

# Chapter 3

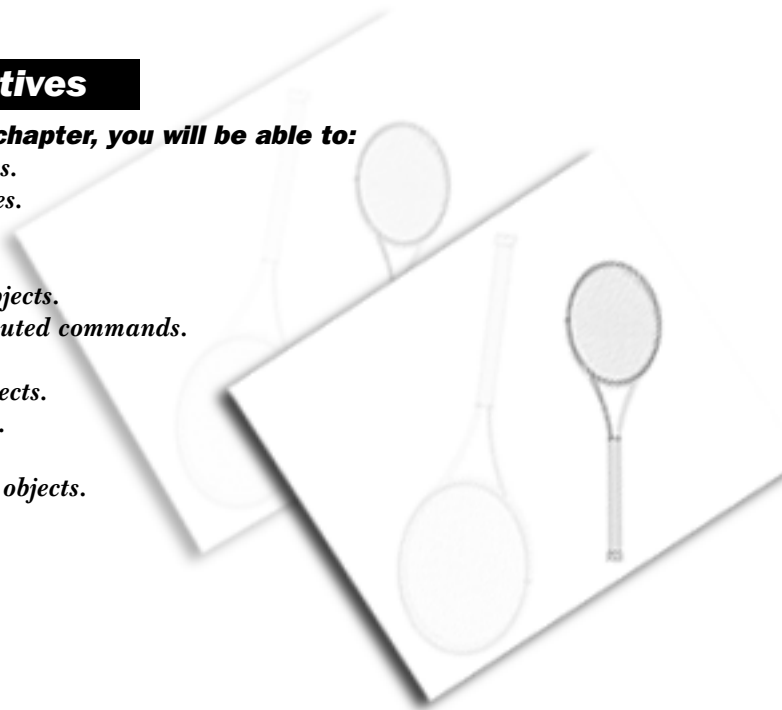
---

## Working with Surfaces-I

### Learning Objectives

**After completing this chapter, you will be able to:**

- *Create revolved surfaces.*
- *Create extruded surfaces.*
- *Create skin surfaces.*
- *Create planar surfaces.*
- *Cut, Copy, and Paste objects.*
- *Undo and redo the executed commands.*
- *Edit keypoint curves.*
- *Create construction objects.*
- *Set construction planes.*
- *Preset grids.*
- *Control the visibility of objects.*



## CREATING SURFACES

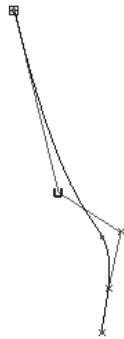
In the previous chapter, you learned about creating curves and primitives, and transforming objects. In this chapter, you will learn to create revolved, extruded, skin, and planar surfaces. These surfaces are discussed next.

### Creating a Revolved Surface

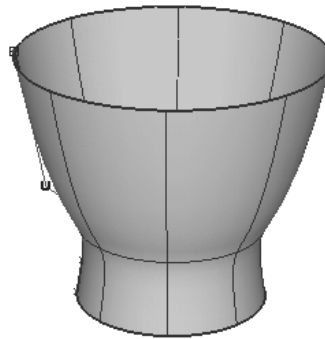
**Palette:** Surfaces > Revolve surface



A revolved surface is created by sweeping a curve around an axis through a specified angle. In AliasStudio, you can create a revolved surface by using the **Revolve surface** tool. To create a revolved surface, first create a curve, as shown in Figure 3-1. Next, choose the **Revolve surface** button from the **Surfaces** tab in the **Palette**; you will be prompted to select the curve to revolve. Select the curve from the active window; the revolved surface will be created, as shown in Figure 3-2. By default, Z-axis will be selected as the axis of rotation. So, the profile will be revolved about the Z-axis.



*Figure 3-1 Profile for the revolved surface*



*Figure 3-2 Revolved surface*

To set the parameters of the **Revolved surface** tool, double-click on the **Revolve surface** button; the **Revolve Options** dialog box will be displayed, as shown in Figure 3-3.

The options in the **Revolve Options** dialog box are discussed next.

### Revolution Axis

The **Revolution Axis** area is used to specify the axis to be used for revolving the curve. The three radio buttons in this area are discussed next.

#### **X**

Select this radio button to revolve the curve around the X-axis.

#### **Y**

Select this radio button to revolve the curve around the Y-axis.



*Figure 3-3 The Revolve Options dialog box*

### **Z**

This radio button is selected by default and is used to revolve the curve around the Z-axis.

### **Axes**

This area is used to specify the coordinate system for the revolved surface. The radio buttons in this area are discussed next.

#### **Local**

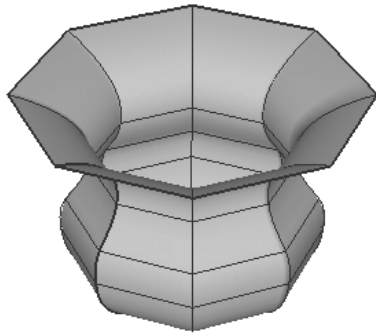
This radio button is selected by default and is used to revolve the curve around its pivot point or local coordinate axes. By default, the pivot point of the curve will be located at the origin of the world coordinate system and the curve will revolve around this axis. If you change the position of the pivot point; a new pivot position will act as the local coordinate system and the curve will revolve around this axis.

#### **Global**

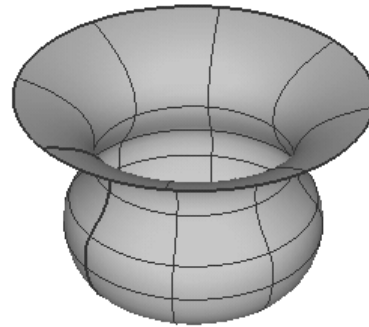
Select this radio button to revolve the curve around the global coordinate system of the current construction plane or axes.

### **Surface Degree**

This area is used to specify the mathematical degree of the revolved surface. The degree of the revolved surface will affect its shape. If you select the **1** radio button, a linear surface with a rough profile will be created, as shown in Figure 3-4. By default, the **3** radio button is selected, which creates a cubic surface with a smooth profile, as shown in Figure 3-5.



**Figure 3-4** Revolved linear surface



**Figure 3-5** Revolved cubic surface

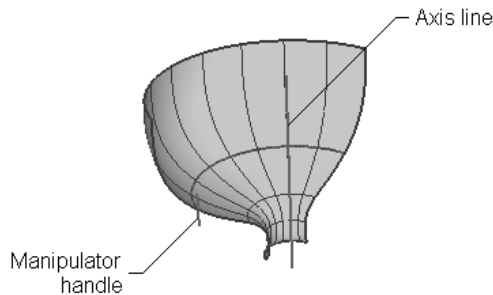
It is clear from these figures that the surface with the surface degree **1** has a rough profile as compared to the surface with the surface degree **3**.

### Sweep Angle

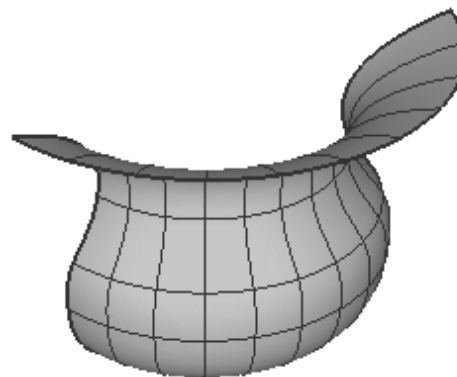
The **Sweep Angle** edit box is used to specify the angle of revolution for the revolved surface. By default, 360 is displayed in this edit box. You can change the sweep angle by entering a desired value in this edit box. Alternatively, you can change this value by dragging the manipulator handle that is displayed on the revolved surface. Figure 3-6 shows the revolved surface with the manipulator handle.

### Sections

The **Sections** edit box enables you to enter the number of sections to be used for creating a revolved surface. You can also change the number of sections by using the slider bar given on the right of this edit box. Generally, the number of sections is kept between 6 to 8. It is recommended that the number of sections should not be more than 12. Figure 3-7 shows a revolved surface having 10 sections with a sweep angle of 150-degree.



**Figure 3-6** Revolved surface with the manipulator handle



**Figure 3-7** Revolved surface having 10 sections with a sweep angle of 150-degree

When you select a curve to create a revolved surface; a manipulator is displayed along with the surface. You can use this manipulator to change the shape, size, and sweep angle of the surface. Click on the manipulator handle and drag the cursor to change the sweep angle. Alternatively, select the manipulator handle and then enter the desired measure of the angle in the promptline. Click at the endpoints of axis line and drag the cursor to move them in the view windows to change the shape of the revolved surface accordingly.

### Create History

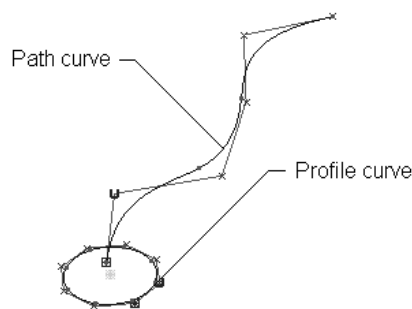
This option is used to save the history of the extruded surface so that it can be modified later. If you create a revolved surface with the **Create History** check box selected, you can modify the profile of the revolved surface by modifying edit points or CVs of the curve. Modifications made on the curve will be reflected in the revolved surface.

### Creating an Extruded Surface

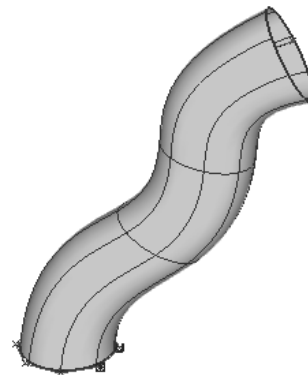
**Palette:** Surfaces > Rail surface > Extrude



An extruded surface is created by extruding a curve or a profile along a specified path. The curve or profile selected for extrusion is termed as the profile curve. The curve selected for guiding the surface is termed as path curve. Figure 3-8 shows a path curve and a profile curve. You can create an extruded surface by using the **Extrude** tool. To invoke this tool, press and hold the left mouse button on the **Rail surface** button from the **Surfaces** tab; a flyout will be displayed. Choose the **Extrude** button from this flyout; you will be prompted to select curve(s) to extrude. These curves can be free curves, curves-on-surface, isoparametric curves, and so on. You can select open, closed, single, or multiple curves as profile curves. Select the profile curve; the **Go** button will be displayed at the lower right corner of the active window. Choose the **Go** button; you will be prompted to select the path curve. Select the path curve; the profile curve will be extruded along the path curve to create an extruded surface, as shown in Figure 3-9.



**Figure 3-8** Profile curve and path curve



**Figure 3-9** Extruded surface

To set the parameters of the **Extrude** tool, double-click on the **Extrude** button; the **Extrude Options** dialog box will be displayed, as shown in Figure 3-10.

Various areas in the **Extrude Options** dialog box are discussed next.

### Style

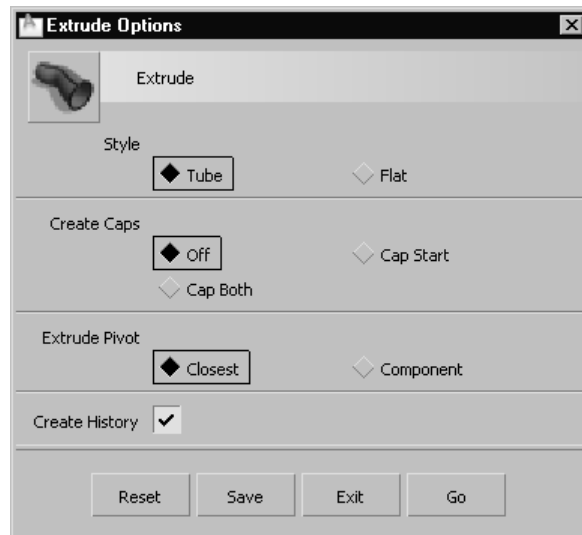
The **Style** area is used to specify the profile when the generation curve is extruded along the path curve. The type of surface that is generated by specifying the profile depends on the orientation of the curve. The radio buttons in the **Style** area are discussed next.

#### Tube

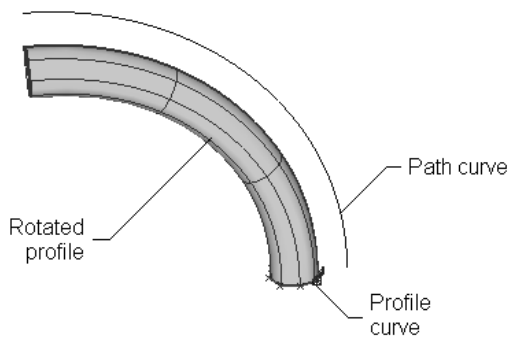
This radio button is selected by default and is used to extrude the surface with the profile maintaining its orientation with the path curve. In other words, the profile rotates with respect to the path curve, as shown in Figure 3-11.

#### Flat

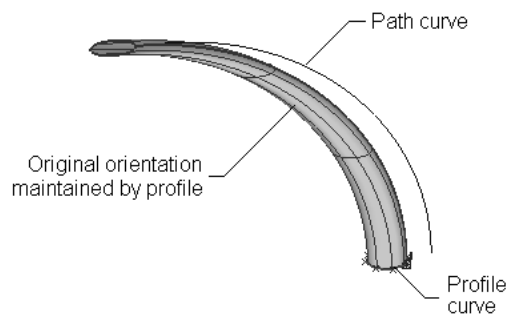
Select the **Flat** radio button to extrude the surface such that the profile maintains its original orientation. In other words, the profile does not rotate while extruding along the path curve, as shown in Figure 3-12.



**Figure 3-10** The **Extrude Options** dialog box



**Figure 3-11** Surface extruded with the **Tube** radio button selected



**Figure 3-12** Surface extruded with the **Flat** radio button selected

### Create Caps

The **Create Caps** area is used to specify whether the caps will be kept at the ends of the extruded surface. The options in this area play significant role in the case of surfaces that are created by using the closed profile curves. The radio buttons in this area are discussed next.

**Off**

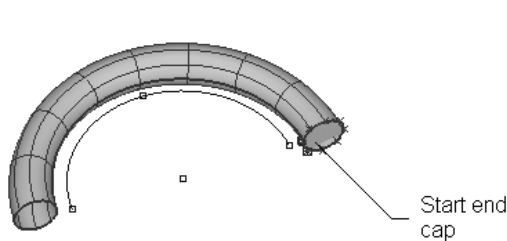
This radio button is selected by default and is used to create an extruded surface that is open at both ends.

