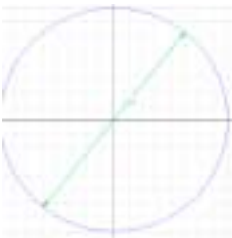
Go to View->Precise Input to enable the toolbar.



TIP: We can enable any toolbar by right clicking on the Standard Toolbar and selecting the desired toolbar.



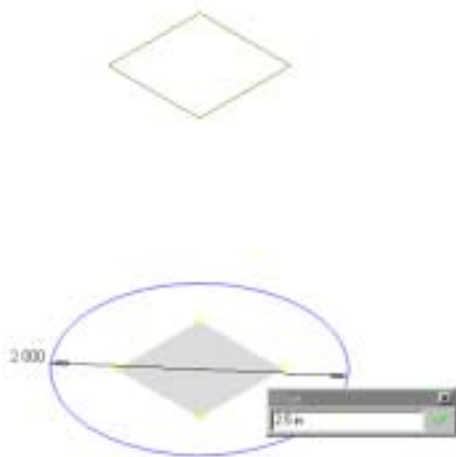
Select the circle tool from the Sketch panel bar. Notice that the X and Y edit windows are now available. Type 0 for X and 0 for Y to locate the circle exactly at 0,0.



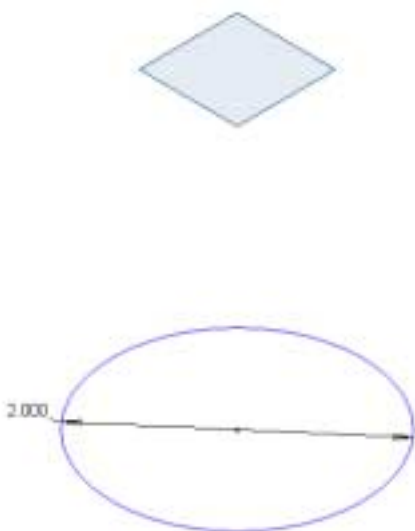
Dimension the circle with a diameter of 2.00.
Select Finish Sketch and switch to 'Isometric View'.



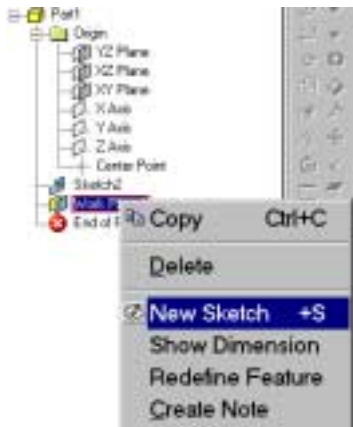
We need to create a workplane offset from the existing XZ plane where the circle sketch is located. Select the 'Workplane' icon located in the Features toolbar.



To locate the new workplane, highlight the XZ plane in the browser to make it visible then select one of the corners of the existing workplane. Hold down the left mouse button and drag the new workplane into position. An Offset text box will appear allowing us to enter in a value. Enter '2.5' and select the green check mark to okay.

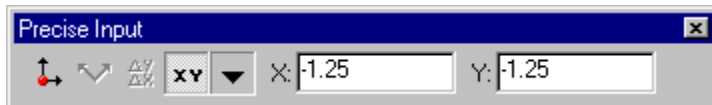


Our new workplane will appear in the Drawing Screen area.

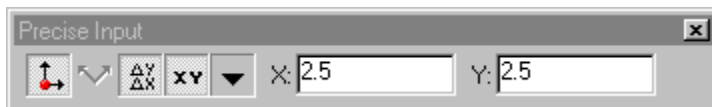


In the Browser, highlight the Workplane, right-click and select 'New Sketch'.

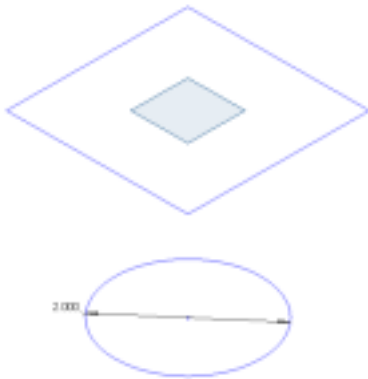
Select the rectangle tool. For the starting corner, type -1.25 for X and -1.25 for Y in the Precise Input toolbar.



Press ENTER to accept the data entry.



