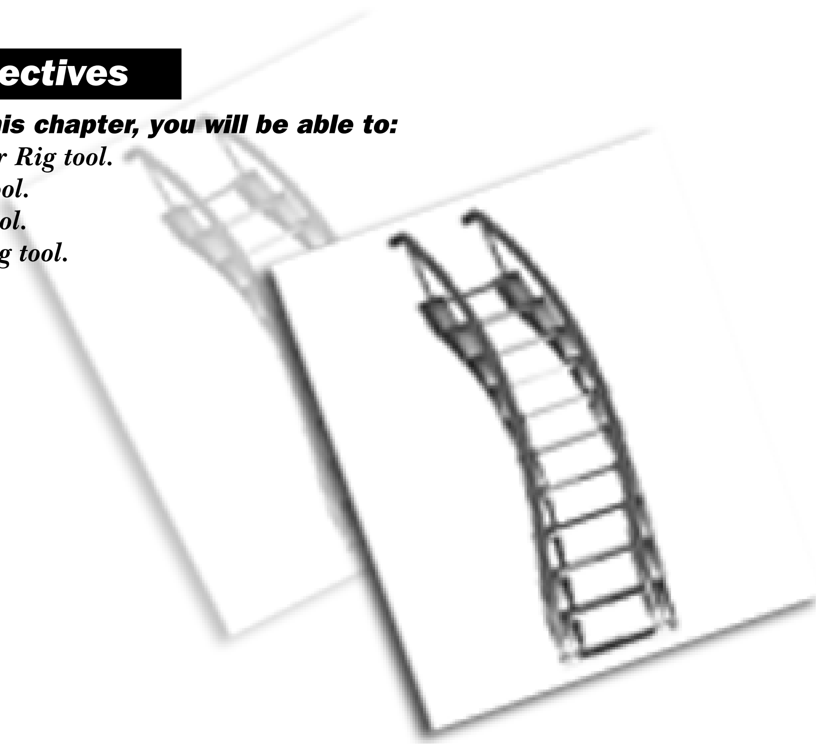
Chapter 11

Advanced Editing Tools



Learning Objectives

After completing this chapter, you will be able to:

- *Use the Transformer Rig tool.*
- *Use the Twist Rig tool.*
- *Use the Bend Rig tool.*
- *Use the Conform Rig tool.*
- *Generate meshes.*
- *Save meshes.*
- *Edit meshes.*
- *Repair meshes.*

DYNAMIC SHAPE MODELING

In the previous chapter, you learned about the **Lattice Rig** tool that is used to modify surfaces, meshes, or hybrid models globally. In this chapter, you will learn about the rest of dynamic shape modeling tools such as **Transformer Rig**, **Twist Rig**, **Bend Rig**, and **Conform Rig**. The important terms used in the dynamic modification of objects are discussed next.

Target

The object or the geometry on which modification will be applied is known as the target. A target can be flexible or rigid. Flexible targets can be moved, rotated, and modified to change their shape by transforming them, whereas rigid targets can be moved and translated but their shape cannot be modified by transforming them. Examples of targets are surface, group of surfaces, meshes, and curves.

Modifier

AliasStudio allows you to add geometries that help you modify targets as desired. These geometries are known as modifiers. Modifiers can be of two types, free and predefined. Free modifiers help you shape all targets at one time. You can select a surface or a curve as a free modifier and then modify it to achieve the desired target geometry. Predefined modifier is a pair of similar parameterized geometries that are used to modify target geometry.

Constraints

Constraints are added to targets for restricting the modification of their shapes. When you select the target geometry for applying transformation by using the **Transformer Rig** tool, AliasStudio analyzes the target and creates a Region of Interest (ROI) on it. The parts of the target geometry that lie inside the ROI can be moved, but the parts that lie outside the ROI are constrained and cannot be moved.

Clampers

Even after applying constraints on the target, the modification may occur outside the target geometry. Therefore, to ensure that modification occurs inside the ROI only, you can use clampers. These clampers enable the **Transformer Rig** tool to understand the ROI better, and help it limit the modification to occur only on the desired part of the target geometry.

Modifying an Object Using the Transformer Rig Tool

Palette: Object Edit > Transformer Rig

Like the **Lattice Rig** tool, the **Transformer Rig** tool is used to modify the surface, group of surfaces, meshes, and hybrid models globally. However, using the **Transformer Rig** tool, you can control the modifications more precisely.

*Figure 11-1 The **Transformer Rig Toolbox***

To modify the shape dynamically using this tool, choose the **Transformer Rig** button from the **Object Edit** tab in the **Palette**; the **Transformer Rig Toolbox** will be displayed, as shown in Figure 11-1. Also, the **Accept Targets** button in inactive state will be displayed at the lower right corner of the active window and you will be prompted to select the target geometry. Select the object to be modified; the object will be highlighted in yellow and the **Accept Targets** button will be activated. Choose the **Accept Targets** button; the **confirm** message box will be displayed. Choose the **Yes** button from the message box; you will be prompted to select modifiers or add new modifiers. Also, the **Go** button in inactive state will be displayed at the lower right corner of the active window. Next, choose the **Add Free Modifiers** button from the **Transformer Rig Toolbox**; you will be prompted to select the geometry to add as modifier. Also, the **Cancel** and **Accept Modifiers** buttons will be displayed at the lower right corner of the active window. Select the modifier from the active window; the modifier geometry will be highlighted in yellow and the **Accept Modifiers** button will get activated. Choose the **Accept Modifiers** button; the color of the modifier will change to light blue and the **Go** button will be displayed at the lower right corner of the active window and you will be prompted to add the constraint geometry. Next, choose the **Add Constraints** button from the **Transformers Rig Toolbox**; you will be prompted to select the geometry to add as constraints. Also, four buttons, **Tan**, **Pos**, **Cancel**, and **Accept Constraints** will be displayed at the lower right corner of the active window. Select the constraint from the active window; the selected constraint geometry will be highlighted in yellow and the **Accept Constraints** button will get activated. Choose the **Accept Constraints** button; the constraint will change its color to red and the **Go** button will be displayed at the lower right corner of the active window. Choose the **Go** button; the **confirm** message box informing about the creation of history of the transformer rig will be displayed. Choose the **OK** button from the **confirm** message box. Next, exit the **Transformer Rig** tool by choosing **Pick nothing** button from the **Pick** tab of the **Palette**. Next, move or scale the modifier by using the **Transform** tools; the image will be modified. Figure 11-2 shows target, modifier, and constraint and Figure 11-3 shows the object modified dynamically.

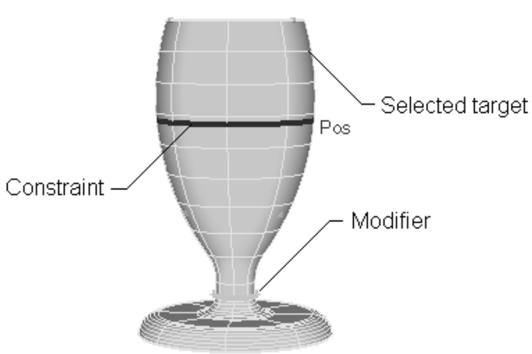


***Figure 11-2** Target, modifier, and constraint*

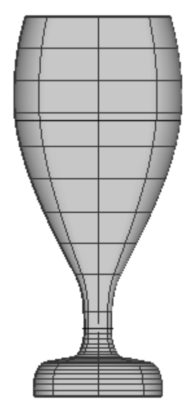


***Figure 11-3** Model after dynamic modification*

Note

Constraints are used to restrict the modification of the shape to the specified area of the object during transformation. You can modify objects dynamically without adding constraints. In such a case, you cannot restrict the movement of the object. The addition of constraints will be discussed in detail later in this chapter.

You can set different parameters of the **Transformer Rig** tool. To do so, select the object and then choose the **Accept Targets** button that will be displayed when you select the object. Next, double-click on the **Transformer Rig** button in the **Transformer Rig Toolbox**; the **Transformer Rig Options** dialog box will be displayed, as shown in Figure 11-4. Most of the options in this dialog box are the same as those discussed in the **Lattice Rig Options** dialog box. The rest of the options in this dialog box are discussed next.

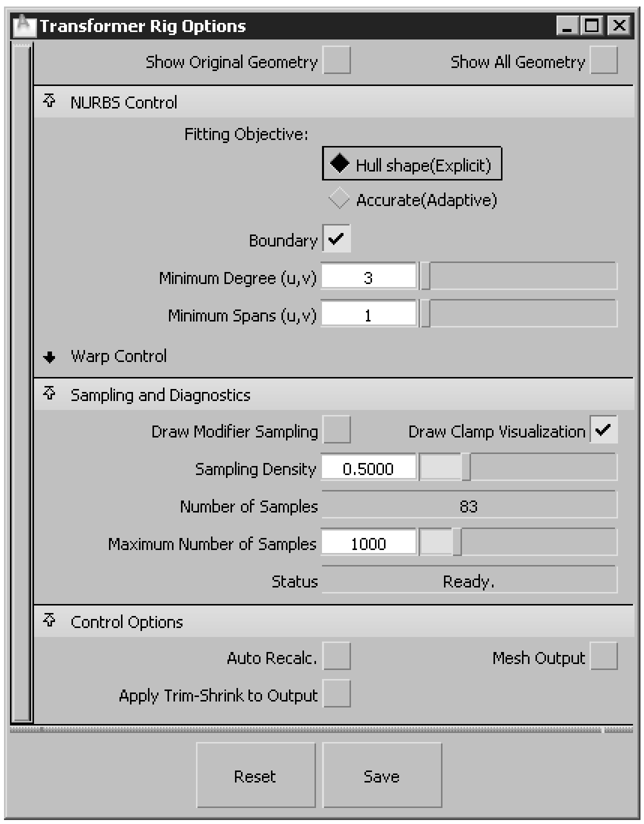


Figure 11-4 *The options in the* ***Sampling and Diagnostics*** *area of the* ***Transformer Rig Options*** *dialog box*

Sampling and Diagnostics

The **Sampling and Diagnostics** area is used to control the quality and the performance of the modifications made on the object by using the **Transformer Rig** tool. Click on the down-arrow on the left of the **Sampling and Diagnostics** area to display the options in it, refer to Figure 11-4.

The options in the **Sampling and Diagnostics** area of the **Transformer Rig Options** dialog box are discussed next.

Draw Modifier Sampling

Select this check box to make sampling of modifiers and constraints visible, as shown in Figure 11-5. If there are few or no constraint in the window, eight tangent constraint points at the corners of an imaginary box surrounding the target will be displayed, as shown in Figure 11-6. The size of this box will be twice the size of the bounding box of the target geometry. These tangent constraint points check the excessive deformation of the target while modifying the geometry. If the tangent constraint points are invisible, select the **Auto Recalc.** check box from the **Control Options** area of the **Transformer Rig Options** dialog box to make them visible.