**Cap Start**

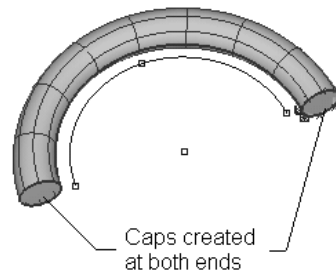
Select this radio button to create the extruded surface with a cap at the first end, as shown in Figure 3-13. However, the surface will be open at the other end.

**Cap Both**

Select this radio button to create the extruded surface with caps at both ends, as shown in Figure 3-14.



**Figure 3-13** Extruded surface created with the **Cap Start** radio button selected



**Figure 3-14** Extruded surface created with the **Cap Both** radio button selected

**Extrude Pivot**

This area is used to choose the pivot point if there are more profile curves. This area will be available only when the **Tube** radio button is selected from the **Style** area. The radio buttons in the **Extrude Pivot** area are discussed next.

**Closest**

The **Closest** radio button is selected by default. As a result, the profile curves are pivoted at the endpoint of the path curve that is closest to the bounding box of all profile curves. For better results, create the profile near the start point or endpoint of the path curve.

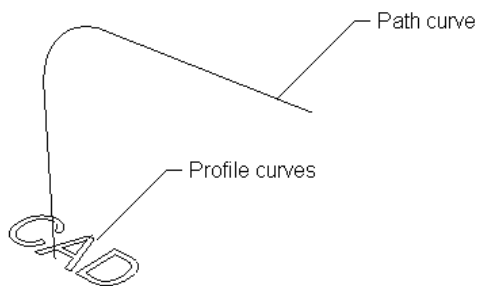
**Component**

On selecting the **Component** radio button, the profile curves pivot around their individual pivot points.

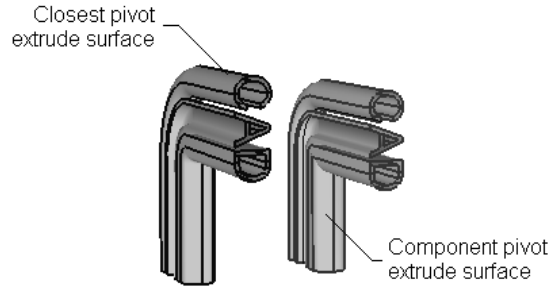
Figure 3-15 shows the profile and path curves and Figure 3-16 shows the closest pivot and component pivot extruded surfaces.



**Tip:** Create the path curve with less number of twists and bends to avoid creating an unwanted twisted surface. Also, it is not mandatory for the generation curve to lie on the path curve.



**Figure 3-15** The profile and path curves



**Figure 3-16** Closest pivot and component pivot extrude surfaces

## Create History

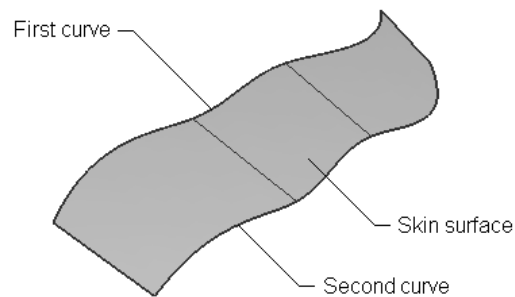
This option is used to save the history of the extruded surface so that it can be modified later. If you create the extruded surface when the **Create History** check box is selected, you can modify the profile of the extruded surface by modifying the edit points and CVs of the profile curve. The modifications made in the profile curve will be reflected in the extruded surface.

## Creating a Skin Surface

**Palette:** Surfaces > Skin surface



You can create a surface between two or more profile curves by using the **Skin surface** tool. This tool allows you to create a freeform surface between the selected curves. Before invoking this tool, create two or more curves. To invoke the **Skin surface** tool, choose the **Skin surface** button from the **Surfaces** tab; you will be prompted to select the first curve. Select one of the curves; you will be prompted to select the next curve or re-select the first curve to undo its selection. Select the other curve from the active window; the skin surface will be created, as shown in Figure 3-17.



**Figure 3-17** Skin surface



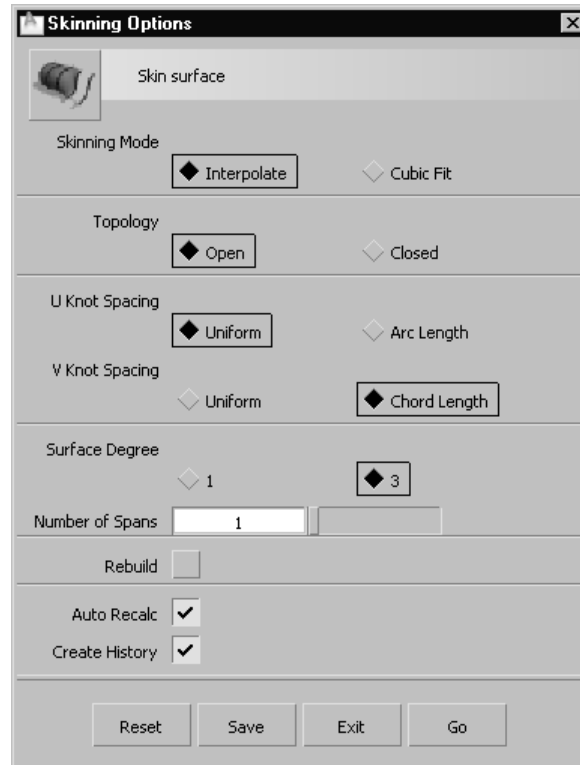
### Note

The display of CVs, hulls, and edit points in Figure 3-17 has been turned off by using the options in the **Display** area of the **Control Panel**.

For creating a skin surface with more than two curves, press the **SHIFT** key while selecting the curves after the selection of the second curve.



To set the parameters of the **Skin surface** tool, double-click on the **Skin surface** button; the **Skinning Options** dialog box will be displayed, as shown in Figure 3-18.



*Figure 3-18 The Skinning Options dialog box*

The options in the **Skinning Options** dialog box are discussed next.

### Skinning Mode

A curve consists of a number of points that can be defined with the help of the U and V parameters. The value of the U and V parameters ranges from 0 to 1. A curve with constant U or constant V parameter is known as isoparametric curve. These parameters are used in mathematical definition of the surface and also for defining paths on the surface. These parameters are not spaced proportionally along the surface. The **Skinning Mode** area is used to control the V parameter of the skin surface. The radio buttons in this area are discussed next.

#### Interpolate

This radio button is selected by default and is used to place V isoparametric curves on original curves. This option gives results only in the case of the curves that have more than one degree.

#### Cubic Fit

On selecting this radio button, V isoparametric curves will be placed on surfaces according to the fitting algorithm.

**Note**

You can increase the number of *V* isoparametric curves by changing the number of spans in the *Pick* area of the **Control Panel**.

**Topology**

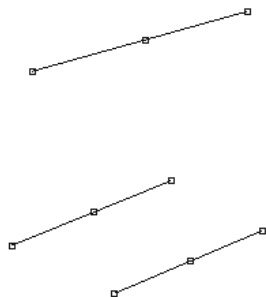
Topology specifies the relationship between objects such as curves, surfaces, boundaries, and so on in a model. The options in the **Topology** area are used to control the spatial relationship between curves used for creating the skin surface. The radio buttons in this dialog box are discussed next.

**Open**

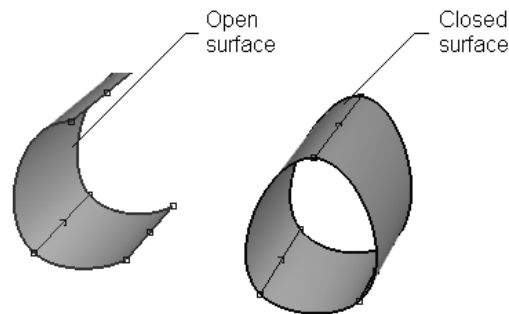
This radio button is selected by default and is used to create an open skin surface.

**Close**

Select this radio button to create a closed periodic skin surface. Using this radio button, you can connect the first curve to the last curve. Figure 3-19 shows three lines to be selected for creating a skin surface. Figure 3-20 shows the open and closed skin surfaces created.



**Figure 3-19** Lines to be selected for creating a skin surface



**Figure 3-20** Open and closed skin surfaces created

**Note**

The skin surfaces in Figure 3-20 have been created by selecting lines with the **SHIFT** key pressed. Therefore, a continuous skin surface is created from a set of more than two curves.

**U Knot Spacing**

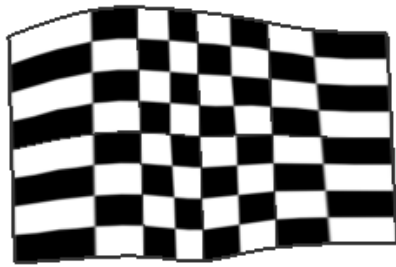
A surface has two directions, *U* and *V*. The *U* direction runs along the length of the curve. The **U Knot Spacing** area controls the *U* parameters in relation to the actual surface. The options in this area are used to control the texture mapping while rendering a model. The radio buttons in this area are discussed next.

**Uniform**

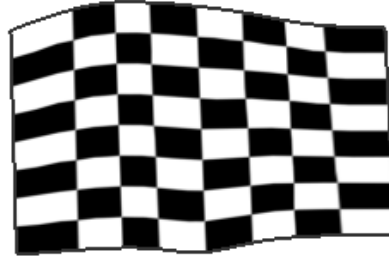
The **Uniform** radio button is selected by default. As a result, the isoparametric curves in the U direction will have the parameter values in integer format such as 1.0, 2.0, 3.0, and so on. Isoparametric curves are spaced unevenly across the surface. As a result, the texture assigned to the curves stretch to fit evenly between isoparametric curves, as shown in Figure 3-21.

**Arc Length**

On selecting this radio button, the edit points of isoparametric curves will be parameterized by average length through original construction curves. The parameter value of the starting edge of the surface is 0.0 and the parameter value at the opposite edge of the surface is equal to the average total length through original construction curves. As parameterization is based on length, isoparms are spaced evenly. As a result, the assigned texture is spaced evenly across the entire surface without stretching, as shown in Figure 3-22.



**Figure 3-21** Assigned texture stretched to fit evenly between isoparametric curves



**Figure 3-22** Assigned texture spaced evenly across the entire surface without stretching

**V Knot Spacing**

The **V Knot Spacing** area is used to control the V parameter in relation to actual surface. The V direction runs across the original construction curves that connects the edit points of the original construction curves. The radio buttons in this area are discussed next.

**Uniform**

On selecting the **Uniform** radio button, the parameter values of isoparametric curves in the V direction will be in the integer format such as 1.0, 2.0, 3.0, and so on. In other words, the isoparametric curves will be parameterized uniformly among the original construction curves through their edit points. This option will be applied only when you select the **Interpolate** radio button from the **Skinning Mode** area.

### Chord Length

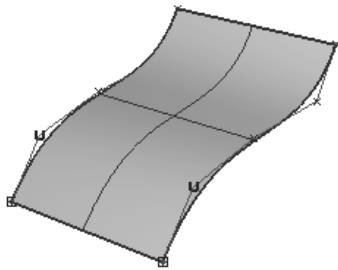
This radio button is selected by default. As a result, the edit points of the isoparametric curves will be parameterized by the chord length of the surface. The parameter value of the surface at starting edge is 0.0 and at the opposite edge is equal to the total length of the surface.

### Surface Degree

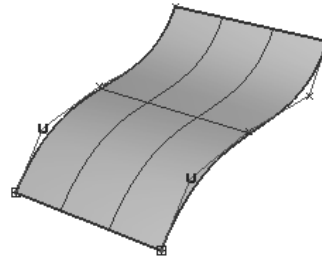
The **Surface Degree** area can be used to specify the degree of surface in the V direction. By default, the **3** radio button is selected and is used to create a cubic surface.

### Number of Spans

The **Number of Spans** edit box is used to specify the number of spans between original isoparametric curves of the skin surface. You can create a surface with a number of spans ranging from 1 to 100. You can also change the number of spans by using the slider bar given on the right of the **Number of Spans** edit box. The default value in this edit box is 1. With this value, only one span can be created on the surface and no additional isoparametric curve will be inserted in the surface. Entering 2 in the edit box will create a surface with two spans. In this case, one additional isoparametric curve will be inserted in the surface, as shown in Figure 3-23. Entering 3 in the edit box will create a surface with three spans. In this case, two additional isoparametric curves will be inserted on the surface, as shown in Figure 3-24.



*Figure 3-23 Surface created with two spans*



*Figure 3-24 Surface created with three spans*

### Rebuild

Select the **Rebuild** check box to match the parameterization of the surface with the first curve. This check box should be selected only when you are sure that the parameterization of the curve and the surface created match each other.

### Auto Recalc

The **Auto Recalc** check box is selected by default and is used to update a surface while adding more curves to it. Note that you need to select the curves to be added with the SHIFT key pressed. When you add curves to the surface, the surface gets automatically updated. If you clear the **Auto Recalc** check box and then select the curves to be added to the skin surface; the **Go** button will be displayed at the lower right corner of the active window. You need to choose the **Go** button to update the surface.

## Create History

This check box is selected by default and is used to save the history of the skin surface so that it can be modified later. If you create the skin surface keeping the **Create History** check box selected, you can modify the profile of the skin surface by modifying the edit points and CVs of the curves that were used to create it. Modifications made on the curves will be reflected in the skinned surface.

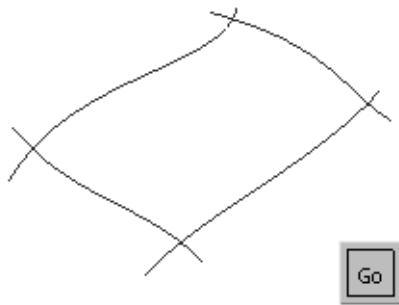
## Creating a Planar Surface

**Palette:** Surfaces > Set planar

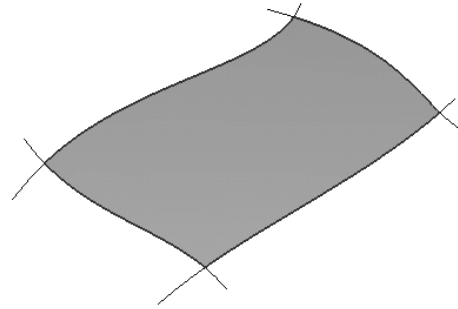


Planar surfaces are the trimmed NURBS surfaces that are created from the curves lying on the same plane. You can create these surfaces by using the **Set planar** tool.

This tool functions only when the selected curves form a closed profile. To create a planar surface, choose the **Set planar** button from the **Surfaces** tab in the **Palette**; you will be prompted to select the first curve. Select the first curve from the active window; you will be prompted to select the other curve(s). Select the curves in such a way that they form a closed profile. You will notice that the **Go** button is displayed on selecting the profile curve(s), as shown in Figure 3-25. Choose the **Go** button after selecting all curves; the planar surface will be created, as shown in Figure 3-26.