For the upper right corner, pick the delta x/delta y button and type 2.5 for X and 2.5 for Y. This sets the rectangle to be 2.5 units wide by 2.5 units high. ENTER to accept the data entry.

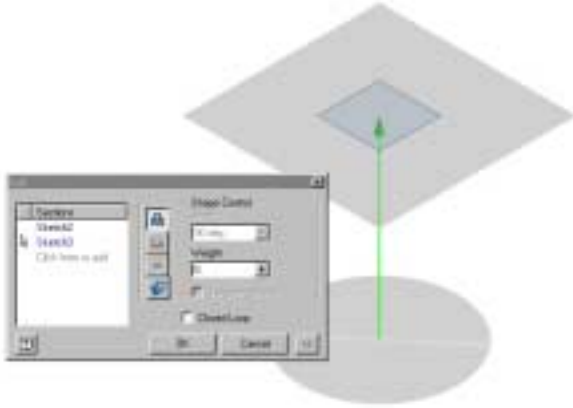


Finish the sketch and switch to 'Isometric View.'



Creating a Loft

Select the 'Loft' tool from the Features toolbar.



Left pick inside the Sections box of the Loft Dialog box.

Then select the circle first and then the rectangle. They should both be highlighted and we should see an arrow indicating the direction of the loft. The direction is based on the order in which the sketches are selected.



TIP: To add a sketch to the Sections window, the user can:

- ◆ Click in the Sections Window and then select the sketch.
- ◆ Pick the sketch in the graphics window and it will automatically be added to the Sections window.
- ◆ Pick the sketch in the browser and it will automatically be added to the Sections window.

Press 'OK'.



Turn off the Visibility of the Work Plane by selecting it in the browser, right click and disabling Visibility.



Use the Orbit tool to rotate the part, so we can see how the loft looks.

Modifying a Loft

Let's edit the 'Loft' feature.

Click on the Loft feature in the browser.



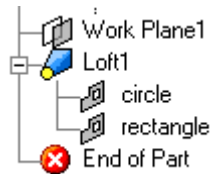
We see the two sketches that comprise the Loft.

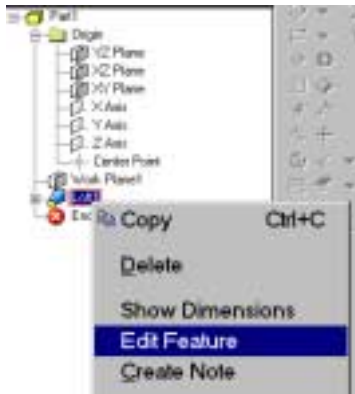
If we pass our mouse over the Sketch, we see the circle and rectangle highlight.

Let's rename each sketch 'Circle' and 'Rectangle'.

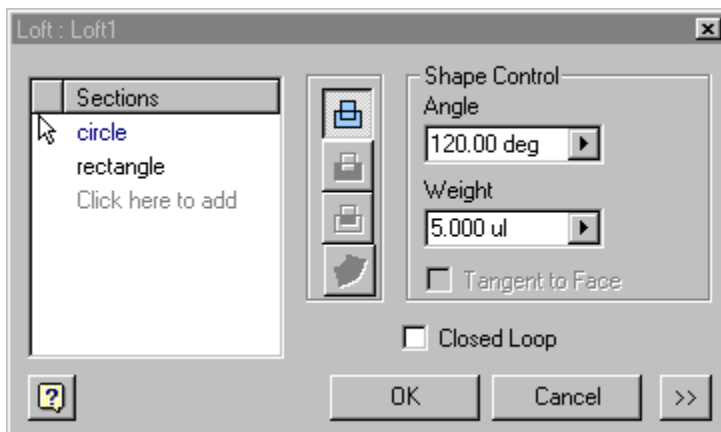
Double-pick on each sketch and rename it appropriately.

(If you double pick quickly, you will open the sketch up for editing. Exit out and try again.)