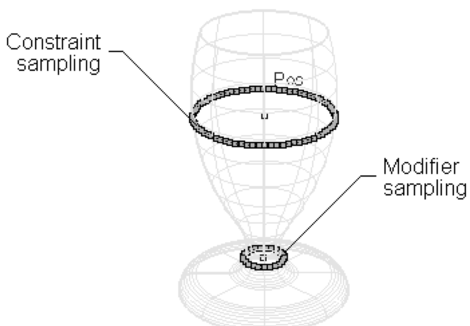


Figure 11-5 *Modifier sampling and constraint sampling*

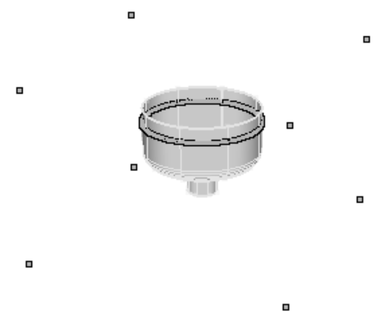


Figure 11-6 *Tangent constraint points*

Draw Clamp Visualization

This check box is selected by default and is used to visualize the regions that cannot be modified. You can differentiate between the regions that can be modified and the regions that cannot be modified by comparing the color of dots on the target. These dots will be displayed only when you add modifiers and constraints to the target. The region with green colored dots can be modified, whereas the region with red colored dots cannot be modified. Select the dots with the middle mouse button to improve the visualization of the dots. Figure 11-7 shows the regions with red and green dots.

Sampling Density

This edit box is used to specify the sampling density of the modifier and the constraint geometry. Alternatively, you can specify the sampling density of the modifier and the constraint geometry by using the slider bar given on the right of this edit box. Samples control the modification of the object. Increasing the sampling density improves accuracy in the modification but it also increases the time to do the modification. The range of sampling density is 0.05 to 2.0. Figure 11-8 shows the model of Figure 11-7 with reduced sampling density.

Number of Samples

This display box displays the number of samples used to modify the object dynamically. This display box is read-only and its value changes as you change the value in the **Sampling Density** edit box.

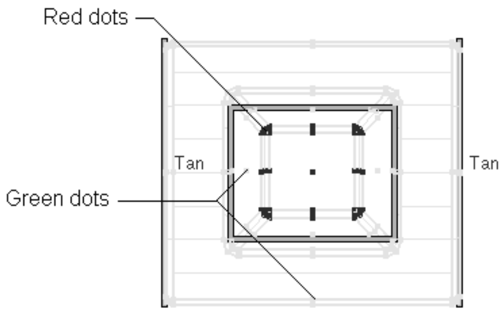


Figure 11-7 Regions showing red and green dots

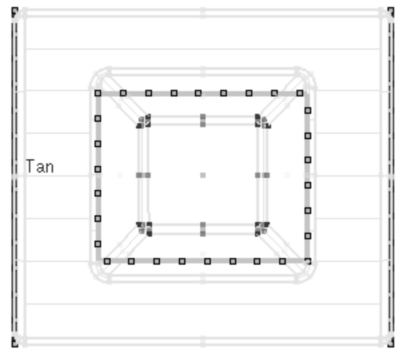


Figure 11-8 Model with reduced sampling density

Maximum Number of Samples

This edit box is used to specify the maximum number of samples to be used for modifying the object dynamically. The range of maximum number of samples is 100 to 5000. You can also specify the maximum number of samples by using the slider bar given on the right of this edit box. If the number of samples exceeds the maximum number of samples specified in the **Maximum Number of Samples** edit box, the message **exceeds maximum allowed** will be displayed in the **Number of Samples** display box.

Status

The **Status** display box shows the current status of the shape modification. If the shape modification fails, a message will be displayed in this display box. Figure 11-9 shows the failed modification of a cube. The message displayed in this case is **Failed. Stress exceeds threshold.**

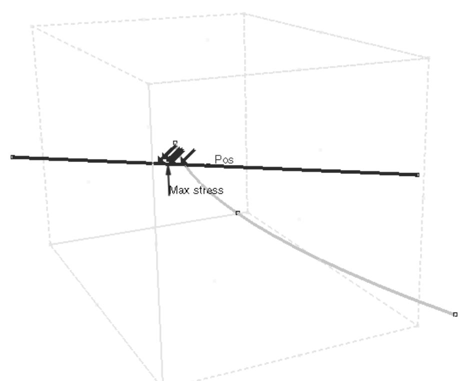


Figure 11-9 Failed modification of the cube

Transformer Rig Toolbox

When you choose the **Transformer Rig** button from the **Object Edit** tab in the **Palette**; the **Transformer Rig Toolbox** will be displayed, refer to Figure 11-1. The buttons in the **Transformer Rig Toolbox** are discussed next.

Transformer Rig

This button is chosen by default and is used to select the geometry to be modified. When you double-click on this button, the **Transformer Rig Options** dialog box will be displayed. The options in this dialog box have been discussed earlier in this chapter.

Add Free Modifiers

To edit an object dynamically, you need to add a modifier to it. You can add free modifiers to the model that modify all selected objects at a time. To do so, choose the **Add Free Modifiers** button from the **Transformer Rig Toolbox**; you will be prompted to select the geometry to act as modifier. Select the required geometry from the active window; the **Accept Modifiers** button will get activated. Next, choose the **Accept Modifiers** button; the selected geometry will be added as a modifier.

Add Constraints

Constraints are added to targets for restricting the shape modification of the target. When constraints are added to target, AliasStudio analyzes the target and creates a Region of Interest (ROI). To add constraints to the target, choose the **Add Constraints** button from the **Transformer Rig Toolbox**; you will be prompted to select the geometry that will act as constraint. Select the required geometry from the active window; the selected geometry will be highlighted in yellow. Next, choose the **Accept Constraints** button displayed at the lower right corner of the active window; the constraints will be added to target and the color of the selected geometry will change to red. Also, default labels, **Pos**, will appear on constraints, as shown in Figure 11-10. These labels indicate the type of continuity achieved by the constraint. You can change the continuity by choosing the **Tan** button before choosing the **Accept Constraints** button from the lower right corner of the active window. You can also change the type of continuity achieved or remove the constraint. To do so, select the constraint with the right mouse button to invoke a shortcut menu and then select the desired continuity from it. Figure 11-11 shows a hair dryer and Figure 11-12 shows the handle of a hair dryer modified after adding constraints.

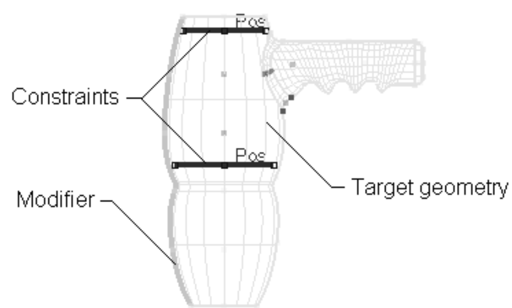


Figure 11-10 *Constraints added to the model*

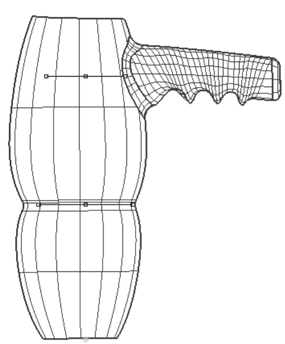


Figure 11-11 *Hair dryer*

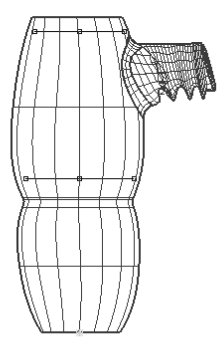


Figure 11-12 *Target modified after using constraints*

Add Flexible Targets

As discussed earlier, flexible targets can be moved, rotated, and modified to change their shape by transforming them. To add a flexible target to a model, choose the **Add Flexible Targets** button from the **Transformer Rig Toolbox**; you will be prompted to select the geometry to be added as target. Select the required geometry and then use the transformation tools (**Move**, **Nonproportional scale**, **Rotate**, or **Scale**) to view the effect of adding flexible targets. You can select curves, surfaces, or meshes as flexible targets. Figure 11-13 shows the flexible target added to an object and Figure 11-14 shows the model that has been modified by using the flexible target.

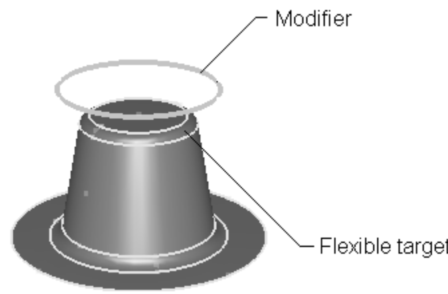


Figure 11-13 *Flexible target added to the model*

Add Rigid Targets

Rigid targets can be moved and translated but their shapes cannot be modified by applying transformation. To add a rigid target to an object, choose the **Add Rigid Targets** button; you will be prompted to select the geometry to be added as target. Select the geometry to be made rigid and then use the transformation tools (**Move** or **Rotate**) to view the effect of adding these targets. Note that you cannot scale or shear the rigid targets. Also, when you select a rigid target and choose the **Accept Targets** button, the selected target will be highlighted in green color with the label **Rigid (0)** on top of it. This indicates that the selected rigid target belongs to the 0 group. If you add another rigid target to the object, the rigid target will be highlighted in yellow with the label **Rigid (1)** on top of it. This rigid target will belong to the 1 group. Figure 11-15 shows the model after modifying it by using the rigid target.