**Figure 3-25** The **Go** button displayed after selecting four curves



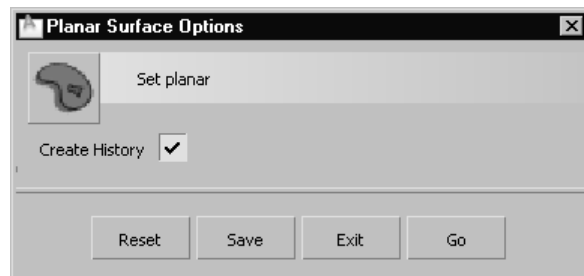
**Figure 3-26** Planar surface created



### Note

The display of CVs, hulls, and edit points in Figures 3-25 and 3-26 has been turned off for a better display of curves and surface. Also, the planar surface in Figure 3-26 has been shaded.

To set the parameters of the **Set planar** tool, double-click on the **Set planar** button; the **Planar Surface Options** dialog box will be displayed, as shown in Figure 3-27. The only option in this dialog box, the **Create History** check box, has been discussed earlier.



**Figure 3-27** The **Planar Surface Options** dialog box

## EDITING FEATURES

You can cut, copy, and paste the objects created by using the options in the **Edit** menu. These options are discussed next.

### Removing an Object

**Menu bar:** Edit > Cut

You can remove a selected object, component, or feature from the active window. To do so, first select an object and then choose **Edit > Cut** from the menu bar; the selected object will be removed from the active window. Alternatively, press CTRL+X to cut an object from the active window.

### Copying an Object

**Palette:** Edit > Copy

You can copy a selected object, component, or a feature from the active window. To do so, first select an object and then choose **Edit > Copy** from the menu bar; the selected object will be copied. Alternatively, press CTRL+C to copy an object from the active window.

### Pasting an Object

**Palette:** Edit > Paste

You can paste a copied or cut object in the active window. To do so, first cut or copy an object and then choose **Edit > Paste** from the menu bar; the copied or cut object will be pasted at the location of the original object. Alternatively, press CTRL+V to paste a copied or cut object in the active window. You can move the copied object away from the original object by using the **Move** tool.

To set the parameters of the **Paste** option, click on the box given on its right; the **Paste Options** dialog box will be displayed, as shown in Figure 3-28.

The options in this dialog box are discussed next.

#### Layers options

This area is used to specify the layers in which the copied or cut objects will be pasted. The radio buttons in this area are discussed next.

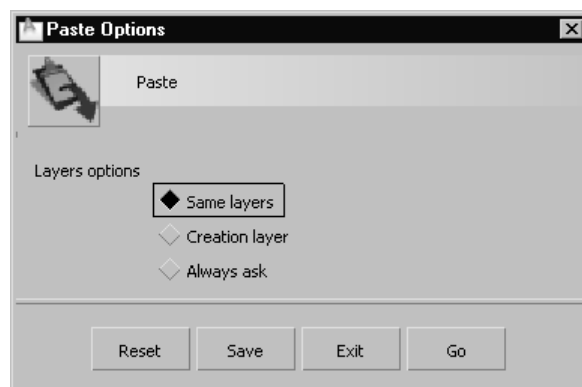


Figure 3-28 The **Paste Options** dialog box

**Same layers**

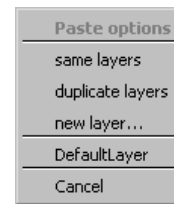
This radio button is selected by default and is used to paste the object in the same layer from where it was copied or cut.

**Creation layer**

Select this radio button to paste the object in the current layer. The current layer will be highlighted in yellow color in the layers bar.

**Always ask**

Select this radio button; the **Paste options** menu will be displayed, as shown in Figure 3-29. The options in this menu are used to specify the layer where the objects will be pasted. Choose the **same layers** option to paste the object in the layer from where it was copied or cut. Choose the **duplicate layers** option to paste the object in a duplicate layer. This layer is named identical to the existing layer. For example, if the existing layer is named as **Line**, the object will be pasted in a duplicate layer named **Line#2**. Choose the **new layer** option to paste the object in a new layer. You can specify the name of a new layer in the **Create a new layer** edit box of the **confirm** message box that will be displayed when you choose the **new layer** option from the menu. Choose the **DefaultLayer** option to paste the copied or cut object in the default layer. If there are existing layers, the menu will allow you to choose the required layer in which the copied or cut object will get pasted. You can cancel the pasting of the selected object by choosing the **Cancel** option.



*Figure 3-29 The Paste options menu*

## Reverting to the Previous Command

**Menu bar:** Edit > Undo

You can reverse the effect of the previous command by choosing **Edit > Undo** from the menu bar. Alternatively, you can undo the previous command by pressing CTRL+Z keys. Also, you can specify the number of undos in the **Miscellaneous** tab of the **General Preferences** dialog box. The **General Preferences** dialog box has been discussed earlier in Chapter 1.

## Re-applying the Previous Command

**Menu bar:** Edit > Redo

If you undo the previous command by mistake, you can re-apply the previous command. To do so, choose **Edit > Redo** from the menu bar.

## Re-invoking the Last Command

**Menu bar:** Edit > Reinvoke last

You can repeat the last command by choosing **Edit > Reinvoke last** from the menu bar. Alternatively, press ALT+I keys to repeat the last command. For example, if the last command executed was **Cut**, choose **Edit > Reinvoke last** to repeat the **Cut** command.

## EDITING KEYPOINT CURVES

In Chapter 2, you have created keypoint curves. You can move, reshape, or edit the attributes of these keypoint curves. These editing operations are discussed next.

### Moving the Keypoints

**Palette:** Curves > Keypoint curve toolbox > Drag keypoints



To move keypoints, choose the **Drag keypoints** button from the **Keypoint Curve Toolbox**; you will be prompted to use the mouse to drag the keypoint or enter the new position of the keypoint. Select the keypoint; the keypoint will be highlighted in blue, as shown in Figure 3-30. Next, drag the cursor in the required direction; the keypoint will be moved, as shown in Figure 3-31. You can move a keypoint in the horizontal and vertical directions by dragging it with the middle and right mouse buttons, respectively.

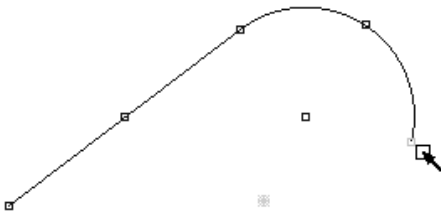


Figure 3-30 Selected keypoint

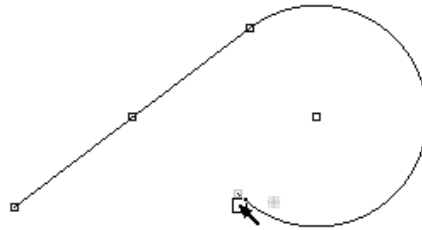


Figure 3-31 New position of the keypoint

### Breaking the Keypoint Curves

**Palette:** Curves > Keypoint curve toolbox > Break curve at keypoint



You can detach keypoint curve at keypoints. To do so, choose the **Break curve at keypoint** button from the **Keypoint Curve Toolbox**; you will be prompted to select keypoints to break curve at. Select the keypoint where you want to break the curve; the curve will be broken at that point. The broken curve will act as a separate object.

### Joining the Keypoint Curves

**Palette:** Curves > Keypoint curve toolbox > Break curve at keypoint > Join curves



You can attach different keypoint curves to form a single curve by using the **Join curves** tool. This tool can also be used to attach regular curves by joining their edit points. To attach curves, choose the **Join curves** button from the **Keypoint Curve**



**Toolbox;** you will be prompted to select the keypoints or edit points to join at. Select keypoints or edit points; the curves will join to form a single curve. Note that the keypoints of two curves to be joined need to be coincident. The joined curves will act as a single object.

## Modifying the Attributes of a Keypoint Curve

**Menu bar:** Windows > Information > Information window

You can edit the attributes of a keypoint curve such as name, length, radius, sweep angle, and so on. To do so, select the keypoint curve from the active window and then choose **Windows > Information > Information window** from the menu bar; the **Information Window** dialog box will be displayed, as shown in Figure 3-32. Alternatively, select the keypoint curve and then press CTRL+5 keys to display the **Information Window** dialog box. To change the name of the curve, enter a new name in the **Name** edit box. Next, select the **Bounding Box** check box to display the bounding box of the curve. Select the **Invisible** check box to hide the curve in the active window. To change the attributes of the curve, click in the **Attributes** area; different edit boxes, depending upon the type of the keypoint, will be displayed. You can change the length, radius, sweep angle of the keypoint curve by entering new values in the respective edit boxes. If the keypoint curve is an arc, you can even change this arc to its complement by choosing the **Arc Complement** button in the **Attributes** area.

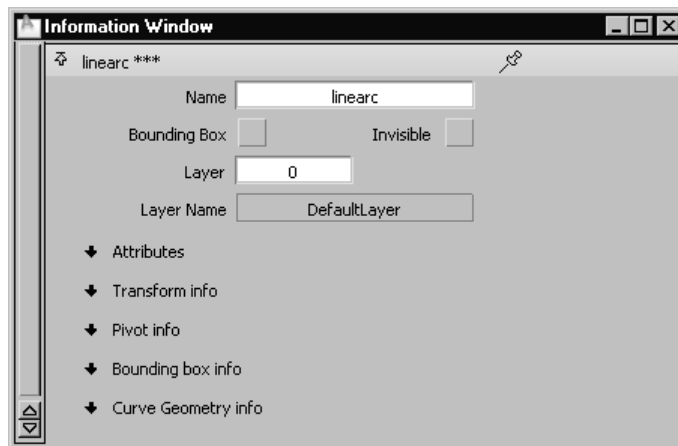


Figure 3-32 The **Information Window** dialog box



**Tip:** The **Information Window** dialog box can also be used to view the pivot point, bounding box, layer, and surface geometry information. This dialog box has been discussed in detail in Chapter 1.

# CONSTRUCTION OBJECTS

Construction objects act as a reference for creating features of a model. In AliasStudio, construction objects used are point, vector, plane, and grid. The creation of these objects are discussed next.

## Placing a Point

**Palette:** Construction > Point



You can create reference points in the view windows by using the **Point** tool. The reference points act as an input to other tools, and are also used as snapping targets and place-holders. To create a reference point, choose the **Point** button from the **Construction** tab in the **Palette**; you will be prompted to select the object or enter the position of the point. If you select a curve as a reference for positioning the point, the point will be defined by three parameters, N1, N2, and du, as shown in Figure 3-33. If you select the boundary edge of the surface as a reference for positioning the point, the point will be defined by three parameters, N, du, and dv, as shown in Figure 3-34. You can also specify the position of the point by entering coordinates in the promptline or by clicking in the active window.

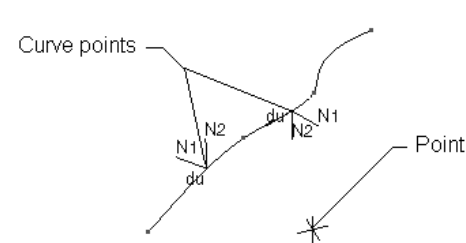


Figure 3-33 Points on a curve

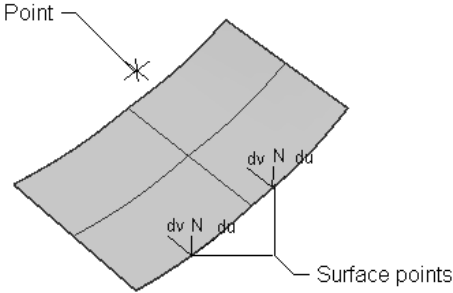


Figure 3-34 Points on a surface

To place a point with its name in the view windows, double-click on the **Point** button; the **Construction Point Options** dialog box will be displayed, as shown in Figure 3-35. Select the **Show Name** check box to display the name of the point in the view windows.

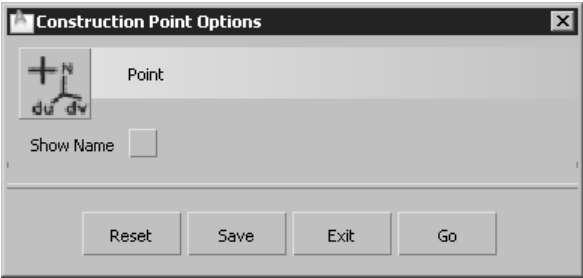


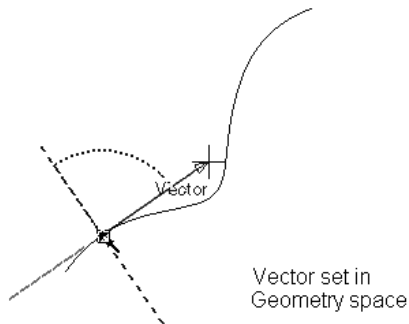
Figure 3-35 The Construction Point Options dialog box

## Creating a Vector

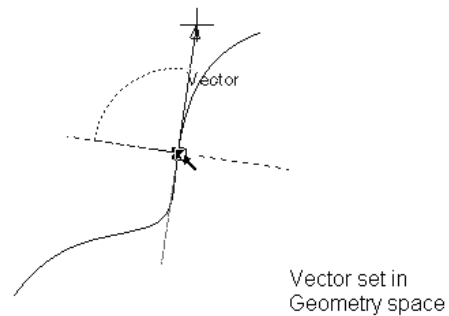
**Palette:** Construction > Vector



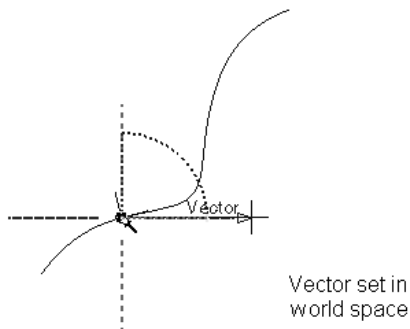
You can create a reference vector by using the **Vector** tool. Reference vector is used as an input for tools that require starting point and directions such as **Project**, **multi-surface Draft**, and **Draft (or Flange)**. To create a vector, choose the **Vector** button from the **Construction** tab in the **Palette**; you will be prompted to specify the position of the origin of the vector. Click in the active window or enter coordinates in the promptline; the manipulator will be displayed and you will be prompted to adjust the manipulator or enter the end position of the vector. Also, the **Next Vector** button will be displayed at the lower right corner of the active window. Choose the **Next Vector** button; the vector with the default setting will be created. You can change the position and length of the vector by dragging the arrow head manipulator to the required position before choosing the **Next Vector** button. If you select an existing object from the window to specify the vector's origin; the vector will be set in geometry space, as shown in Figure 3-36. Also, the **XYZ** and **Next Vector** buttons will be displayed at the lower right corner of the active window. If you move the origin of the vector along the geometry, its orientation will get changed with respect to the change in geometry, as shown in Figure 3-37. Choose the **XYZ** button to set the vector in world space, as shown in Figure 3-38. If you move the origin of the vector along the geometry, the vector will not change its orientation with respect to the change in geometry, as shown in Figure 3-39. You can switch back to geometry space by choosing the **GEOM** button.