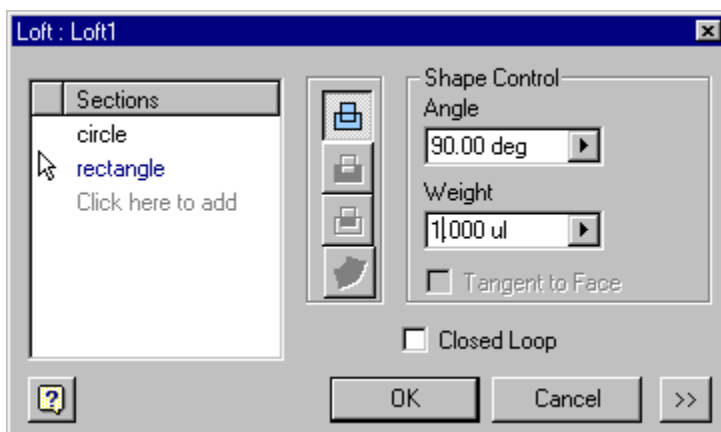


In the Browser, highlight the 'Loft'. Right-click and select 'Edit Feature'.



In the Loft dialog box, highlight the circle. In the 'Weight' edit box, type '5'.
In the Angle edit box, type '120'.

The Angle Edit Box will be greyed out until a value other than 0 is typed in the Weight edit box.



Next, select the rectangle. In the 'Weight' edit box, type '1'. In the Angle edit box, enter '90' as shown. Press 'OK'.



The modified model.



TIP: The larger the weight applied to a loft the more gradual the transition. A small weight creates an abrupt transition. The weight value is a ratio based on the size of the model.

Use the Orbit tool to examine the model.

Experiment with different weights and angles to see how they affect the lofting process.

Save our part as 'loft2.ipt'.



TIP: If your loft shows too great a transition between profiles, you can either reduce the distance between the work planes or add additional profiles to the loft.



TIP: To delete a sketch to be used as a section, pick it with the left mouse button, so that it highlights in blue, then press the 'Delete' key on your keyboard.

Review Questions

1. Lofts require all of the following EXCEPT:
 - A. Two or more profiles
 - B. A Path
 - C. Each profile to be on it's own work plane
 - D. Work Planes can not be perpendicular

2. True or False

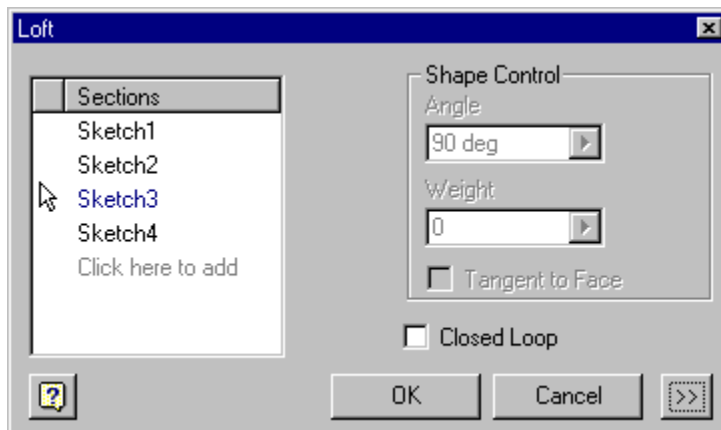
Lofts are used when the feature has an irregular profile.

3. True or False

Weight controls the amount of transition between profiles

4. True or False

You cannot use an existing face as the start or end of a loft.



5. To eliminate a sketch from the Sections window:
 - A. Highlight the sketch in the Sections window, right click and select 'Delete'.
 - B. Highlight the sketch in the browse, right click and select 'Delete'.
 - C. Highlight the sketch in the graphics window right click and select 'Delete'
 - D. Highlight the sketch in the Sections window and press the 'Delete' key on your keyboard.

6. To add a sketch to the Sections window:
- A. Click in the Sections Window and then select the sketch.
 - B. Pick the sketch in the graphics window and it will automatically be added to the Sections window.
 - C. Pick the sketch in the browser and it will automatically be added to the Sections window.
 - D. All of the above
7. The direction of the loft is based on:
- A. the size of the profiles with the start of the loft using the largest profile and the end being the smallest profile
 - B. the weight of each profile
 - C. the order in which the profiles are selected
 - D. the angle between profiles
8. In order for a profile to be used in a loft, it must be:
- A. fully constrained
 - B. on a different work plane than the other profile(s)
 - C. extruded
 - D. a circle or a rectangle
9. True or False
- Once a profile has been added to a loft, it cannot be deleted.
10. True or False
- Once a loft has been created, additional profiles cannot be added.

ANSWERS: 1) B; 2) T; 3) T; 4) F; 5) D; 6) D; 7) C; 8) B 9) F 10) F