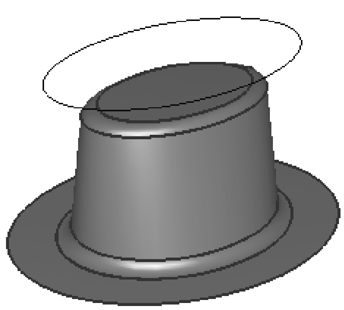


Figure 11-14 *Model modified by using the flexible target*

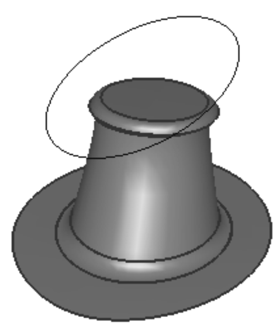


Figure 11-15 *Model modified by using the rigid target*

Add Predefined Modifiers

You can also add predefined modifiers to modify the object dynamically. Predefined modifier is a set of two similar parameterized geometries. While adding predefined modifiers, you need to select two geometries, the origin and the destination. To add a predefined modifier to an object, choose the **Add Predefined Modifiers** button from the **Transformer Rig Toolbox**; you will be prompted to select the origin. Select the geometry that will act as the origin for the predefined modifier; the **Accept Origin** button will get activated and the selected geometry will be highlighted in yellow. Choose the **Accept Origin** button; the color of the selected origin geometry will change to light blue and you will be prompted to select the destination. Select the geometry that will act as destination for the predefined modifier; the **Accept Destination** button will get activated and the selected geometry will be highlighted in yellow. Choose the **Accept Destination** button; the **Go** button will be displayed at the lower right corner of the active window and the color of the selected destination geometry will change to light blue. Choose the **Go** button; the **confirm** message box informing about the creation of the transformer rig history will be displayed. Choose the **OK** button from the **confirm** message box; the predefined modifier will be added to the object to be modified. Figure 11-16 shows the origin and destination geometries of a predefined modifier and Figure 11-17 shows the target modified by the added predefined modifiers.

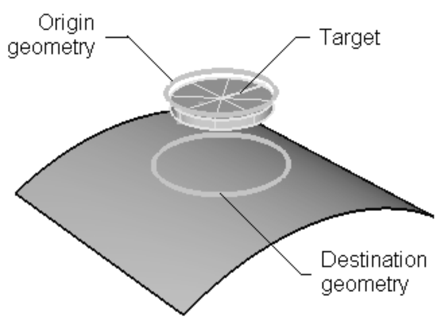


Figure 11-16 *Origin and destination geometries of a predefined modifier*

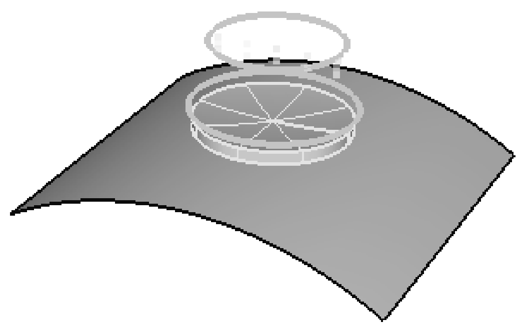


Figure 11-17 *Target modified by the predefined modifiers*

Add Clampers

Sometimes, it is not possible for constraints to control the modification of the target completely. As a result, the shape is modified outside the ROI. This happens because the **Transformer Rig** tool fails to identify the parts to be modified properly . To limit the modification of the target inside the ROI only, you can use clampers. These clampers provide a clue to the **Transformer Rig** tool about the regions to be modified and the regions to be remain stationary. Clampers are added only after adding modifiers and constraints to the object. To add a clamper to an object, choose the **Add Clampers** button from the **Transformer Rig Toolbox**; you will be prompted to click the geometry to add clamp to. Also, the **Done** button will be displayed at the lower right corner of the active window. Click on the target geometry that needs to be fixed at the required location; a pink indicator will be placed at the specified location, confirming about the setting of the clamper. After adding the clamper to the target, choose the **Done** button. Now, modify the target using any of the transform tools; you will notice that modification of the target gets restricted after adding clampers. Figure 11-18 shows the clampers added to the target and Figure 11-19 shows the shape modification restricted to the lower part of the hair dryer.

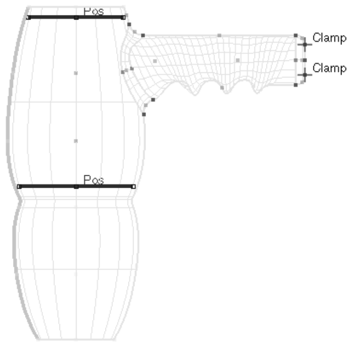


Figure 11-18 *Clampers added to the target*

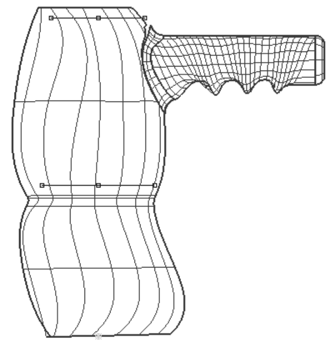


Figure 11-19 *Shape modification restricted to the lower part of the hair dryer*

If you add clampers to the model that has been modified dynamically, the model will get updated automatically to adjust with the constraints applied by clampers. Also, when you add clampers to the target, the green dots near the clamper turn to red.

Remove From Rig

Sometimes, you may add wrong modifiers, constraints, or clampers to the target. To remove them, choose the **Remove From Rig** button from the **Transformer Rig Toolbox**; you will be prompted to select the geometry to be removed. Also, the **Cancel** (active state) and **Remove Selected** (inactive state) buttons will be displayed at the lower right corner of the active window. Select the geometry (modifier, constraint, or clamper) to be removed; the selected geometry will be highlighted in yellow and the color of the target will change to pink. Also, the **Remove Selected** button will get activated. Choose the **Remove Selected** button from the lower right corner of the active window; the selected geometry (modifier, constraint, or clamper) will be removed from the target.

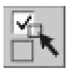
Revert

When a target is accepted for dynamic modification, the duplicate copy of the target is created. You may need to restore the original target geometries and delete the modified target geometries. To restore the original target, choose the **Revert** button from the **Transformer Rig Toolbox**; the **confirm** message box informing about the deletion of the modified target geometry and its construction history will be displayed. Choose the **Yes** button from the **confirm** message box; the original target geometries will be restored.

Commit

Choose the **Commit** button to accept the modification done in an object or a model. When you choose this button, the original geometry along with its construction history will be deleted. After modifying the target by using the **Transformer Rig** tool, if you choose the **Commit** button; the **confirm** message box informing about the deletion of the original target geometry and its construction history will be displayed. Choose the **Yes** button from the **confirm** message box to accept modification.

Show Pick Mask



The **Show Pick Mask** button is used to select the required entities and filter out the unwanted entities while modifying object dynamically. If you invoke the **Transformer Rig** tool and then select the required target, the entities such as surfaces, points, meshes, and curves will be selected by default. Choose the **Show Pick Mask** button before selecting the target; the **Transformer Rig Pick Mask** dialog box will be displayed, as shown in Figure 11-20. Select the check boxes of the entities that you want to pick. Now, whenever you select or accept the target, modifier, or constraint, only the entities selected in the **Transformer Rig Pick Mask : Add Target Mode** dialog box will be picked.

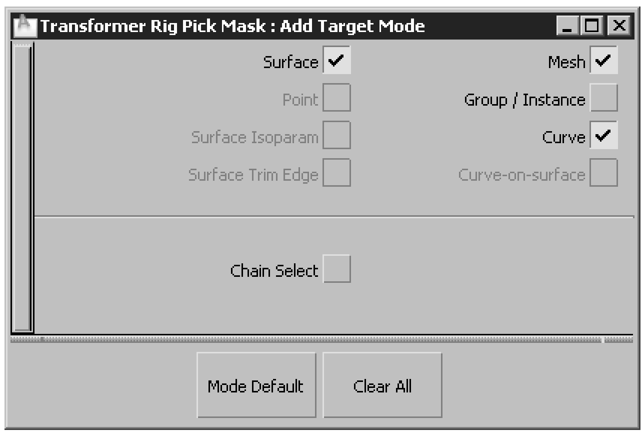


Figure 11-20 *The* ***Transformer Rig Pick Mask*** *dialog box*

Note

The ***Transformer Rig Pick Mask*** *dialog box changes its mode depending upon the entity (target, modifier, constraint, or clamper) selected or accepted.*

The **Show Pick Mask** button will be active only when you invoke the **Transformer Rig** tool. Also, this button will be active whenever you select or accept a target, modifier, constraint, or clamper. Select or clear the check boxes in the **Transformer Rig Pick Mask** dialog box to pick or unpick the corresponding entities from the model. This dialog box can be used similar to the **Lattice Rig Pick Mask** dialog box.

Modifying an Object Using the Twist Rig Tool

Palette:	Object Edit > Transformer Rig > Twist Rig

You can modify an object (a surface or a group of surfaces) dynamically by twisting it about an axis. To do so, choose the **Twist Rig** button from the **Object Edit** tab in the **Palette**; the **Twist Rig Toolbox** will be displayed, as shown in Figure 11-21. Also, you will be prompted to select the geometry, and the **Accept Targets** button in inactive state will be displayed at the lower right corner of the active window. Select the target from the active window; the **Accept Targets** button will get activated. Choose the **Accept Targets** button; you will be prompted to select the axis. Also, the **Accept Axis Curve** button in the inactive state will be displayed at the lower right corner of the active window. Select the curve that will act as an axis; the selected curve will be highlighted in yellow and the **Accept Axis Curve** button will get activated, as shown in Figure 11-22. Choose the **Accept Axis Curve** button; the target will be twisted about the selected axis and four twist handles will be displayed on it, as shown in Figure 11-23.

Figure 11-21** The **Twist Rig Toolbox

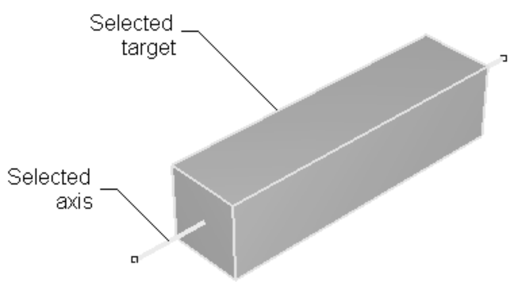


***Figure 11-22** Target and axis for the twist rig*

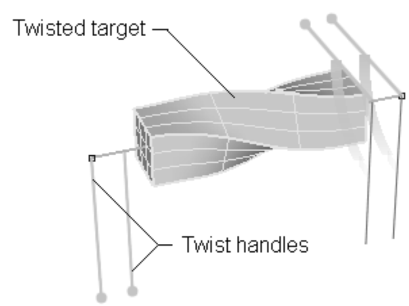


***Figure 11-23** Target twisted about the twist axis*

When you accept the axis to twist the object about, the target will be twisted by 90-degree in the counter-clockwise direction. Also, a yellow colored twist angle indicator will be displayed between the original and final positions of the twist handles. To change the twist angle, select the twist handle with the left mouse button; the color of the twist handle will change to white. Enter the angle measure in the promptline or drag the selected twist handle about an axis; the target will be twisted by the specified angle. Also, when you select any twist handle, the twist angle and the relative position of the twist handle will be displayed numerically on it. The twist angle is represented by the alphabet **T** followed by the angle value and the relative position of the handle is represented by the parametric value enclosed in a red box. You can change the position of the twist handle by dragging it vertically with the middle mouse button. Figure 11-24 shows the twisted model with the changed twist angle.