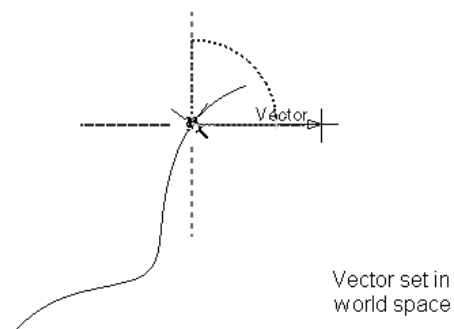
**Figure 3-36** Vector set in geometry space



**Figure 3-37** Changed orientation of the vector



**Figure 3-38** Vector set in world space



**Figure 3-39** Unchanged orientation of the vector

To create a vector with its name in the view windows, double-click on the **Vector** button; the **Construction Vector Options** dialog box will be displayed, as shown in Figure 3-40. Select the **Show Name** check box to display the name of the vector in the view windows.

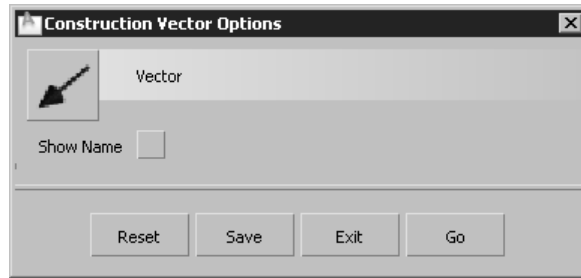


Figure 3-40 The Construction Vector Options dialog box

## Creating a Reference Plane

**Palette:** Construction > Plane



A construction plane is used as a reference for creating advanced surfaces. The construction plane is also used as an input for other tools such as **Curve planarize**, **Section a group of curves**, and so on. You can create a reference/construction plane by using the **Plane** tool. This reference plane can also be used as the construction plane. To create a reference plane, choose the **Plane** button from the **Construction** tab in the **Palette**; you will be prompted to select the plane construction method. Also, five buttons, **View**, **Slice**, **3 Pt**, **Geom**, and **World** will be displayed at the lower right corner of the active window. Choose the **World** button; you will be prompted to specify the center point or the geometry point of the plane. Click in the active window or enter coordinates in the promptline; the reference plane will be placed at the specified position. Also, the **Next Plane** and **Set Construction Plane** buttons will be displayed in the active window and you will be prompted to move the plane or move the point. Next, choose the **Next Plane** button to create the construction plane, as shown in Figure 3-41. If you choose the **Set Construction Plane** button, the reference plane will be set as the construction plane. This construction plane can be set to an arbitrary coordinate system that can be oriented with respect to the world coordinate system. You can create a plane in different ways by choosing the five buttons that have been discussed above. These buttons are discussed next.

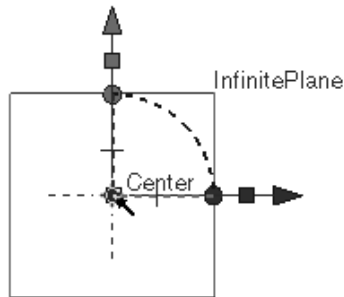


Figure 3-41 Construction plane

### View

Choose this button to create a plane by specifying its center point. The Z-axis of the plane will be parallel to the view vector. When you choose this button; you will be prompted to specify the center point or the geometry point. Specify the position of the center point in the promptline or click in the active window; the plane with the manipulators will be displayed in

the active window, as shown in Figure 3-42. You can also modify the size, angle, or position of the plane by using different manipulator handles that will be discussed later in this chapter. Choose the **Next Plane** button; the plane will be created, as shown in Figure 3-43.



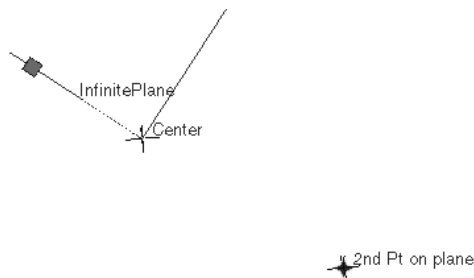
**Figure 3-42** Plane with manipulators



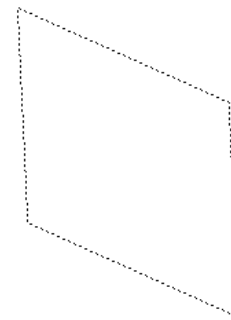
**Figure 3-43** Reference plane created

### Slice

Choose this button to create a plane by specifying two points. When you choose this button, you will be prompted to specify the center point or the geometry point of the plane. Specify the position in the promptline or click in the active window at the required point; the plane will be created in the active window and you will be prompted to specify the second point in the plane. Specify the second point; the plane passing through the center point and second point will be created. This plane will be perpendicular to the view window. Figure 3-44 shows two points specified for creating the plane and Figure 3-45 shows the plane passing through these points.



**Figure 3-44** Two points specified for plane

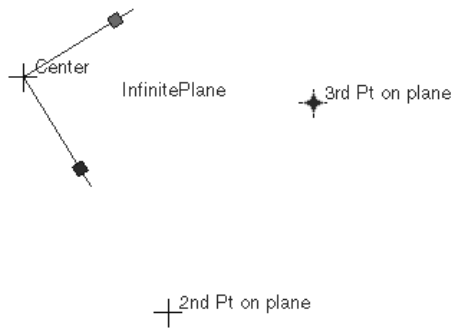


**Figure 3-45** Plane passing through two points

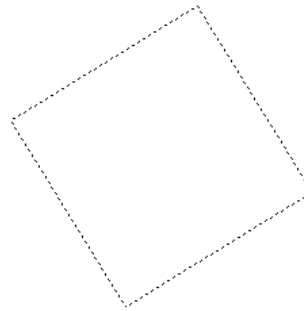
### 3 Pt

Choose this button to create a plane by specifying three points. On choosing this button, you will be prompted to specify the center point or the geometry point of the plane. Specify the position in the promptline or click in the active window at the required point; the plane will be created in the active window and you will be prompted to specify the second point in the plane. Specify the second point by clicking in the active window or entering coordinates in

the promptline; you will be prompted to specify the third point in the plane. Specify the third point; a plane passing through the specified points will be created. Figure 3-46 shows three points specified for creating the plane and Figure 3-47 shows the plane passing through these points.



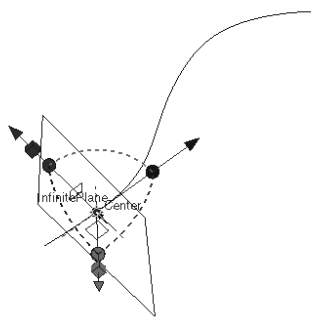
**Figure 3-46** Three points specified for creating the plane



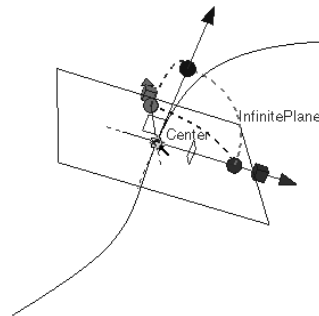
**Figure 3-47** Plane passing through three points

## Geom

Choose this button to create a plane on a geometry such that its Z-axis is oriented along the surface normal or the curve tangent. On choosing this button, you will be prompted to specify the center point or the geometry point of the plane. Click on the curve or surface at the required position; the plane will get snapped to the clicked object with the Z-axis oriented along the surface normal or the curve tangent. To view the changing orientation of the plane, move the plane along the curve or across the surface by using the manipulator. Figure 3-48 shows the plane snapped to a curve and Figure 3-49 shows the changed orientation of the plane after moving it to other location along the curve.



**Figure 3-48** Plane snapped to a curve



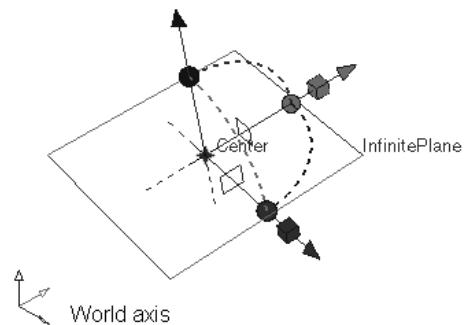
**Figure 3-49** Changed orientation of the plane

## World

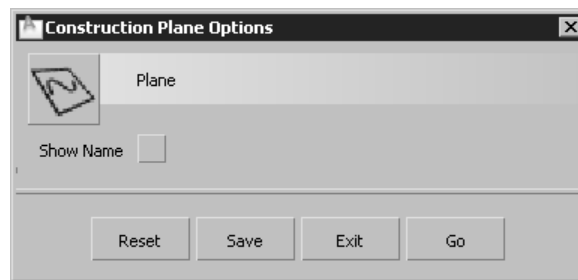
Choose this button to create a plane which is oriented along the world axes, as shown in Figure 3-50. On choosing this button, you will be prompted to specify the center point or

the geometry point of the plane. Specify the center point or the geometry point by clicking in the active window or by entering coordinates in the promptline; the plane with its axes oriented along the world axes will be created in the active window.

To create a construction plane with its name displayed in the view windows, double-click on the **Plane** button; the **Construction Plane Options** dialog box will be displayed, as shown in Figure 3-51. Select the **Show Name** check box to display the name of the construction plane in the view windows.



**Figure 3-50** Plane oriented along the world axes



**Figure 3-51** The Construction Plane Options dialog box

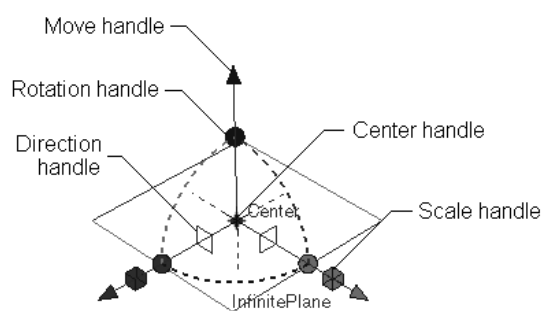
You can edit a plane with the help of a plane manipulator. Different handles of a plane manipulator are shown in Figure 3-52. The significance of these handles is discussed next.

#### Move handle

The Move handles (arrows) are visible on all three axes. Click and drag these handles to move the plane along their respective axes. You can also enter the translation amount in the promptline.

#### Scale handle

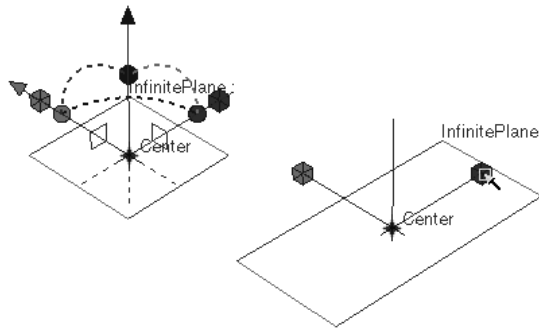
The Scale handles (cubes) are visible on two axes that lie on the plane. Click and drag one of the handles to scale the plane along it. You can also enter the scale factors in the promptline. Figure 3-53 shows the construction plane scaled along the X-axis.



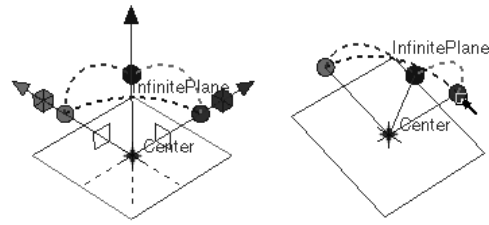
**Figure 3-52** Different handles of the plane manipulator

### Rotation handle

The Rotation handles (spheres) are visible on all three axes. Click and drag these handles on one of the axes to rotate the plane about it. You can also enter the rotation angles in the promptline to rotate the plane. Figure 3-54 shows the construction plane rotated about X-axis.



**Figure 3-53** Construction plane scaled along X-axis



**Figure 3-54** Construction plane rotated about X-axis

### Direction handle

The Direction handles (rectangular boxes) are visible on two axes that lie on the plane. Click one of the handles to flip the direction of the plane. Click on the dotted line that lie opposite to the axis to reflect the plane.

### Center handle

The Center handle is used as the pivot point for moving, rotating, scaling, and flipping the plane. It also acts as the origin of the construction plane. You can drag this handle to change the position of the plane.

## Setting the Construction Plane

**Palette:** Curve Edit > Set construction plane



The **Set construction plane** tool is used to set the reference plane as the construction plane. To do so, choose the **Set construction plane** button from the **Construction** tab in the **Palette**; you will be prompted to select a construction plane. Select a plane to set it as the construction plane. At a time, only one plane can be set as the construction plane. Rest of the planes act as the reference planes. You can set any of the reference planes as the construction plane by using this tool. Note that when you set a reference plane as the construction plane, the axis triad changes accordingly. Also, the name of the construction plane will be displayed at the lower left corner of all view windows in the interface.



### Note

The **Plane** tool allows you to set the reference plane as the construction plane without using the **Set construction plane** tool. To do so, choose the **Set Construction Plane** button that is displayed when you specify the center point or the geometry position of the plane while using the **Plane** tool.



## Toggling between the Planes

**Palette:** Curve Edit > Toggle construction plane



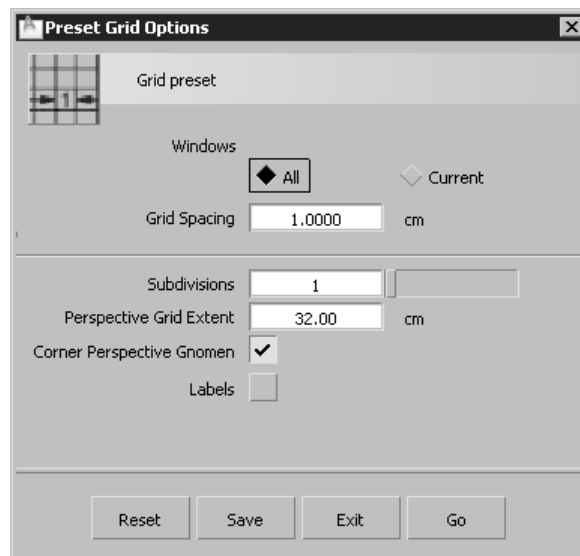
You can toggle between the reference plane that is set as the construction plane and the plane defined by world space (XYZ) axes by using the **Toggle construction plane** tool. You need to set the reference plane as the construction plane before using this tool. Toggling between the construction planes changes the axis triad accordingly.

## Presetting the Grid

**Palette:** Curve Edit > Grid preset



Grid is a network of uniformly spaced lines, superimposed on the screen. It is used as a reference for sketching, placing, snapping objects, and so on. You can turn the grid display on or off by choosing **WindowDisplay > Toggles > Grid**. The grid lines are spaced horizontally and vertically with some spacing between them. You can change this spacing by using the **Grid preset** tool. To set the parameters of the **Grid preset** tool, double-click on the **Grid preset** button; the **Preset Grid Options** dialog box will be displayed, as shown in Figure 3-55.



*Figure 3-55 The Preset Grid Options dialog box*

The options in this dialog box are discussed next.

### Windows

The **Windows** area is used to specify the window in which the grid spacing will be changed. The radio buttons in this area are discussed next.

**All**

This radio button is selected by default and is used to change the grid parameters in all windows of the interface screen.

**Current**

Select this radio button to change the grid spacing in the current or active window.

**Grid Spacing**

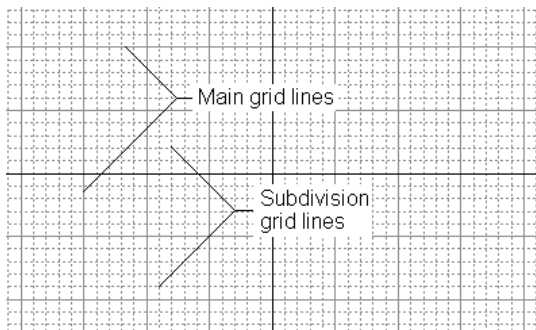
This edit box is used to specify the spacing or distance between grid lines in the current linear units. By default, the unit of grid spacing is centimeter. To change the unit spacing of grid, choose **Preferences > Construction Options** from the menu bar; the **Construction Options** dialog box will be displayed. Next, choose **Units > Linear** from the dialog box to display options under it. Click on the **cm** button given on the right of the **Main Units** area; a flyout showing different units will be displayed. Choose the required unit from the flyout and then close the **Construction Options** dialog box.

**Subdivisions**

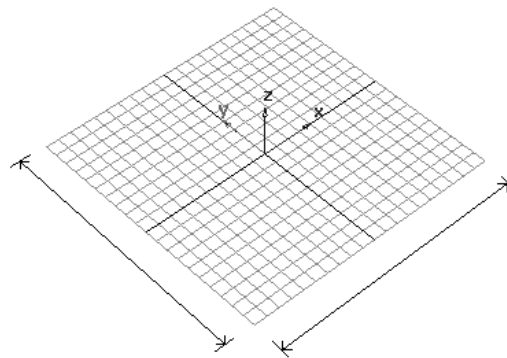
This edit box is used to specify the number of subdivision lines between the two main grid lines. The subdivision lines appear light and dotted unlike the main grid lines that appear continuous and dark. The subdivision lines are also uniformly spaced with respect to each other. You can also change the distance between the subdivision lines by using the slider bar given on the right of the **Subdivisions** edit box. Figure 3-56 shows the main grid lines and subdivision grid lines.

**Perspective Grid Extent**

The **Perspective Grid Extent** edit box is used to specify the size of grid in the **Perspective** window in linear units. By default, the grid size of 32 cm is displayed in the **Perspective Grid Extent** edit box. You can change this value by entering a new value in this edit box or by using the slider bar given on the right of this edit box. Figure 3-57 shows the perspective window with the perspective grid extent of 10 units.



**Figure 3-56** Main grid lines and subdivision grid lines



**Figure 3-57** The **Perspective** window with perspective grid extent of 10 units

**Note**

When you change the units using the **Construction Options** dialog box, the default values of the respective units also change. If the unit is cm and the default value of the **Perspective Grid Extent** edit box is 32, then after changing the units to mm, the default value will be 320.

**Corner Perspective Gnomon**

Gnomon is an indicator that is placed at the lower left corner of the view window. This is significant as it displays the orientation of three mutually perpendicular colored axes. The **Corner Perspective Gnomon** check box is selected by default and displays the gnomon in the **Perspective** window. Clear this check box to hide the gnomon from the **Perspective** window. The grey arrow in this indicator points toward the origin.

**Labels**

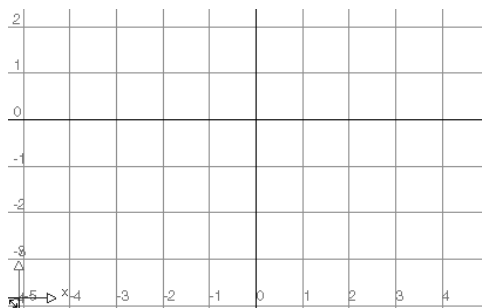
Select this check box to label the main grid lines and subdivision grid lines with their corresponding unit values. These labels appear only in the orthographic windows.

**Label Font Properties**

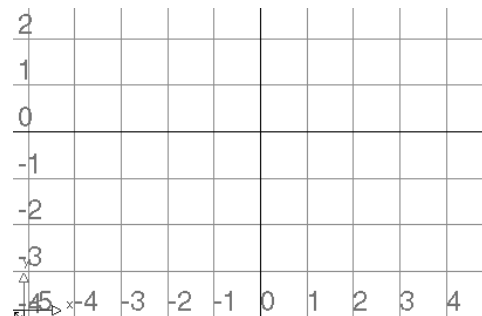
This area will be available only when you select the **Labels** check box. This area is used to specify the size of the font of label. By default, the **Default** button is chosen from this area. As a result, the labels with the default font properties are displayed, as shown in Figure 3-58. However, you can change the default settings as well. To do so, click on the **Default** button, a flyout will be displayed. You can choose the **Custom** button from this flyout to change the font size.

**Label Font Size**

This area will be available only when you choose the **Custom** button from the **Label Font Properties** area. By default, the **10** button is chosen from this area and is used to display the labels with the font size of 10. Click on the **8** button to display the flyout and choose the required label font size from this flyout; the label with the chosen label font size will be displayed in the orthographic windows. Figure 3-59 shows the grid label with label font size of 18.



**Figure 3-58** Grid label with default label font size



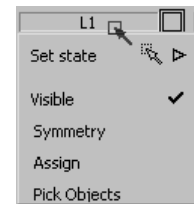
**Figure 3-59** Grid label with label font size of 18

## WORKING WITH LAYERS

Like other software packages, AliasStudio allows you to organize objects. By organizing objects, you can control the selection, display, symmetry, and so on of the objects. These actions can be performed by using the layers that are displayed in the Layers bar. By default, a layer named DefaultLayer will be displayed in the Layers bar. To create a new layer, choose **Layers > New** from the menu bar; a new layer L1 will be displayed in the Layers bar. You can rename the layer by double-clicking on the layer and then entering a new name in it. Different operations that can be performed on objects by using layers are discussed next.

### Assigning the Objects to Layers

You can assign objects to layers. To do so, select the required objects and keep the left mouse button pressed on the layer in the Layers bar; a flyout will be displayed, as shown in Figure 3-60. Choose the **Assign** option from this flyout; the objects will be assigned to the layer.



**Figure 3-60** The layer flyout

### Selecting the Objects

To select objects assigned to a layer, keep the left mouse button pressed on the layer in the Layers bar to invoke the flyout associated with the layer. Next, choose the **Pick Objects** option from the flyout; the objects assigned to the layer will be picked. You can also control the selection of objects in the selected layer by using the options in the Layers menu. To select objects in the selected layer, choose **Layers > Select > Objects on selected layers** from the menu bar. To pick layers with the selected object assigned to it, choose **Layers > Select > Layers by picked objects** from the menu bar. To select all layers in the Layers bar, choose **Layers > Select > All layers** from the menu bar. To select the layer within a range, choose **Layers > Select > Layer range** from the menu bar and then enter the range in the promptline.

### Controlling the Visibility of Objects

You can control the display of objects in the layers. By default, objects assigned to a layer will be visible. To turn off the visibility of objects in a layer, press and hold the left mouse button on the layer; a flyout will be displayed. Choose the **Visible** option from the flyout; the visibility of objects in the layer will be turned off. You can also control the objects in the selected layer by using the options in the Layers menu. To turn on the visibility of objects assigned to a selected layer, choose **Layers > Visibility > Visible** from the menu bar; and to turn off the visibility of objects assigned to a selected layer, choose **Layers > Visibility > Invisible** from the menu bar. You can also toggle between the visibility states of objects in a layer by clicking the box given on the right of the layer in the Layers bar.

### Setting the Symmetry of Objects

You can set the symmetry of objects in the layer. By setting the symmetry of objects in the layer, you can display the symmetric (mirror) copy of objects in the view window. By default, the symmetry of objects will be turned off. To set the symmetry of objects in a layer, press and hold the left mouse button on the layer; a flyout will be displayed. Choose the **Symmetry** option from the flyout; the symmetry of objects will be set. You can also set the symmetry of objects in the selected layers with the Layers menu. To do so, choose **Layers > Symmetry**

> **On** from the menu bar. To turn off the symmetry of objects in the selected layers, choose **Layers > Symmetry > Off** from the menu bar. To convert the symmetric objects to real geometry, choose **Layers > Symmetry > Create geometry** from the menu bar.

## Deleting the Layers

To delete the selected layers, choose **Layers > Delete > Selected layers** from the menu bar. To delete the unused layers, choose **Layers > Delete > Unused layers** from the menu bar. Note that the **Delete** command will delete only the layer, not objects.

## Controlling the State of Objects

You can control the state of objects in the selected layers. To set the state of the selected layer to pickable, choose **Layers > Set state > Pickable** from the menu bar. The pickable layer and objects assigned to this layer will be displayed in grey. You can pick an object assigned to the pickable layer. To set the state of the selected layer to reference, choose **Layers > Set state > Reference** from the menu bar. The reference layer and objects assigned to this layer will be displayed in light brown. You can snap to objects assigned to the reference layer without picking up them. To set the state of the selected layer to inactive, choose **Layers > Set state > Inactive** from the menu bar. The inactive layer and objects assigned to this layer will be displayed in light blue and also objects assigned to the inactive layer will be visible. However, you cannot pick or snap to objects assigned to the inactive layer.

## Applying the Colors to Layers

You can apply colors to objects assigned to the selected layers. To do so, press and hold the left mouse button on the square box on the right of the layer; a flyout with different color swatches will be displayed, as shown in Figure 3-61. Select the required color swatch to apply color to the object. To edit the color of the objects, choose the **Edit** button from this flyout; the **Layer Color** editor will be displayed, refer to Figure 3-61. Select the required color from this editor to assign the color to the object.



Figure 3-61 The Layer Color editor

## Toggling between the Visibility States of Layers

You can toggle between the visibility states of layers to toggle the visibility of the entities assigned to them. By default, the visibility of layers is turned on. To turn the visibility of layers off, choose **Layers > Toggle layers** from the menu bar; all layers will be disabled. Also, the visibility of layers will be turned off. To turn the visibility of Layers bar off, choose **Layers > Toggle Layers Bar** from the menu bar; the visibility of all layers will be turned off. However, you can access layers by using the **Object Lister** window. You can also toggle between the visibility states of unused layers by choosing **Layers > Toggle Unused Layers** from the menu bar.

## CONTROLLING THE DISPLAY OF OBJECTS

You can control the display of control vertices, edit points, hulls, and so on, on all or active objects. To do so, choose the **ObjectDisplay** option from the menu bar; a flyout will be displayed. Click on the box given on the right of the **Control** option; the **DisplayControl** options box will be displayed, as shown in Figure 3-62. The options in the **DisplayControl** options box are discussed next.

### Scope

This area is used to specify the objects whose display will be affected by using the **DisplayControl** options box. You can control the display of active objects, entire model, new curves, and so on by choosing the respective button from this area.

### Type

This area will be displayed only when you choose the **NEW CRV** button from the **Scope** area. The options in this area are used to specify the type of curve to be affected by using the **DisplayControl** option box. You can control the display of a free curve, blend curve, or keypoint curve by choosing the corresponding button.

### All

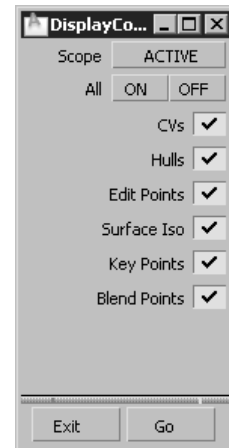
The **All** area is used to automatically turn on or off the check boxes in this area. By default, all check boxes are selected, thereby ensuring that the display of all components of an object is turned on in the view window. Choose the **OFF** button from this area to clear all check boxes automatically such that the display of the corresponding options in the view windows is turned off. You can turn the display of features on or off by selecting or clearing the corresponding check boxes manually.

### Exit

Choose this button to close the **DisplayControl** options box.

### Go

Choose this button to update changes in the display of objects in the view windows.



*Figure 3-62 The DisplayControl options box*

## CONTROLLING THE VISIBILITY OF OBJECTS

You can turn the visibility off or on of objects in the view windows. The ways to control the visibility of objects are discussed next.

### Turning off the Visibility of Objects

To turn off the visibility of objects, select the objects from the view window; the selected objects will be highlighted. Next, choose **ObjectDisplay > Invisible** from the menu bar; the visibility of the selected objects will be turned off. In other words, the objects will be invisible.

## Turning on the Visibility of Objects

To turn on the visibility of hidden objects, choose **ObjectDisplay > Visibility** from the menu bar; the hidden or invisible objects will be displayed in the view window. You can control the visibility of various features associated with the object. To do so, click on the box on the right of the **Visibility** option; the **Visible Options** dialog box will be displayed, as shown in Figure 3-63. By default, the **All** radio button is selected from the **Scope** area of the **Visible Options** dialog box and is used to display all hidden objects in the view window. If you select the **Pick** radio button from the **Scope** area; the visibility of hidden objects selected from the **SDB** window will be turned on. This radio button is also used to turn on the visibility of the objects whose name is entered in the promptline.



Figure 3-63 The *Visible Options* dialog box

## Turning Objects into Templates

You can turn an object into template when you do not want them to be disturbed or affected while modifying other objects in the view window. Templates are also used as the reference objects or construction objects. To turn objects into templates, select the required objects from the window; the selected objects will be highlighted. Next, choose **ObjectDisplay > Template** from the menu bar; the objects will be turned into templates. The templates in the inactive state will be displayed in grey color, whereas while as the selected templates are displayed in pink color.