You can specify the twist angle in relative or absolute mode by entering **r** or **a** in the promptline. In absolute mode, the specified angle is assigned to all selected handles and the values of all multiple twist handles becomes same. But in case of relative mode, the specified angle is added to the twist angles of the handles and increases their values by the specified values. Figures 11-25 and 11-26 show the twisted model with multiple handles twisted with an angle of 30-degree in absolute and relative modes, respectively.

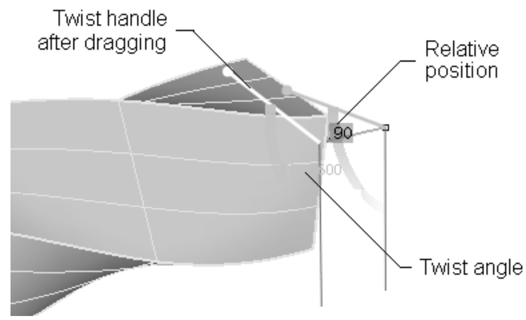


Figure 11-24 *Twisted model with the changed twist angle*

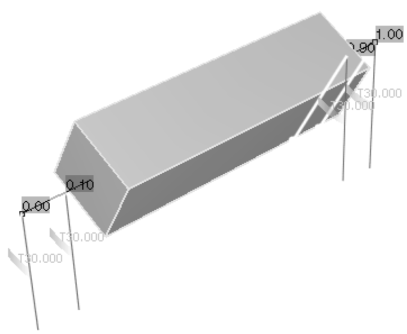


Figure 11-25 *Model twisted after specifying the angle in absolute mode*

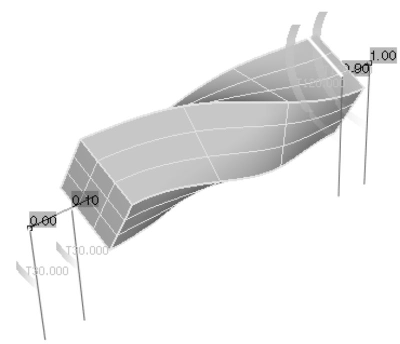


Figure 11-26 *Model twisted after specifying the angle in relative mode*

You can set different parameters of the **Twist Rig** tool. To do so, choose the **Accept Target**s button that is displayed when you select an object to be modified. Next, double-click on the **Twist Rig** button in the **Twist Rig Toolbox**; the **Twist Rig Options** dialog box will be displayed. Most of the options in this dialog box are the same as those discussed in the **Lattice Rig Options** dialog box. The rest of the options in this dialog box are discussed next.

Twist Rig Toolbox

When you choose the **Twist Rig** button from the **Object Edit** tab in the **Palette**; the **Twist Rig Toolbox** will be displayed, refer to Figure 11-21. Different buttons in the **Twist Rig Toolbox** are discussed next.

Twist Rig

This button is chosen by default and prompts you to select the geometry to be modified.

Add Twist Angle

When you choose the **Accept Curve Axis** button from the lower right corner of the active window, four twist handles will be displayed on the twist axis. You can add more twist handles to the selected axis. To do so, choose the **Add Twist Angle** button from the **Twist Rig Toolbox**; a new twist handle will be added to the axis. This twist handle will be positioned between a pair of existing handles with the largest separation. You can add a number of new twist handles to the axis. The new twist handles will have different twist angles. If you have selected some existing handles and add new handles to the twist axis, the selected handles will be deselected, thereby giving preference to new handles.

Remove Twist Angles

You can remove twist handles from a selected axis. To do so, select the twist handle(s) to be removed; the selected handle will be highlighted in white with the twist angle and the relative position displayed on it. Next, choose the **Remove Twist Angles** button from the **Twist Rig Toolbox**; the selected twist handle will be removed from the selected axis. Alternatively, select the handle and press the DELETE key.

Note

You cannot delete all twist handles. There must be atleast one twist handle in the rig.

The other buttons in the **Twist Rig Toolbox** are the same as those discussed in the **Transformer Rig Toolbox**.

Modifying an Object Using the Bend Rig Tool

Palette: Object Edit > Transformer Rig > Bend Rig

You can modify an object dynamically by bending it about an axis or a group of continuous curves. To do so, choose the **Bend Rig** button from the **Object Edit** tab in the **Palette**; the **Bend Rig Toolbox** will be displayed, as shown in Figure 11-27. Also, you will be prompted to select the geometry, and the **Accept Targets** button in the inactive state will be displayed at the lower right corner of the active window. Select the target from the active window; the **Accept Targets** button will get activated. Choose the **Accept Targets** button; you will be prompted to select the axis or a set of continuous curves that will act as bend axis. Also, the **Accept Axis Curve** button in the inactive state will be displayed at the lower right corner of the active window. Select the curve or the continuous curve; the selected curve(s) will be highlighted in yellow, as shown in Figure 11-28. Also, the **Accept Axis Curve** button will get activated. Choose the **Accept Axis Curve** button; the target will bend along the bend axis, as shown in Figure 11-29. The original geometry along with three buttons, **Translate**, **Rotate**, and **Scale** will be displayed at the lower right corner of the active window. By default, the **Translate** button is chosen. As a result, you will be prompted to enter the translation value or click-drag to slide the target along the axis. If you choose the **Rotate** button, you will be prompted to enter the rotation angle or click-drag to rotate the target about an axis. You can scale the target along different directions by choosing the **Scale** button. Click-drag the left mouse button to scale the target uniformly along all directions.

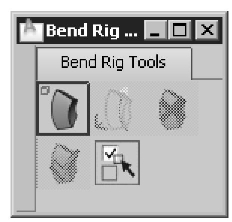


***Figure 11-27** The Bend Rig Toolbox*

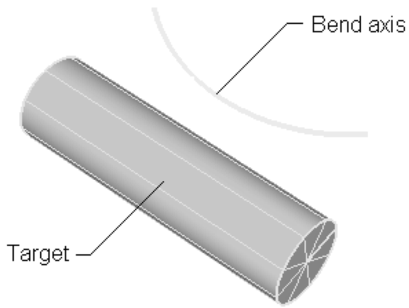


Figure 11-28 *Target and the bend axis*

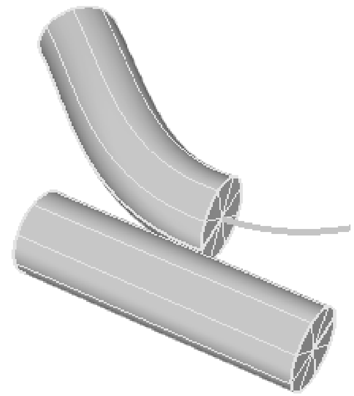


Figure 11-29 *Target bent about the bend axis*

Click-drag the middle mouse button to scale the target along the selected axis. If you click-drag the right mouse button, you can scale the target across its width. The length and width of the target will be displayed in the promptline and their values will keep on changing when you scale the target with any mouse button. Figure 11-30 shows a cube bent about the bend axis with the **Translate** button chosen. Figures 11-31, 11-32, and 11-33 show the same cube scaled by dragging the mouse with the left, middle, and right mouse buttons, respectively.

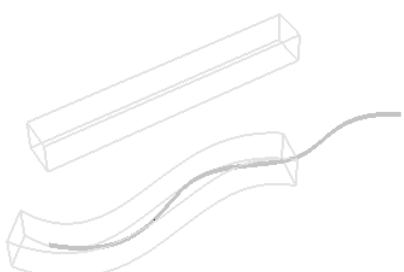


Figure 11-30 *Cube bent about an axis with the **Translate** button chosen*

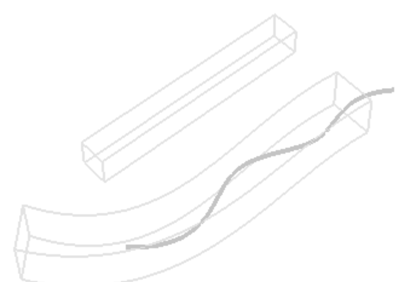


Figure 11-31 *Cube bent about an axis after scaling with the left mouse button*

You can set different parameters of the **Bend Rig** tool. To do so, choose the **Accept Target**s button that is displayed when you select an object for modification. Next, double-click on the **Bend Rig** button in the **Bend Rig Toolbox**; the **Bend Rig Options** dialog box will be displayed, as shown in Figure 11-34. Most of the options in this dialog box are the same as those discussed in the **Lattice Rig Options** dialog box. The rest of the options in this dialog box are discussed next.

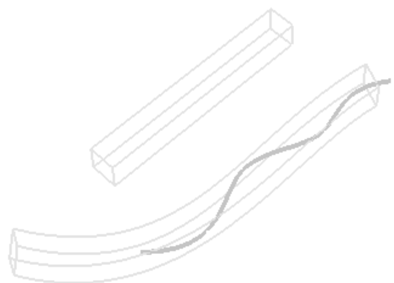


Figure 11-32 Cube bent about an axis after scaling with the middle mouse button

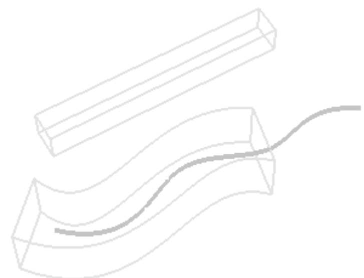


Figure 11-33 Cube bent about an axis after scaling with the right mouse button

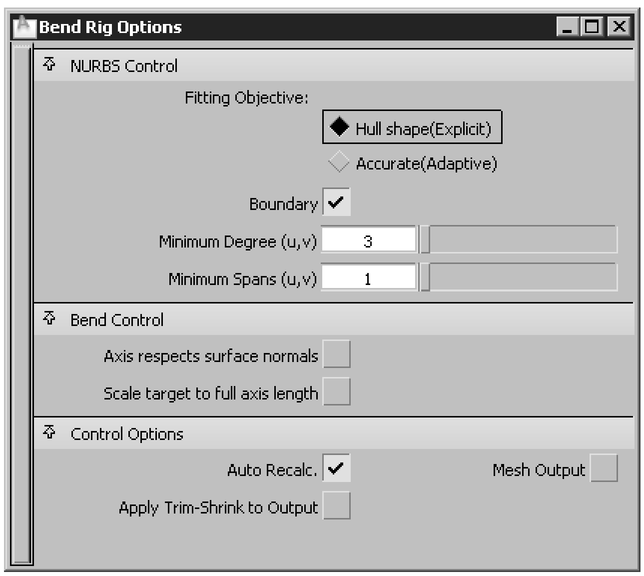


*Figure 11-34 The **Bend Rig Options** dialog box*

Bend Control

The **Bend Control** area of the **Bend Rig Options** dialog box is used to control the bending of the target. The options in this area are discussed next.

Axis respects surface normals

Sometimes, you may need to select a surface curve as a bend axis. In this case, if you select the **Axis respects surface normals** check box, the target geometry at a point will be bent according to the bend axis at that point. In other words, the **Bend Rig** tool is used to bend the target according to the normals of the bend axis. Figures 11-35 and 11-36 show the targets bent with the **Axis respects surface normals** check box cleared and selected, respectively.

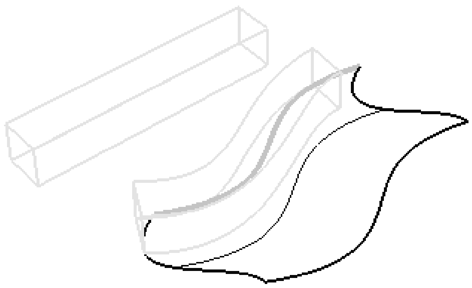


*Figure 11-35 Cube bent with the **Axis respects surface normals** check box cleared*

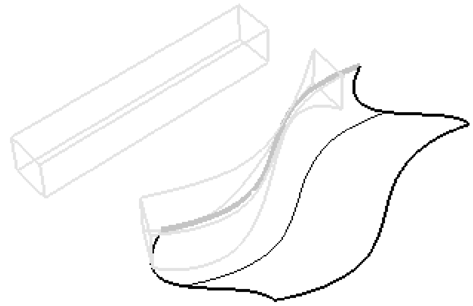


*Figure 11-36 Cube bent with the **Axis respects surface normals** check box selected*

Scale target to full axis length

By default, the **Bend Rig** tool bends the target to a certain length of the axis. If you select the **Scale target to full axis length** check box, the target will be bent according to the axis and will stretch or compress to whole length of the axis. Figures 11-37 and 11-38 show the targets bent with the **Scale target to full axis length** check box cleared and selected, respectively.

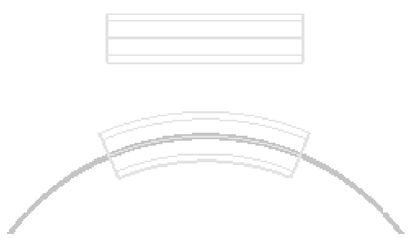


*Figure 11-37 Cylinder bent with the **Scale target to full axis length** check box cleared*

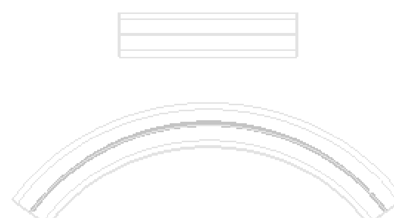


*Figure 11-38 Cylinder bent with the **Scale target to full axis length** check box selected*

Bend Rig Toolbox

When you choose the **Bend Rig** button from the **Object Edit** tab in the **Palette**; the **Bend Rig Toolbox** will be displayed, refer to Figure 11-27. The buttons in the **Bend Rig Toolbox** are discussed next.

Bend Rig

This button is chosen by default and prompts you to select the geometry to be modified.

AxisLine

When you select and accept the bend axis, the target bends about the selected bend axis with default orientation. If you choose the **AxisLine** button from the **Bend Rig Toolbox**, four buttons, **X**, **Y**, **Z**, and **Done** will be displayed at the lower right corner of the active window. Choose the **X**, **Y**, or **Z** button to set the orientation of the target along the corresponding direction. After setting the orientation of the target, choose the **Done** button to accept the orientation and proceed further. Figures 11-39 and 11-40 show the cylinders bent with the orientation set along the X and Z-axes, respectively.

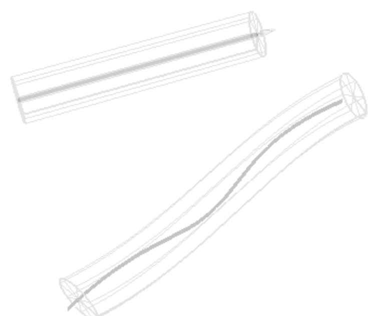


Figure 11-39 *Cylinder bent with the orientation set along the X-axis*

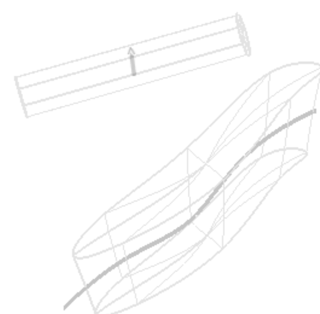


Figure 11-40 *Cylinder bent with the orientation set along the Z-axis*

The rest of the buttons in the **Bend Rig Toolbox** are the same as discussed in the **Transformer Rig Toolbox**.

Modifying an Object Using the Conform Rig Tool

Palette: Object Edit > Transformer Rig > Conform Rig

Figure 11-41 *The* ***Conform Rig Toolbox***

AliasStudio allows you to deform a flat surface such that it gets mapped on to a curved surface to conform to its shape. You can do so by using the **Conform Rig** tool. While using this tool, you need to select the target (object to be modified) and the destination (object to be conformed with). To deform or modify an object to conform to another object, choose the **Conform Rig** button from the **Object Edit** tab in the **Palette**; the **Conform Rig Toolbox** will be displayed, as shown in Figure 11-41. Also, you will be prompted to select the geometry to be added as targets, and the **Accept Targets** button in the inactive state will be displayed at the lower right corner of the active window. Select the required target; the **Accept Targets** button will get activated. Choose the **Accept Targets** button; the target will be highlighted in light green and you will be prompted to select the destination geometry of the conform rig. Also, the **Accept Conform Destination** button in the inactive state will be displayed at the lower right corner of the active window. Select the required destination geometry (surface, meshes, or surface curves); the **Accept Conform Destination** button will get activated. Choose the **Accept Conform Destination** button; the target will be mapped onto the destination geometry conforming to its shape.

Figure 11-42 shows the target geometry to conform to the destination geometry as well as the destination geometry and Figure 11-43 shows the conformed target. Also, the five buttons, **Translate**, **Rotate**, **Scale**, **Elevate**, and **Flip** will be displayed at the lower right corner of the active window. These buttons are discussed next.

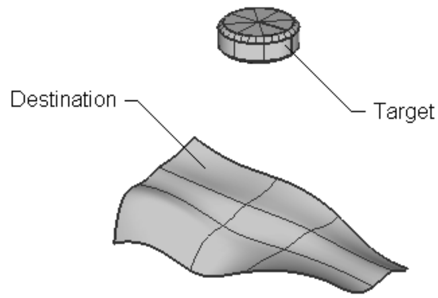


Figure 11-42 *Target and destination geometries*

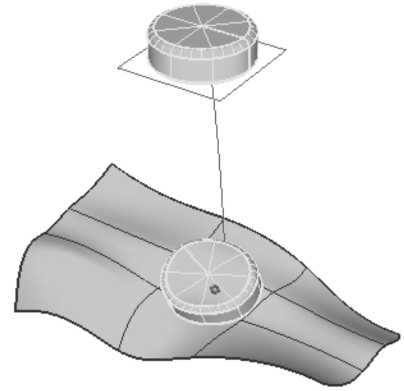


Figure 11-43 *Target conformed to destination*

Translate

The **Translate** button is used to change the position of the target geometry with respect to the destination geometry. Each target defines a point known as contact point that controls the translation of the target. When you set the conform rig, the contact point is positioned at the midpoint of the base of the bounding box of the target. If the destination consists of surfaces, mesh of surfaces, or a set of surfaces, the contact point will be positioned on the destination at any point. If the destination consists of a surface curve, the contact point will be constrained to move along the surface curve. You can also use the snapping tools (**Snap to CV/Edit Point**, **Snap to Grid**, and **Snap to Curve**) to snap the contact point to the required curve division, intersection, and so on. If you select a single surface point as destination, the target will not move. However, if you move this single surface point, the target will follow its movement. To change the position of the target, choose the **Translate** button; you will be prompted to slide the target along the surface. Click-drag the target along the destination; the proxy of the target being translated will be displayed, as shown in Figure 11-44. Release the mouse button at the desired location to translate; the target will be conformed to the destination, as shown in Figure 11-45.

Rotate

Choose the **Rotate** button to rotate the target about the contact point on the destination geometry. You can specify the rotation angle in the promptline or change it by clicking and dragging the mouse button. You can specify the rotation angle in absolute or relative mode. Figure 11-46 shows the target being rotated about the contact point.

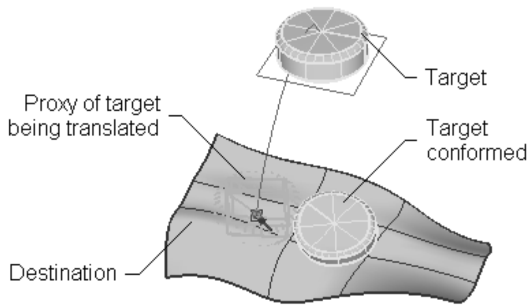


Figure 11-44 *Proxy of target being translated*

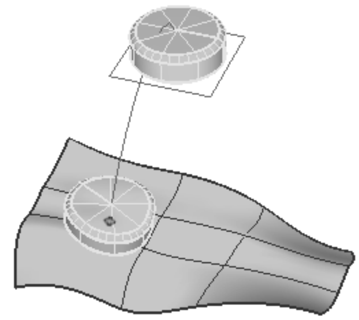


Figure 11-45 *Target conformed to the destination*

Scale

You can scale the target with respect to the destination. To do so, choose the **Scale** button from the lower right corner of the active window. The target will be scaled using the contact point as the scale pivot. You can scale the target uniformly or nonuniformly. Click-drag the left mouse button to scale the target uniformly, as shown in Figure 11-47. Click-drag the middle mouse button to scale the target nonuniformly, so that the size of the target increases without changing its thickness, as shown in Figure 11-48. Click-drag the right mouse button to scale the target nonuniformly, that is with varied thickness, as shown in Figure 11-49.