## Changing the Appearance of Data

You can change the appearance of surfaces, curves, and section data. To do so, choose the **ObjectDisplay** option from the menu bar; a flyout will be displayed. Click on the box on the right of the **Draw style** option in the flyout; the **DrawStyle** options box will be displayed, as shown in Figure 3-64. The areas in this option box are discussed next.

### Surface

This area is used to modify the appearance of features related to a surface. You can change a single surface to appear as double surface and vice versa. You can also change the icons of control vertices of the surface by choosing the corresponding button from the **CV Icon** area.

## Curves

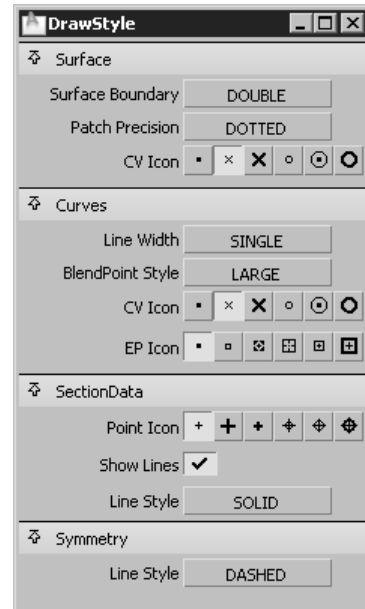
This area is used to modify the appearance of features related to a curve. You can change the line width and blend point style of the curve. Also, you can change the icons of control vertices and edit points of the curve by choosing the required buttons from the corresponding area.

## SectionData

This area is used to modify the appearance of the features related to section curves. You can change the line style, section point icon, and so on by choosing the required buttons from the corresponding areas.

## Symmetry

This area is used to specify the appearance of objects that are set symmetric. To do so, choosing **Layers > Symmetry > On** from the menu bar. You can change the symmetric object to dashed or solid object by choosing the corresponding option from the **Line Style** area.



**Figure 3-64** The *DrawStyle* options box

# TUTORIALS

## Tutorial 1

In this tutorial, you will create the model shown in Figure 3-65. After creating the model, you will shade it to give it a realistic look. **(Expected time: 45 min)**



**Figure 3-65** Model for Tutorial 1

The following steps are required to complete this tutorial:

- Start AliasStudio and then start a new wire file.
- Create the base feature of the model by using the **Torus** tool.



- c. Scale the base feature nonuniformly by using the **Nonproportional scale** tool.
- d. Create the profile and path curves for the extrude feature by using the **Circle** and **Arc (two point)** tools.
- e. Create the extrude feature by using the **Extrude** tool.
- f. Copy and rotate the extrude feature by using the **Edit** menu option and the **Rotate** tool, respectively.
- g. Create the handle of the model by using the **Extrude** tool.
- h. Create the cap of the handle by using the **Cylinder** tool.
- i. Create the net by using the **Line** tool.
- j. Turn off the display of the extrude curves.
- k. Shade the model by using diagnostic shading.
- l. Save the model and exit.

### Starting a New Wire File


1. Choose **Start > Programs > Autodesk > AliasStudio 2009 > AliasStudio 2009** from the taskbar; the Autodesk Studio window informing about the start of the software is displayed.

As AliasStudio 2009 gets started, a Studio screen is displayed. You need to start a new wire file for creating the model.

2. Choose **File > New** from the menu bar; a new AliasStudio wire file gets started and four windows are displayed on the screen.


### Creating the Base Feature

The base feature is created by using the **Torus** tool. Before placing the torus in the active window, you need to set the parameters of the torus by using the **New Torus Options** dialog box.

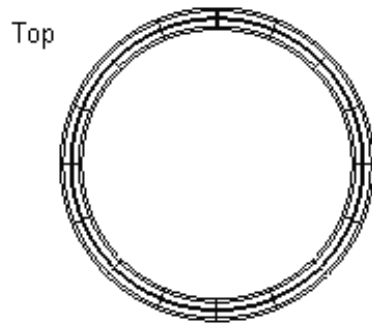
1. Choose the Maximize box located at the upper right corner of the top window; the top window expands to fill the entire interface screen. You can also resize the **Top** window by choosing **Layouts > Top** from the menu bar or by pressing the F5 key.
2. Double-click on the **Torus** button from the **Surfaces** tab in the **Palette**; the **New Torus Options** dialog box is displayed. 
3. Enter **2** and **0.15** in the **Major radius** and **Minor radius** edit boxes, respectively, and choose the **Go** button; you are prompted to specify the position of the torus.
4. Enter **0** in the promptline; the base feature is created, as shown in Figure 3-66.

### Scaling the Base Feature

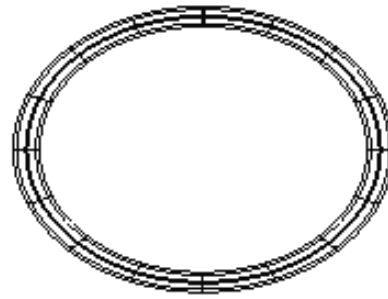
The base feature created may not be of the required shape and size. So, you need to scale it by using the transform tools.

1. Choose the **Nonproportional scale** button from the **Transform** tab in the **Palette**; you are prompted to enter the scale factors along the X, Y, and Z-axes. 

2. Enter **1, 0.75, and 1** separated by space in the promptline and press ENTER; the base surface is scaled, as shown in Figure 3-67. Alternatively, drag the left mouse button downward to scale the base surface nonproportionally. In this case, you can scale it arbitrarily.







**Figure 3-66** Base feature

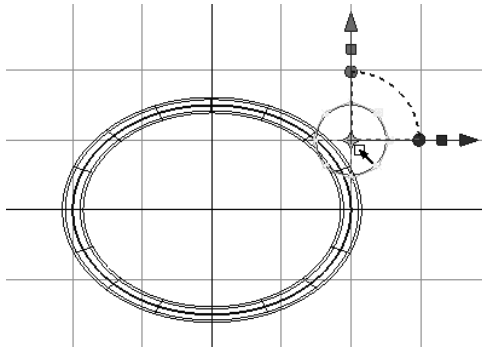


**Figure 3-67** Base feature after scaling nonproportionally

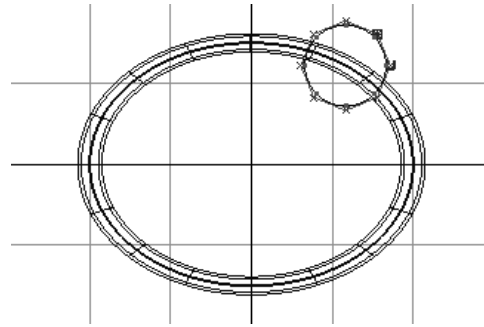
### Creating the Profile and Path Curves for the Extrude Feature

Next, you need to create the profile curve and path curve for the extrude feature at the top of the base feature by using the **Circle** and **Arc (two point)** tools.

1. Choose the **Circle** button from the **Curves** tab in the **Palette**; you are prompted to specify the position of the circle. 
2. Turn on the display of the grid by choosing **WindowDisplay > Toggles > Grid** from the menu bar.
3. Press the ALT key and click at the grid; the circle is created on the plane on which the base feature is placed, as shown in Figure 3-68.
4. Next, choose the **Move** button from the **Transform** tab in the **Palette**; you are prompted to specify the new position of the circle. 
5. Drag the cursor to move the circle to the desired position on the base feature, as shown in Figure 3-69.
6. Choose the **Rotate** button from the **Transform** tab in the **Palette**; you are prompted to enter the rotation angles about the X, Y, and Z-axes. 
7. Enter **90, 90, and 60** in the prompt line and press ENTER; the circle is rotated as specified about the X, Y, and Z-axes, as shown in Figure 3-70.
8. Choose the **Scale** button from the **Transform** tab in the **Palette**; you are prompted to enter the scale factor. 

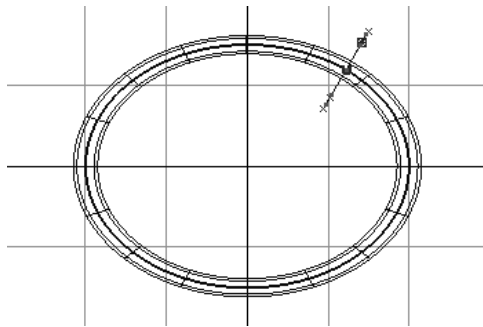


**Figure 3-68** Circle created as the generation curve

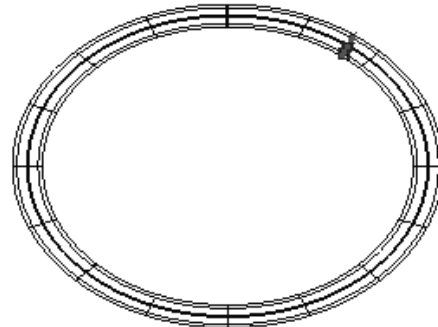


**Figure 3-69** Circle moved on the base feature curve

9. Enter **0.2** in the prompt line and press ENTER; the circle is scaled uniformly along the X, Y, and Z-axes.
10. Choose **WindowDisplay > Toggles > Grid** from the menu bar; the grid display is turned off. The scaled circle after turning off the grid display is shown in Figure 3-71.



**Figure 3-70** Circle rotated



**Figure 3-71** Scaled circle after turning off the grid display

11. Next, draw the arc by using the **Arc (two point)** tool, as shown in Figure 3-72.

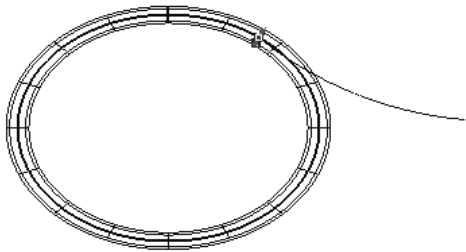
### Creating the Extrude Feature

The extrude feature can be created by using the circle and the arc as the profile and the path curves, respectively.

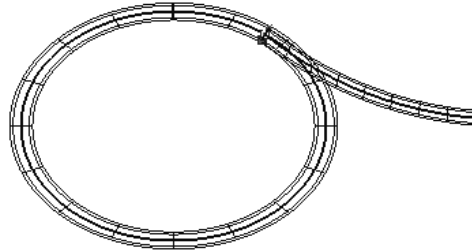
1. Press and hold the left mouse button on the **Rail surface** button from the **Surfaces** tab in the **Palette**; a flyout is displayed.
2. Choose the **Extrude** button from this flyout; you are prompted to select the curves to extrude.



3. Select the circle from the window; you are prompted to select or deselect curves to undo the selection of the first curve. Also, the **Go** button is displayed at the lower right corner of the window.
4. Choose the **Go** button from the window; you are prompted to select the path curve.
5. Select the arc from the window; the extrude feature is created, as shown in Figure 3-73.




*Figure 3-72 Arc created by using the **Arc (two point)** tool*



*Figure 3-73 Extruded feature*


### Copying and Rotating the Extrude Feature

Next, you need to copy the extrude feature and then rotate it to place it exactly opposite to the original extrude feature at the bottom of the base surface.

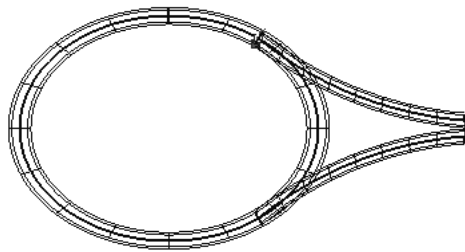
1. Select the extrude feature and then choose **Edit > Copy** from the menu bar; the extrude feature is copied.
2. Next, choose **Edit > Paste** from the menu bar; the extrude feature is pasted in the window and placed exactly on the original extrude feature.
3. Choose the **Rotate** button from the **Transform** tab in the **Palette**; you are prompted to enter the rotation angle about the X, Y, and Z-axes. 
4. Enter **180, 0**, and **0** in the promptline and press ENTER; the copied feature is rotated about the X, Y, and Z-axes, as shown in Figure 3-74.

### Creating the Handle of the Model

You will create the handle of the model by using the **Extrude** tool.

1. Choose **Layouts > Left** from the menu bar or press the F6 key. The **Left** window is expanded to fill the entire interface screen.
2. Next, choose the **Circle** button from the **Curves** tab in the **Palette**; you are prompted to specify the position of the circle. 

3. Click near the ends of two extrude surface features; the circle is placed at the specified position, as shown in Figure 3-75.





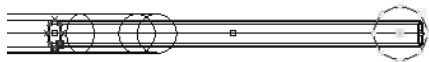
**Figure 3-74** The copied feature after rotation

Left



**Figure 3-75** Circle created near the ends of the extrude surface features

4. Choose the **Scale** button from the **Transform** tab in the **Palette**; you are prompted to enter the scale factor. 
5. Enter **0.4** in the promptline and press ENTER; the sphere is scaled uniformly along the X, Y, and Z-axes, as shown in Figure 3-76.
6. Choose the **Rotate** button from the **Transform** tab in the **Palette**; you are prompted to enter the rotation angle about the X, Y, and Z-axes. 
7. Enter **0, 0, and 90** in the prompt line and press ENTER; the circle is rotated about the X, Y, and Z-axes accordingly, as shown in Figure 3-77.




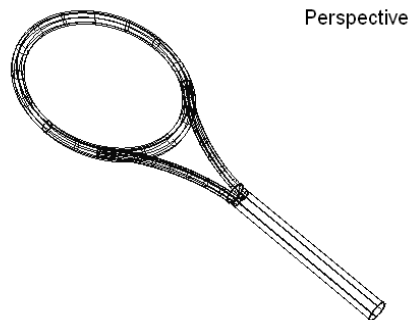
**Figure 3-76** Circle after scaling uniformly



**Figure 3-77** The rotated circle and the line

8. Choose the **Pick nothing** button from the **Pick** tab in the **Palette** to exit the **Rotate** tool.
9. Next, choose the **Line** button and create the line, refer to Figure 3-77.



10. Double-click on the **Extrude** button from the **Surfaces** tab in the **Palette**; the **Extrude Options** dialog box is displayed. 
11. Select the **Cap Both** radio button from the **Create Caps** area of this dialog box and then choose the **Go** button; you are prompted to select the curve(s) to extrude.
12. Select the circle from the window; the **Go** button is displayed on the lower right corner of the active window.
13. Choose the **Go** button; you are prompted to select the extrude path.
14. Select the line; the handle is created, as shown in Figure 3-78.



*Figure 3-78 Handle created by extrusion*

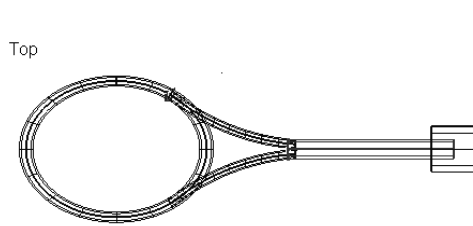
### Creating the Cap of the Handle

The cap of the handle is created by using the **Cylinder** tool. You can also scale the cap to the required size by using the **Nonproportional scale** tool.