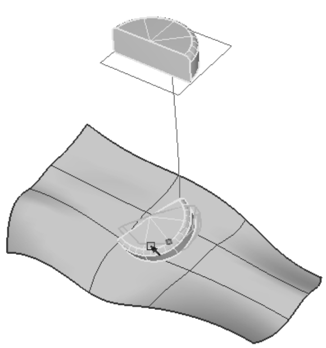


Figure 11-46 *Target being rotated about the contact point*

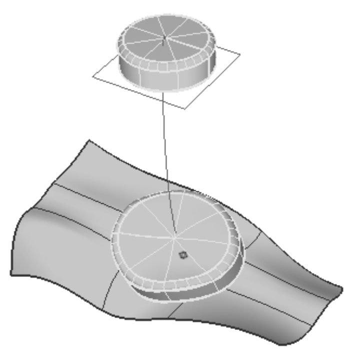


Figure 11-47 *Target scaled uniformly by using the left mouse button*

Elevate

You can move the target away from or toward the destination. You can also lower or sink the target into the destination surface. To move the target, choose the **Elevate** button from the lower right corner of the active window and click-drag the mouse button toward or away from the destination. You can also specify the sinking or elevation value in the promptline in absolute or relative mode. Figures 11-50, 11-51, and 11-52 show the target in normal, elevated, and sinking positions, respectively.

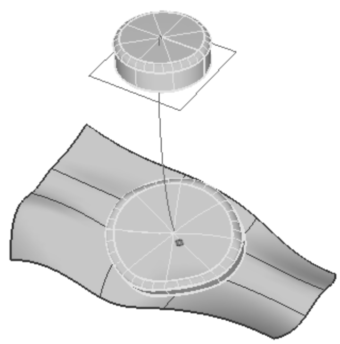


Figure 11-48 *Target scaled nonuniformly by using the middle mouse button*

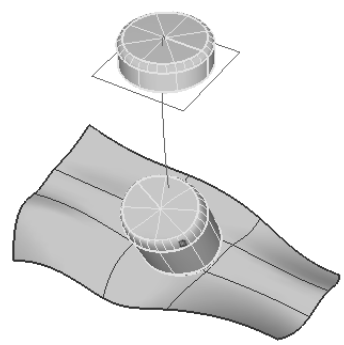


Figure 11-49 *Target scaled by using the right mouse button*

Flip

If you use the **Conform Rig** tool initially, the target will conform to the outward-facing side of the destination surface. You can change the direction of the target by choosing the **Flip** button from the **Conform Rig Toolbox**, as shown in Figure 11-53. Alternatively, you can reverse the orientation of the destination surface by using the **Reverse surface orientation** tool of the **Surface Edit** tab in the **Palette**.

Figure 11-50 *Target in normal position*

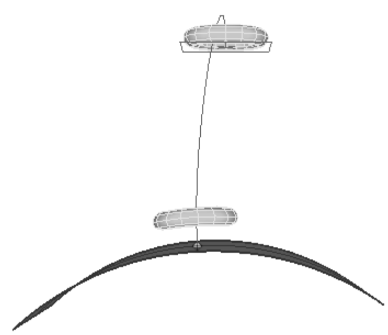


Figure 11-51 *Target in elevated position*

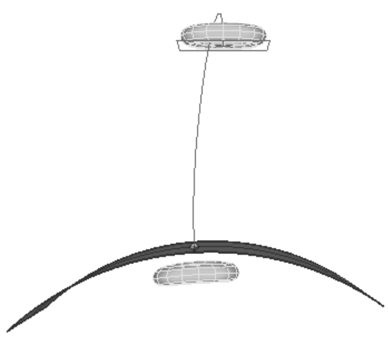


Figure 11-52 *Target in sinking position*

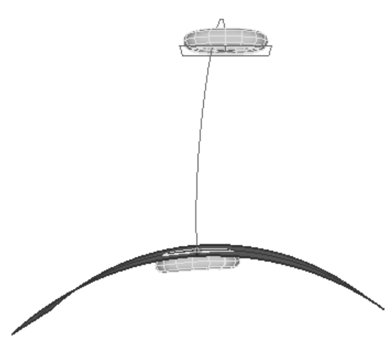


Figure 11-53 *Flipped target*

You can set different parameters of the **Conform Rig** tool. To do so, choose the **Accept Target**s button that is displayed when you select an object for modification. Next, double-click on the **Conform Rig** button in the **Conform Rig Toolbox**; the **Conform Rig Options** dialog box will be displayed, as shown in Figure 11-54. Most of the options in this dialog box are the same as those discussed in the **Lattice Rig Options** dialog box. The rest of the options in this dialog box are discussed next.

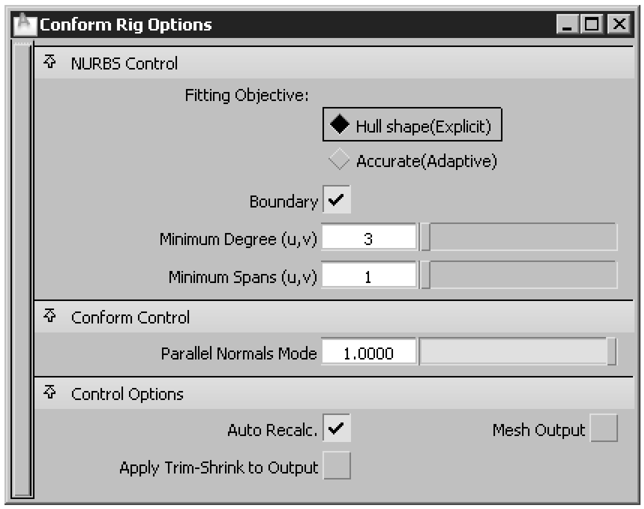


Figure 11-54 *The **Conform Rig Options** dialog box*

Conform Control

The **Conform Control** area is used to control the mapping of the target on the destination geometry. The edit box in this area is discussed next.

Parallel Normals Mode

This edit box is used to determine the condition of the edges of the target when it is mapped on to the destination. The target are mapped based on the normals of the destination surface. By default, 1 is displayed in the **Parallel Normals Mode** edit box. With this default value, normals of the destination surface at the center of the target will be used for mapping. As a result, the sides of the target will be parallel, as shown in Figure 11-55. If you enter 0 in this edit box, all local normals of the destination will be used for mapping the target. Therefore, the sides of the target will be oriented along the normals of the destination geometry, as shown in Figure 11-56.

Conform Rig Toolbox

When you choose the **Conform Rig** button from the **Object Edit** tab in the **Palette**; the **Conform Rig Toolbox** will be displayed, refer to Figure 11-41. The buttons in the **Conform Rig Toolbox** are discussed next.

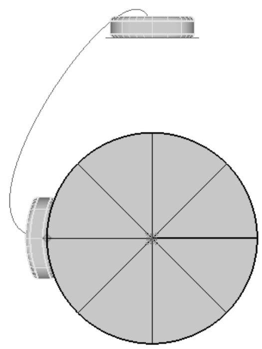


Figure 11-55 *Sides of the target constrained to be parallel*

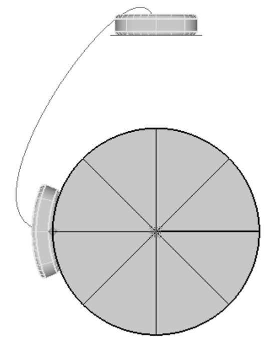


Figure 11-56 *Sides of the target oriented along normals*

Conform Rig

This button is chosen by default and prompts you to select the geometry to be modified.

Change Contact Point

As discussed earlier, the contact point acts as pivot for scaling and rotating the target. By default, the contact point is positioned at the center of the target. You can move the target to the required position by using the contact point. To do so, choose the **Change Contact Point** button from the **Conform Rig Toolbox**; manipulators will be displayed on the target, as shown in Figure 11-57. Also, the two buttons, **Reset** and **Done** will be displayed at the lower right corner of the active window. You can use the manipulator handles to move the contact point relative to the target. Changing the position of the contact point updates the position of the mapped target. You can also snap the contact point on the original object to a particular point on the original target by using the manipulator. The use of manipulator in the **Conform Rig** tool is similar to the use of manipulator in the **Plane** tool of the **Construction** tab in the **Palette**. After moving the contact point, choose the **Done** button; the position of the mapped target will be changed. To restore the contact point to its original position, choose the **Reset** button from the lower right corner of the active window. Also, by default, the reference plane is oriented parallel to XY plane, refer to Figure 11-57. You can also change the orientation of the reference plane. To do so, select the required rotation handle and drag it to the required position, as shown in Figure 11-58.

The rest of the buttons in the **Conform Rig Toolbox** are the same as those discussed in the **Transformer Rig Toolbox**.

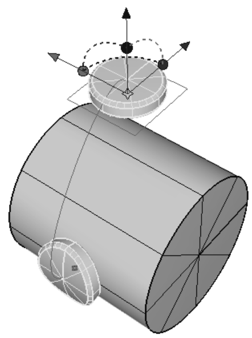


Figure 11-57 *Default orientation of the reference plane*

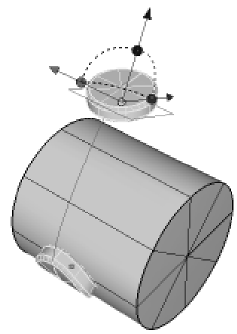


Figure 11-58 *Changed orientation of the reference plane*

INTRODUCTION TO MESHES

A mesh is a large polygonal object that is created when you scan or digitize a physical model to create data model. A mesh consists of sets of polygons of triangular or quadrilateral shape. These polygonal shapes are joined with each other at vertices, known as nodes. Due to the polygonal structure of the meshes, they are preferred for storing large and detailed data of physical objects. You can separate meshes and shade them as well. You can also delete meshes or turn their visibility off. Meshes cannot be built directly in AliasStudio. You can only retrieve meshes by opening an existing file containing them. You can also obtain meshes by converting polysets or NURBS surfaces to meshes. In AliasStudio, meshes are represented by brown triangular polygons, as shown in Figure 11-59.

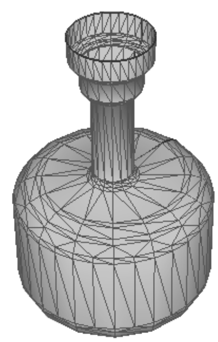


Figure 11-59 *Mesh generated on surface*

Generating Meshes

As mentioned earlier, meshes cannot be built directly in AliasStudio. You can either retrieve the existing meshes or convert polysets or NURBS surfaces to meshes. The methods of generating meshes are discussed next.

Retrieving an Existing Mesh

You can retrieve a mesh from a file containing polygonal data (existing mesh). The types of files used to retrieve meshes into AliasStudio are wire, STL, and OBJ. The wire file is the most efficient format for storing meshes.

To retrieve meshes from an existing wire file, choose **File > Open** from the menu bar; the **Open** dialog box will be displayed. Select the required wire file from the list box given below the **Look in** area and choose the **Open** button; the mesh will be opened in AliasStudio. Note that the wire files to be used for retrieving meshes need to contain meshes.

To retrieve meshes from files with STL or OBJ format, choose **File > Import > File** from the menu bar; the **Import File** dialog box will be displayed. Select the required wire file from the list box given below the **Look in** area and choose the **Open** button; the mesh will be opened. The polygonal models created in Spider or EvalViewer will be imported as meshes automatically.

Creating Meshes from NURBS or Shells

You can create meshes from NURBS or shells. To do so, select the required NURBS or shell from the active window and then choose the **Nurbs to mesh** button from the **Mesh** tab in the **Palette**; the mesh will be created.

Creating Meshes from Displacement Map

A displacement map is used to make a surface rough or bumpy by changing the tessellated triangle vertices while rendering. To create meshes from displacement map, first assign the displacement map to the shader and then apply this shader to the desired object or surface. Next, select the required objects having the displacement maps and then choose the **Displacement Map to mesh** button from the **Mesh** tab in the **Palette**; the mesh will be generated. You can control the meshing display by adjusting the tessellation, which will be discussed in Chapter 14.

Saving Meshes

After generating meshes, you can save them for further use. AliasStudio provides you a number of ways to save meshes. You can save meshes in wire, STL, or OBJ file format. Various methods of saving meshes in these formats are discussed next.

Saving Meshes in Wire File Format

To save the generated meshes in the wire file format, choose **File > Save** from the menu bar; the **Save Wire** dialog box will be displayed. Specify the location and name of the file in the **Save in** drop-down list and the **Object name** edit box, respectively. Next, choose the **Save** button; the meshes will be saved in the wire file format.

Saving Meshes in STL Format

To save the generated meshes in the STL format, select the meshes and then choose **File > Export > STL** from the menu bar; the **Accept** button will be displayed at the lower right corner of the active window. Choose the **Accept** button; the **Export STL** dialog box will be displayed. Specify the location and name of the file and then choose the **Save** button; the meshes will be saved in the STL format.

Saving Meshes in OBJ Format

You can save the generated meshes in the OBJ file format. To do so, select the meshes and then choose **File** from the menu bar to invoke the flyout. Click on the box given on the right of the **Save as** option in the flyout; the **Save All Options** dialog box will be displayed. Next, press and hold the **WIRE** button from the **File Formats** option of the **Basic Save Options** area; a flyout will be displayed. Choose the **OBJ** button from this flyout and then choose the **Save** button; the **Save OBJ** dialog box will be displayed. Specify the location and name of the file and then choose the **Save** button; the meshes will be saved in the OBJ format.

WORKING ON MESHES

Like other objects, meshes can be edited, trimmed, cut, merged. You can also repair the meshes, reverse their orientation, and project curves on them. The different operations that are performed on meshes are discussed next.

Creating Meshes from NURBS or Shells

Palette:	Mesh > Nurbs to mesh

To create meshes from NURBS or shells, select the surface or shell from the active window. Next, choose the **Nurbs to mesh** button from the **Mesh** tab in the **Palette,** the mesh will be created on the selected surface or shell. You can hide the original surface or shell to view the mesh properly by selecting the surface or shell and then choosing **ObjectDisplay > Invisible** from the menu bar. Figures 11-60 and 11-61 show the NURBS surface and the mesh generated, respectively.

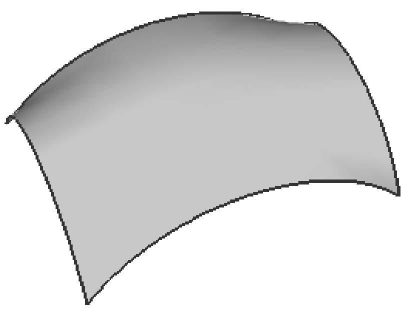


Figure 11-60 *NURBS surface*

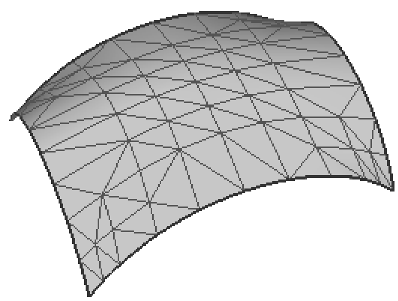


Figure 11-61 *Mesh generated on NURBS surface*

Note

*You can pick meshes by choosing the **Mesh** option from the Marking Menus that are displayed on clicking the left mouse button with SHIFT+CTRL keys pressed.*

To set the parameters of the **Nurbs to mesh** tool, double-click on the **Nurbs to mesh** button; the **Nurbs To Mesh Options** dialog box will be displayed, as shown in Figure 11-62.

The options in this dialog box are discussed next.

Use existing tessellation

Select this check box to specify the tessellation to be used for the meshes generated. By default, this check box is cleared. As a result, the tessellation will be recalculated based on the options set in the **Nurbs to Mesh Options** dialog box. Select this check box to use the existing tessellation. Tessellation exists if you have shaded the NURBS surface or shell by hardware shading. Note that the other options of the **Nurbs to Mesh Options** dialog box will not be displayed if you select the **Use existing tessellation** check box.

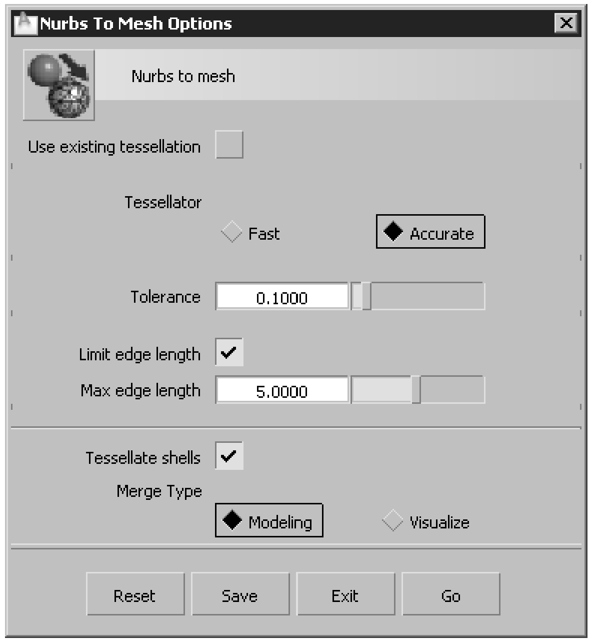


Figure 11-62 *The* ***Nurbs To Mesh Options*** *dialog box*

Tessellator

The **Tessellator** area is used to specify the accuracy in the tessellation of the generated meshes. The radio buttons in this area are discussed next.

Fast

Select this radio button to tessellate the meshes quickly. In this case, tessellation will be less accurate.

Accurate

This radio button is selected by default and is used to tessellate the meshes slowly. In this case, tessellation will be more accurate.

Tolerance

The **Tolerance** edit box is used to specify the accuracy of tessellation for the generated mesh. The default value in this edit box is 0.1 and the range of the tolerance value is 0.0001 to 1.0. You can also change the tolerance value by using the slider bar given on the right of this edit box. The tessellation quality obtained from this tolerance is the same as that of the hardware shading.

Limit edge length

This check box is selected by default and is used to control the size of triangles in meshes. This check box will be available only when the **Accurate** radio button is selected from the **Tessellator** area.

Max edge length

This edit box will be available only when you select the **Limit edge length** check box. This edit box is used to increase or decrease the size of triangles in meshes. Enter a smaller value in the **Max edge length** edit box to increase the number of triangles in the mesh. You can also change the size of triangles by using the slider bar given on the right of this edit box. Figures 11-63 and 11-64 show the meshes with small and large size of triangles, respectively.

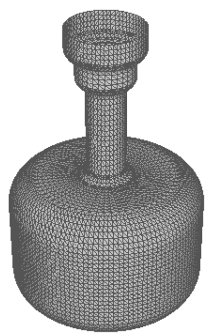


Figure 11-63 *Mesh with small triangles*

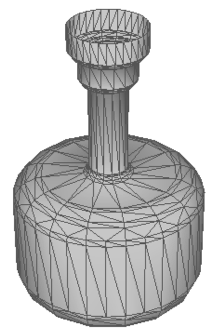


Figure 11-64 *Mesh with large triangles*

Tessellate shells

This check box is selected by default and is used to convert shells to meshes without any gap or T-edge. This is very important for producing better results for visualization and conversion to the STL format.

Merge Type

This area will be available only when you select the **Tessellate shells** check box. This area is used to control the merger of the coincident vertices of the meshes. A mesh is represented by a set of triangles whose vertices may or may not be merged. Also, each vertex of the triangle has a surface normal. The surface normal ensures the realistic shading of object while rendering. The radio buttons in this area are discussed next.

Modeling

This radio button is selected by default and is used to merge the coincident vertices of the mesh in shells. In this case, normals are placed at the average position between the merged vertices. This method is recommended for the purpose of modeling. In this type of merger, the size of the file decreases.

Visualize

Select this radio button to merge the coincident vertices of the triangles only if triangles have coincident normals. This method is recommended for shells for visualization purposes.

Projecting the Curve on the Mesh

Palette:	Mesh > Mesh project curve

Like generating curves-on-surface, you can generate curves-on-mesh by projecting curves on a mesh. These curves-on-mesh help separate, merge, or cut meshes. To project a curve on a mesh, choose the **Mesh project curve** button from the **Mesh** tab in the **Palette**; you will be prompted to select the meshes on which the curve has to be projected. Also, the **Go** button in the inactive state will be displayed at the lower right corner of the active window. Select the required mesh; the **Go** button will be activated. Choose the **Go** button; you will be prompted to select the curve to project or the projection vector. Also, the **Go** button will get deactivated. Select the curve(s) as well as the projection vector; the **Go** button will get activated again. Choose the **Go** button; the selected curve(s) will be projected on the mesh. In this case, the curve-on-mesh generated is one degree NURBS curve. If you modify the original curve, the curve-on-mesh will get updated. Figure 11-65 shows the mesh, projection vector, and the curve to be projected on it. Figure 11-66 shows the curve-on-mesh generated after projecting the curve.