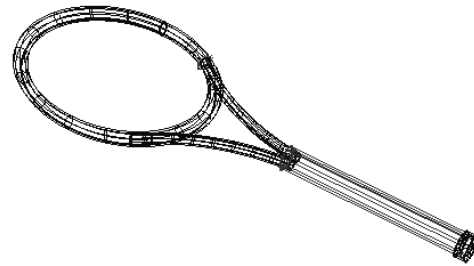
1. Choose the **Cylinder** button from the **Surfaces** tab in the **Palette**; you are prompted to specify the position of the cylinder. 
2. Click at the end of the handle; the cylinder is created, as shown in Figure 3-79.
3. Choose the **Nonproportional scale** button from the **Transform** tab in the **Palette**; you are prompted to enter the scale factors along the X, Y, and Z-axes. 
4. Enter **0.5, 0.2, and 0.2**, separated by a space, in the promptline and press ENTER. The cylinder is scaled along the X, Y, and Z-axes accordingly, as shown in Figure 3-80.

### Creating the Net of the Model

Next, you will create the net of the model by using the **Line** tool. You will also create a set of horizontal and vertical lines.

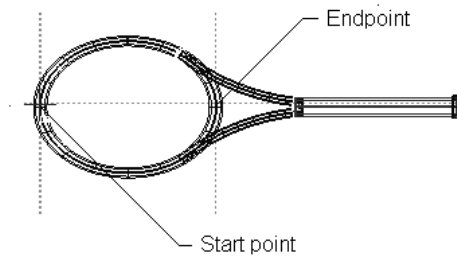


**Figure 3-79** Cylinder created for the cap of the handle

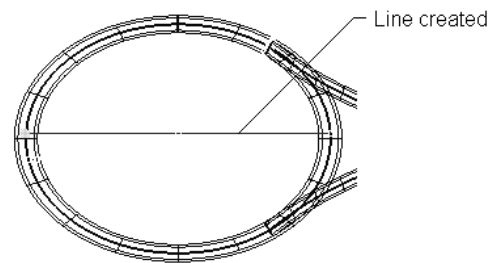


**Figure 3-80** Cap created after scaling the cylinder nonuniformly

1. Choose the **Line** button from the **Keypoint Curve Toolbox** and click on the torus to specify the start point and endpoint, as shown in Figure 3-81.
2. Choose the **Pick nothing** button from the **Pick** tab in the **Palette** to display the line created above, as shown in Figure 3-82.



**Figure 3-81** Start point and endpoint of the line



**Figure 3-82** Line created

3. Create the rest of horizontal lines by pressing the middle mouse button while the **Line** tool is still active.
4. Create vertical lines by pressing the right mouse button while the **Line** tool is still active. The model, after creating the net, is shown in Figure 3-83.



**Note**

When you draw lines with the **Line** tool, keypoints are also displayed. You need to turn the display of these keypoints off to suit to the background color. The procedure of changing the color related to the interface has been discussed in Chapter 1.



*Figure 3-83 Model after creating the net*

### Turning off the Display of Curves Used in the Extrude Feature

For clear display of the model, you need to turn off the display of the curves used in extrude feature. You can do so by using layers.

1. Choose **Layers > New** from the menu bar; a new layer L1 is displayed in the Layers bar.
2. Double-click on the layer L1; the layer text edit box is displayed.
3. Enter **Extrude Curves** in the layer text edit box and press ENTER; the layer L1 is renamed to Extrude Curves.
4. Next, select the profile curve and the extrude path used in the extrude feature; the selected curves get highlighted.
5. Press and hold the left mouse button on the Extrude Curve layer; a flyout is displayed.
6. Choose the **Assign** option from this flyout; curves are assigned to the Extrude Curves layer.
7. Press and hold the left mouse button to display the layer flyout again.
8. Choose the **Visible** option from the layer flyout; the visibility of the curves assigned to the Extrude Curves layer is turned off.

### Diagnostic Shading of the Model

The model you created will be in the wireframe mode. For a better view, you need to shade the model.

1. Choose **ObjectDisplay > Diagnostic shading** from the menu bar; the **Diagnostic shading** dialog box is displayed. You can also invoke this dialog box by choosing **Window > Control Panel** from the menu bar; the **Control Panel** is displayed along with the **Diagnostic Shading** area at the bottom of this panel.



2. Choose the **User-defined texture** button from the **Diagnostic Shading** area; the model is shaded, as shown in Figure 3-84.



*Figure 3-84 Model after diagnostic shading*

### Saving the File

To save the above model, create a new project directory, *c03\_tutorials*. The new directory will help you store all files efficiently.

1. Choose **File > Save** from the menu bar; the **Save Wire** dialog box is displayed. You can also invoke this dialog box by pressing the ALT and S keys together.
2. Choose the down arrow next to the **Project** field and select the **New Project** option from the drop-down list; the new project is displayed in the **Save in** drop-down list.
3. Browse to the *aliasstudio\_2009* folder in the **Save in** drop-down list, all files and folders are displayed in the list box.
4. Choose the **Create New Folder** button from the **Save Wire** dialog box and name the new folder as *c03\_tutorials*.
5. Double-click on the *c03\_tutorials* folder to display it in the **Save in** drop-down list.
6. Next, choose the **Set Current** option from the **Project** field and then enter *c03\_tut01* in the **Object name** edit box.
7. Choose the **Save** button; AliasStudio saves the file as *c03\_tut01.wire*.

The location of this file is given below:

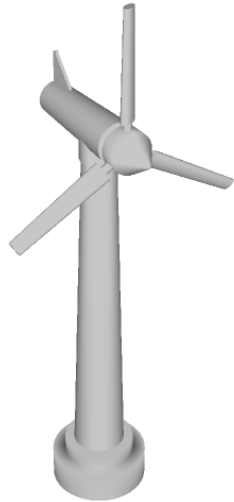
*/aliasstudio\_2009\c03\_tutorials\c03\_tut01.wire*

8. Choose **File > Exit** from the menu bar to close the file.

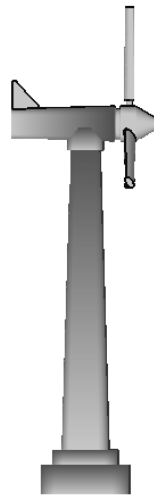
## Tutorial 2

In this tutorial, you will create the model shown in Figure 3-85. Figure 3-86 shows the model in the **Left** window. After creating the model, you will shade it to give a realistic look.

(Expected time: 1 hr 30 min)



*Figure 3-85 Model for Tutorial 2*



*Figure 3-86 View of the model in the Left window*

The following steps are required to complete this tutorial:

- a. Start a new wire file.
- b. Create the base feature of the model by using the **Revolve surface** tool.
- c. Create the central part of the model by using the **Skin surface** tool.
- d. Turn off the visibility of curves used in the base feature and central part.
- e. Create the blade joint by using the **Cylinder** tool.
- f. Create the blade of the model by using the **Skin surface** tool.
- g. Close the ends of the blade by using the **Set planar** tool.
- h. Group the blade joint with the close-ended blade by using the **Edit** menu.
- i. Copy and rotate the above grouped parts by using the **Edit** menu and the **Rotate** tool, respectively.
- j. Create the tail piece by using the **Skin surface** tool.
- k. Close the ends of the tail piece by using the **Set planar** tool.
- l. Shade the model by using diagnostic shading.
- m. Save the model and exit.

### Starting a New Wire File

1. Choose **Start > Programs > Autodesk > AliasStudio 2009 > AliasStudio 2009** from the Taskbar; the Autodesk Studio window informing about the start of the software is displayed.

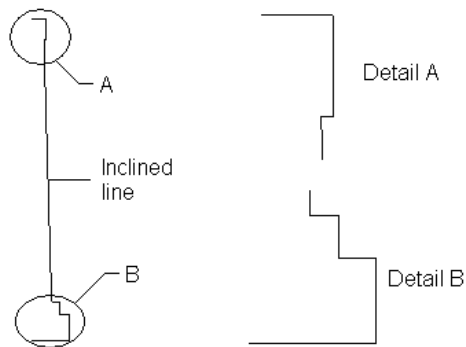
As AliasStudio 2009 gets started, a Studio screen is displayed. You need to start a new wire file for creating the model.

2. Choose **File > New** from the menu bar; a new AliasStudio wire file gets started and four windows are displayed on the screen.

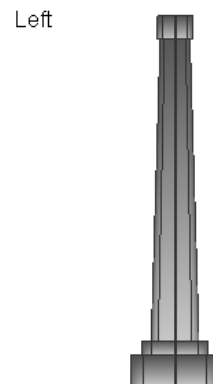
### Creating the Base Feature

The base feature of the model is created by using the **Revolve surface** tool. First, create the sketch of the required profile by using the **Polyline** tool and then revolve the same profile.

1. Choose **Layouts > Left** from the menu bar; the **Left** window expands to fill the entire interface screen.
2. Press and hold the left mouse button on the **Line** button; a flyout is displayed.
3. Choose the **Polyline** button from this flyout and create the profile, as shown in Figure 3-87. You can use the **Snap to Grid** option to create the profile.
4. Double-click on the **Revolve surface** button; the **Revolve Options** dialog box is displayed.
5. Select the **Z** and **Global** radio buttons from the **Revolution Axis** and **Axes** areas, respectively, and then choose the **Go** button to exit the **Revolve Options** dialog box.
6. Select the polyline from the **Left** window; the base feature is created, as shown in Figure 3-88.



**Figure 3-87** Profile for the base feature



**Figure 3-88** Base feature of the model









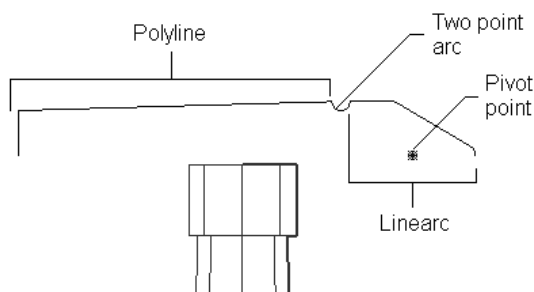
#### Note

The endpoints of profiles need to lie on the Z-axis. Keeping the endpoint of the profile on Z-axis allows you to create the closed revolved feature. Also, the lower endpoint need to be at the origin.

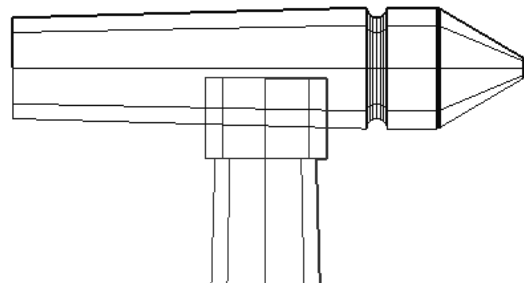
## Creating the Central Part

After creating the base feature, you need to create the central part of the model by using the **Revolve surface** tool.

1. Choose the **Polyline** button from the **Keypoint Curve Toolbox** and create polyline, as shown in Figure 3-89. 
2. Choose the **Arc (two point)** button from the **Keypoint Curve Toolbox** and create an arc, refer to Figure 3-89. 
3. Next, choose the **Line-arc** button from the **Keypoint Curve Toolbox** and create the linearc, refer to Figure 3-89. 
4. Choose the **Join Curves** button from the **Keypoint Curve toolbox**; you are prompted to select keypoints. 
5. Click in the active window where the keypoints of the polyline and the arc coincide; the curves are attached.
- Similarly, attach the linearc with the two point arc.
6. Next, choose the **Set pivot** button; you are prompted to specify the new position of the pivot point. 
7. Drag the pivot point to the location, refer to Figure 3-89.
8. Double-click on the **Revolve surface** button from the **Surfaces** tab in the **Palette**; the **Revolve Options** dialog box is displayed. 
9. Select the **X** and **Local** radio buttons from the **Revolution Axis** and **Axes** areas, respectively, and then choose the **Go** button.
10. Select the joined curves; the central part of the model is created, as shown in Figure 3-90.



**Figure 3-89** Profile of the central part



**Figure 3-90** Central part



### Turning off the Display of Curves

For clear display of the model, you need to turn off the display of the curves used to create the base feature and central part. You can do so by using layers.

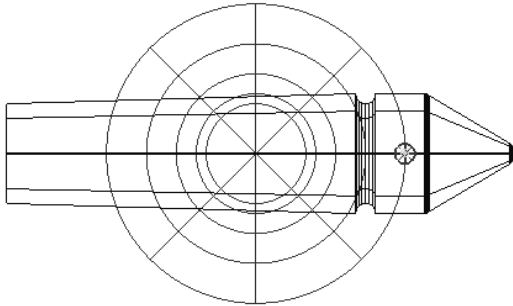
1. Choose **Layers > New** from the menu bar; a new layer L1 will be displayed in the Layers bar.
2. Double-click on the layer L1; the layer text edit box is displayed.
3. Enter **Base Feature Curve** in the layer text edit box and press ENTER; the layer L1 is renamed to Base Feature Curve.
4. Next, select the profile curve used in the base feature; the selected curve gets highlighted.
5. Press and hold the left mouse button on the layer Base Feature Curve; the layer flyout is displayed.
6. Choose the **Assign** option from the layer flyout; the curves are assigned to the layer Base Feature Curve.
7. Press and hold the left mouse button to display the layer flyout again.
8. Choose the **Visible** option from the layer flyout; the visibility of the curve assigned to the layer Base Feature Curve is turned off.
9. Similarly, create a new layer and rename it as Central Part Curves. Assign the joined curves (polyline, two point arc, and linearc) to this layer and then turn its visibility off.

### Creating the Blade Joint

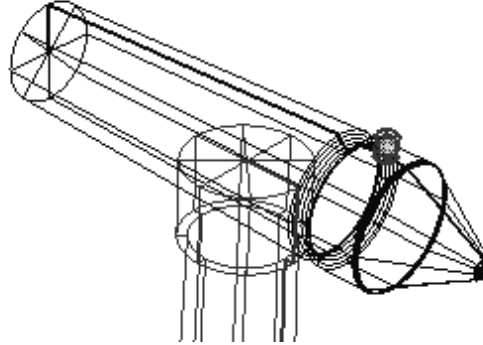
The blade joint is created on the top of the central part by using the **Cylinder** tool.

1. Choose the **Cylinder** button from the **Surfaces** tab in the **Palette**; you are prompted to specify the position of the cylinder. 
2. Click in the **Top** window to specify the position of the cylinder; you will notice in the **Perspective** window that the cylinder is created at the base feature.
3. Use the manipulator in the **Left** window to position the cylinder at the required position.
4. Choose the **Scale** button from the **Transform** tab in the **Palette**; you are prompted to enter the scale factor. 
5. Enter **0.4** in the promptline and press the ENTER key; the blade joint is scaled to the specified scale, as shown in Figure 3-91. Figure 3-92 shows the scaled blade joint in the **Perspective** window.

Top



*Figure 3-91 Blade joint after scaling*

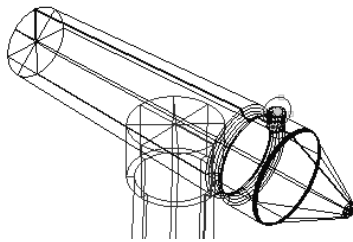


*Figure 3-92 Blade joint displayed in the Perspective window*

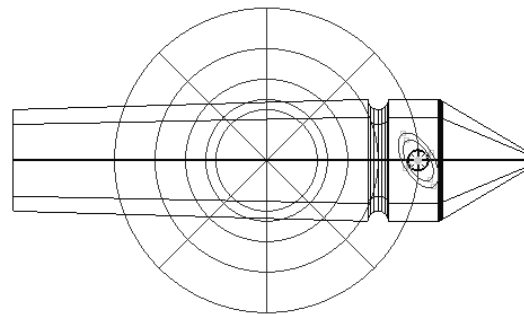
### Creating the Blade

Blades are created by using the **Skin surface** tool. You need to create two closed curves for the skin surface.

1. Choose the **Ellipse** button from the **Keypoint Curve Toolbox**; you are prompted to specify the position of the ellipse.
2. Click in the window to specify the position of the first ellipse.
3. Create another ellipse in line with the first ellipse, as shown in Figure 3-93. Figure 3-94 shows two ellipses in the **Top** window.



*Figure 3-93 Two ellipses created in line with the skin surface*



*Figure 3-94 Ellipses displayed in the Top window*


4. Next, choose the **Skin surface** button from the **Surfaces** tab in the **Palette**; you are prompted to select the first curve.

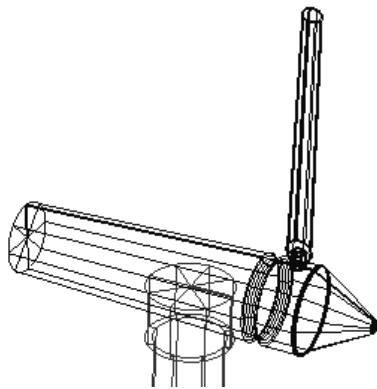


5. Select the first ellipse; you are prompted to select the next curve.
6. Select the second ellipse; the skin surface is created, as shown in Figure 3-95.

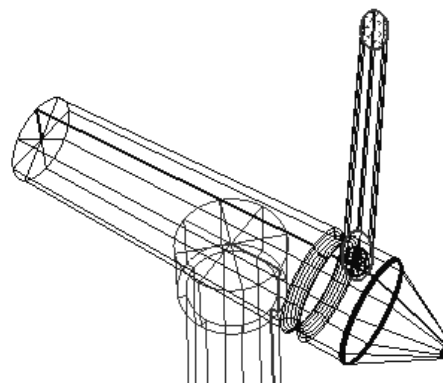
### Closing the Ends of the Blade

The ends of the blade can be closed by using the **Set planar** tool.

1. Choose the **Set planar** button from the **Surfaces** tab in the **Palette**; you are prompted to select the curve. 
2. Select the top and bottom edges of the skin surface and then choose the **Go** button; the planar surfaces are created, as shown in Figure 3-96.



*Figure 3-95 Skin surface*



*Figure 3-96 Planar surfaces created at the top and bottom of the blade*


### Grouping the Blade, Blade Joint, and Planar Surfaces

You need to group the blade joint, blade and its lower and upper planar surfaces.

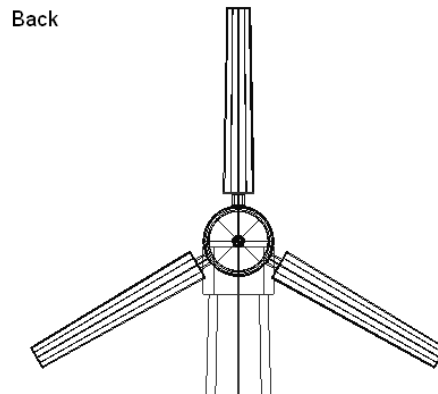
1. Select the blade joint, blade and the lower and upper planar surfaces of the blade; the selected entities are highlighted.
2. Choose **Edit > Group** from the menu bar; the selected entities are grouped.

### Creating the other Instances of Grouped Entities

After grouping the above-mentioned entities, you need to create two more instances of grouped entities.

1. Choose **Edit > Copy** and then **Edit > Paste** from the menu bar; the grouped entity is copied and placed exactly at the original grouped feature.
2. Next, choose the **Rotate** button from the **Transform** tab in the **Palette**; you are prompted to enter the rotation angles about the X, Y, and Z-axes. 




3. Enter **120, 0**, and **0** in the promptline and press ENTER. The grouped entity is rotated, as specified about the X, Y, and Z-axes.
4. Create the third instance of the grouped entity, as shown in Figure 3-97.



*Figure 3-97 Model after creating all instances of grouped entity*

### Creating the Tail Piece

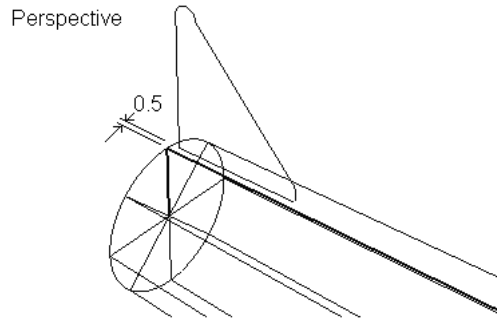
The tail piece is created by using the **Skin surface** tool. You need to create two curves for creating a skin surface.

1. Choose the **Line-arc** button from the **Keypoint Curve Toolbox** and create profile. You need to create the profile at an offset of 0.1 from the center of the central part of the model, as shown in Figure 3-98. 
2. Choose **Edit > Copy** and then **Edit > Paste** from the menu bar; the linearc profile is copied and placed exactly at the original curve.
3. Next, choose the **Move** button from the **Transform** tab in the **Palette** and move the linearc to a new position, as shown in Figure 3-99. 
4. Choose the **Skin surface** button from the **Surfaces** tab in the **Palette**; you are prompted to select the first curve. 
5. Select the first linearc profile; you are prompted to select the next curve.
6. Select the copied linearc profile; the skin surface is created, as shown in Figure 3-100.

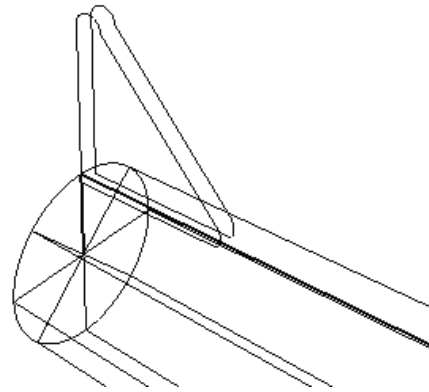
### Closing the Front and Back Faces of the Tail Piece

You can close the front and back faces of the tail piece by using the **Set planar** tool.




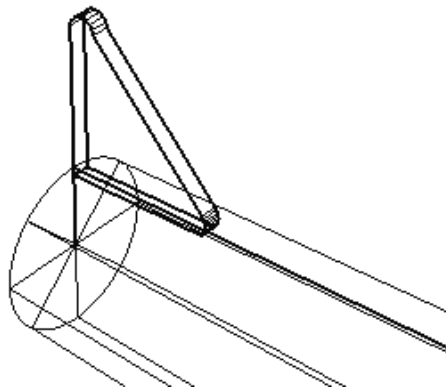


**Figure 3-98** Profile for tail piece

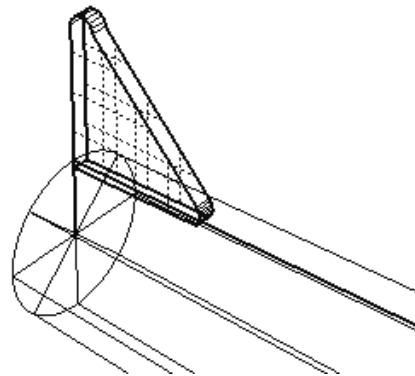


**Figure 3-99** Copied profile moved to a new position

1. Choose the **Set planar** button from the **Surfaces** tab in the **Palette**; you are prompted to select the curve. 
2. Select both edges of the skin surface and choose the **Go** button; the two planar surfaces are created, as shown in Figure 3-101. Figure 3-102 shows the complete model.



**Figure 3-100** Skin surface



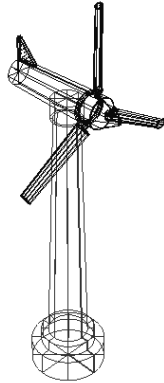
**Figure 3-101** Planar surfaces created at the front and back faces

### Diagnostic Shading of the Model

The model you created will be in the wireframe mode. For a better view, you need to shade the model.

1. Choose **Object display > Diagnostic shading** from the menu bar; the **Diagnostic shading** dialog box is invoked. You can also invoke this dialog box by choosing **Windows > Control Panel** from the menu bar; the **Control Panel** is displayed along with the **Diagnostic Shading** area at the bottom of this panel.

2. Choose the **User-defined texture** button from the **Diagnostic Shading** area; the model is shaded, as shown in Figure 3-103. You can choose the rest of the buttons in this area to get other shades.



**Figure 3-102** Complete model



**Figure 3-103** Model after shading



**Note**

After shading the model, select the complete model from the active window and clear the **Isoparm U** and **V** check boxes from the **Display** area of the **Control Panel**. It will hide the isoparametric **U** and **V** curves, thus giving a more realistic view of the model.

*You can turn off the visibility of curves used for creating blades and tail piece by using layers, as discussed earlier in this tutorial.*

## Saving the File

You need to save the created model in the *c03\_tutorials* folder with the name *c03\_tut02*. As you have already created the *c03\_tutorials* folder in *Tutorial 1* and set it as the current project, you need not create a new folder.

1. Choose **File > Save as** from the menu bar; the **Save Wire** dialog box is invoked. You can also invoke this dialog box by pressing the ALT and S keys together.
2. Click on the down arrow given on the right of the **Go** field to display the drop-down list and choose **Current Project** from this list.
3. Enter *c03\_tut02* in the **Object name** edit box and then choose the **Save** button; AliasStudio saves the file as *c03\_tut02.wire*.

The location of this file is given below:

`\\aliasstudio_2009\\c03_tutorials\\c03_tut02.wire`

4. Choose **File > Exit** from the menu bar to close the application.

### Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. In the **Revolution Axis** area of the **Revolution Options** dialog box, the **Z** radio button is selected by default. (T/F)
2. The **Create Caps** area is used with open generation curves. (T/F)
3. For creating a skin surface, you need atleast two curves. (T/F)
4. The **Set planar** tool is used to create a trimmed surface. (T/F)
5. AliasStudio allows you to cut, copy, and paste objects. (T/F)
6. Press \_\_\_\_\_ keys to copy an object or a component.
7. The \_\_\_\_\_ command is used to repeat the last executed command.
8. In case of uniform knot spacing, the \_\_\_\_\_ of the assigned texture occurs to fit evenly between the isoparametric curves.
9. Selecting the \_\_\_\_\_ radio button in the **Layers options** area of the **Paste Options** dialog box allows you to specify the layer where the objects are pasted.
10. While pasting an object using the **duplicate layers** option from the **Paste options** menu, a duplicate layer named \_\_\_\_\_ will be created if the existing layer is named as **Line**.

### Review Questions

Answer the following questions:

1. Which of the following degrees is selected by default in the **Surface Degree** area of the **Revolve Options** dialog box?  
(a) **1** (b) **2**  
(c) **3** (d) **7**
2. How many additional isoparametric curves are created on the surface if you enter **3** in the **Number of Spans** edit box in the **Skinning Options** dialog box?  
(a) **1** (b) **2**  
(c) **3** (d) **4**

3. Which of the following keys is pressed to create a single skin surface from a set of multiple curves?
 

|           |         |
|-----------|---------|
| (a) SHIFT | (b) ALT |
| (c) CTRL  | (d) TAB |
4. In the **Revolve Options** dialog box, the **Global** radio button is selected by default in the **Axes** area. (T/F)
5. In the **Extrude Options** dialog box, the **Extrude pivot** area is displayed only when the **Tube** radio button is selected from the **Style** area. (T/F)
6. A curve of constant U or constant V parameter is known as isoparametric curve. (T/F)
7. The \_\_\_\_\_ edit box in the **Preset Grid Options** dialog box is used to specify the number of subdivision lines between the two main grid lines.
8. The \_\_\_\_\_ area is used to specify the condition at the ends of the extruded surface.
9. The **Set planar** tool works only on the profiles that form a \_\_\_\_\_ loop.
10. You can reverse the effect of the previous command by choosing the \_\_\_\_\_ from the menu bar.

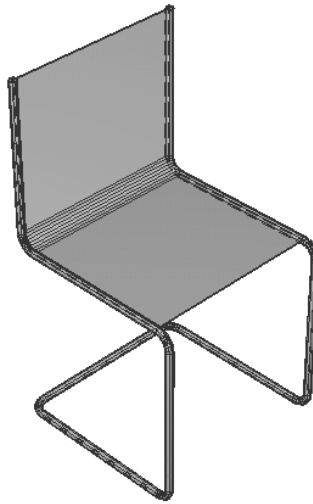
## Exercises

### Exercise 1

Create the model, as shown in Figure 3-104. After creating the model, you will shade it to give a realistic look. **(Expected time: 30 min)**

#### Hint

1. Create the chair padding (legs) by using the **Extrude** tool.
2. Choose the **Circle** button from the **Curves** tab to create the profile curve.
3. After creating the generation curve, create the path curve by using the **Line-arc** tool.
4. Specify the position of different segments of the linearc by entering coordinates in the promptline.
5. Extrude the circle along the path curve to create chair padding.
6. Create the seat of the chair by using the **Skin surface** tool.



*Figure 3-104 Model for Exercise 1*

## Exercise 2

Create the model, as shown in Figure 3-105. After creating the model, you will shade it to give a realistic look. **(Expected time: 45 min)**



*Figure 3-105 Model for Exercise 2*

**Answers to Self-Evaluation Test**

**1. T, 2. F, 3. T, 4. T, 5. T, 6. CTRL+C, 7. Reinvoke last, 8. stretching, 9. Always ask, 10. Line#2**