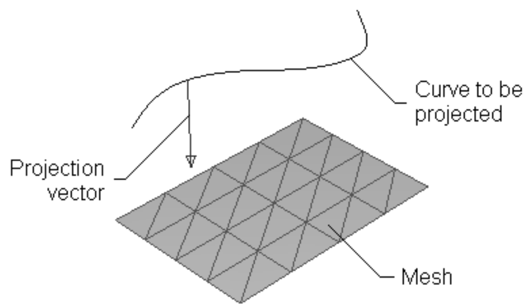


Figure 11-65 *Mesh, projection vector, and the curve to be projected*

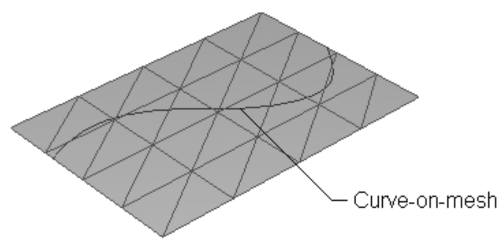
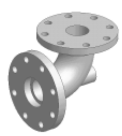


Figure 11-66 *Curve-on-mesh generated*



Tip: *To generate curve-on-mesh without selecting the projection vector, you need to select the curve to be projected in such a way that it is perpendicular to the projection direction.*

Projecting the Curve Normal to the Mesh

Palette:	Mesh > Mesh project curve > Mesh project normal

You can also project a curve normal to a mesh without using the projection vector. To do so, choose the **Mesh project normal** button from the **Mesh** tab in the **Palette**; you will be prompted to select the meshes on which the curve has to be projected normally. Also, the **Go** button in the inactive state will be displayed at the lower right corner of the active window. Select the required mesh; the **Go** button will get activated. Choose the **Go** button; you will be prompted to select the curves to project. Also, the **Go** button will get deactivated. Select the curve; the **Go** button will get activated again. Choose the **Go** button; the selected curve will get projected on the mesh, resulting in the generation of mesh-on-curve.

Creating Curves on Mesh Boundaries

Palette: Mesh > Mesh project curve > Mesh Boundaries

AliasStudio allows you to create the degree one NURBS curve on the boundaries of the mesh. Like keypoint or edit point curves, these curves can be used to create surfaces. To create curves on mesh boundaries, choose the **Mesh Boundaries** button from the **Mesh** tab in the **Palette**; you will be prompted to pick a mesh with boundaries to extract. Also, the two buttons, **Extract** and **Extract All** in the inactive state will be displayed at the lower right corner of the active window. Select the mesh; the boundary of the mesh will be highlighted in green. Also, you are prompted to select the boundary from the mesh. Select the mesh boundary from which you want to extract the curve; the selected mesh boundary will get highlighted in yellow. Also, the **Extract** and **Extract All** buttons will get activated. If you select a single boundary from the mesh, choose the **Extract** button to extract the selected mesh boundary. If you select multiple mesh boundaries, choose the **Extract All** button to extract all mesh boundaries. After choosing the required button, the mesh boundary will be created. Figure 11-67 shows the mesh with three boundaries and Figure 11-68 shows the curves extracted from these boundaries.

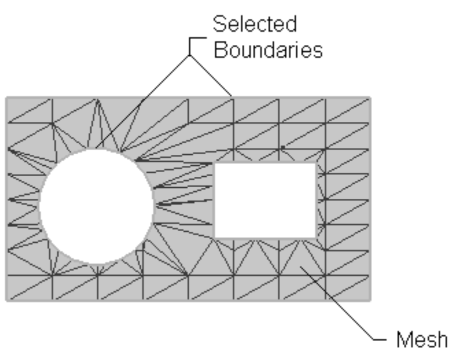


Figure 11-67 *Mesh with three boundaries*

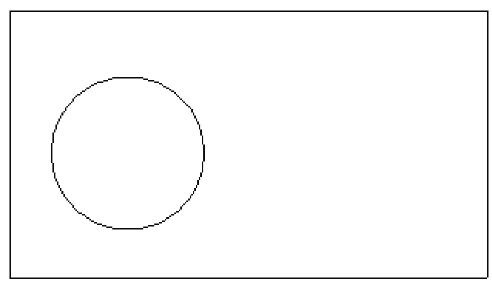


Figure 11-68 *Curves extracted from the mesh boundaries*

Note

Boundaries are extracted from the position of the existing mesh boundaries. To view these extracted boundaries, move the mesh away from mesh boundaries. Alternatively, create a new layer and then assign the mesh to the layer. Next, turn the visibility of the layer off. Figure 11-68 shows the curves extracted after turning the display of the mesh off.

Cutting Meshes

Palette: Mesh > Mesh subset > Mesh cut

The **Mesh cut** tool is used to cut a mesh into several components. A component is defined as a mesh that consists of a number of connected triangles. To cut a mesh, choose the **Mesh cut** button from the **Mesh** tab in the **Palette**; you will be prompted to select the mesh to cut. Also, the **Cut** button in the inactive state will be displayed at the lower right corner of the active window. Select the required mesh; you will be prompted to select the curve-on-mesh. Select the curve-on-mesh; the triangles of the mesh that will be affected by cutting will be highlighted in sky blue color and the **Cut** button will get activated. Choose the **Cut** button; the mesh will be cut into different components with the changed orientation of triangles, and three buttons, **Keep**, **Discard**, and **Divide** in the inactive state will be displayed at the lower right corner of the active window. The promptline informs you about the creation of different components and prompts you to select components to subset meshes. Select the required component from the mesh to subset; the three buttons, **Keep**, **Discard**, and **Divide** will get activated. Choose the **Keep** or **Discard** button to retain or remove the selected mesh components, respectively. Choose the **Divide** button to divide the mesh without removing any component. Figure 11-69 shows the mesh and the curve-on-mesh and Figure 11-70 shows the mesh cut along the curve-on-mesh.

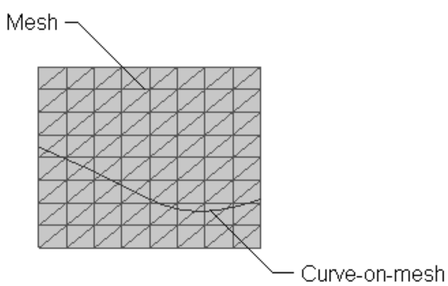


Figure 11-69 *Curve-on-mesh generated on a mesh*

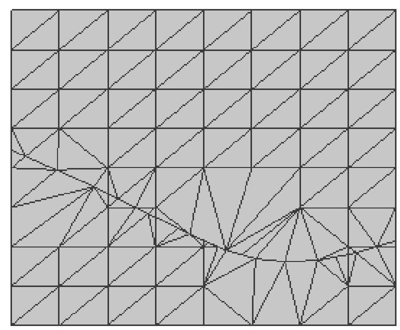


Figure 11-70 *Mesh cut along the curve-on-mesh*

Note

For cutting a mesh, make sure the projected curve-on-mesh divides the mesh into more than one component or creates a closed boundary on the mesh.

Tip: *The **Keep**, **Discard**, and **Divide** buttons are used to retain, remove, or divide the meshes, respectively. These buttons function the way as the buttons of the **Trim** tool. You can exit the **Mesh cut** tool and perform the required operations later on by using the other tools of the **Mesh** tab. These tools will be discussed later in this chapter.*

To set the parameters of the **Mesh cut** tool, double-click on the **Mesh cut** button; the **Mesh Cut Options** dialog box will be displayed, as shown in Figure 11-71.

The option in this dialog box is discussed next.

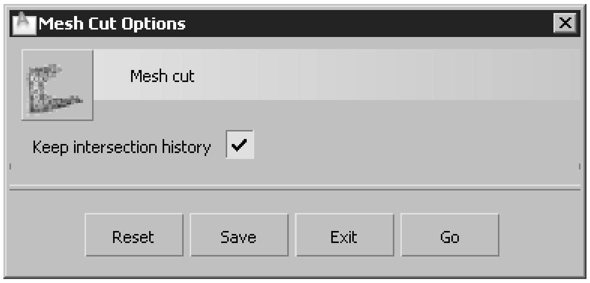


Figure 11-71 *The* ***Mesh Cut Options*** *dialog box*

Keep intersection history

This check box is selected by default and is used to retain construction history when meshes are intersected, cut, or subset. If you modify the objects (mesh-on-curve) that are involved in intersecting, cutting, or subsetting the mesh, the mesh will get updated accordingly.

Subsetting of Meshes

Palette: Mesh > Mesh subset

You can separate a mesh into different parts that can be deleted or toggled between the visibility modes to reduce the processing time. Such type of separation of the meshes is known as subsetting of meshes. Subsetting a mesh allows you to shade parts of the mesh separately. To subset a mesh, choose the **Mesh subset** button from the **Mesh** tab in the **Palette**; you will be prompted to select a mesh. Also, the seven buttons, **Select**, **Grow Once**, **Grow All**, **Invert**, **Hide**, **Delete**, and **Subset** will be displayed at the bottom of the active window. Select the required mesh; you will be prompted to define the region to subset. Click on the mesh to define the region to subset. You need to specify atleast three points to define a region. After defining the region, the **Select** button will get activated. Choose the **Select** button; the defined region will be highlighted in light blue and all other buttons at the bottom of the active window will get activated. Choose the **Grow Once** button to add one more row of triangles on each side of the defined region. Choose the **Grow All** button to add all triangles of the mesh to the defined region, thus overlapping it. Choose the **Invert** button to invert the selection of triangles. Choose the **Hide** button to turn the visibility of the defined region off. As the region gets hidden, the **Select**, **Grow Once**, **Grow All**, and **Invert** buttons will get deactivated and the **Unhide** button will replace the **Hide** button. Choose the **Unhide** button to turn the display of the selected region on. After defining the region for the subset, choose the **Subset** button; the subset of the mesh will be created. You can select this subset and then assign the required shader or the material properties to it, separately. You can also delete the defined region by choosing the **Delete** button. Figure 11-72 shows the region defined on the mesh to subset and Figure 11-73 shows the subset mesh with different shades applied to it.

To set the parameters of the **Mesh subset** tool, double-click on the **Mesh subset** button; the **Mesh subset options** dialog box will be displayed, as shown in Figure 11-74.

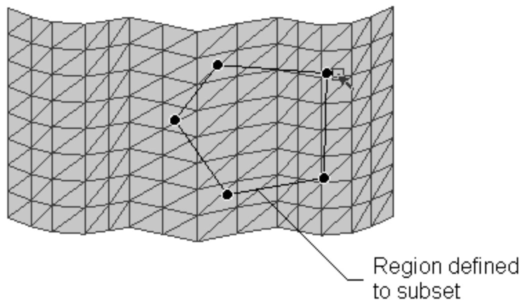


Figure 11-72 *Region defined to subset*

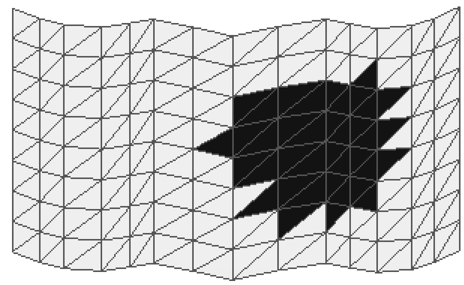


Figure 11-73 *Different shades applied to subset*

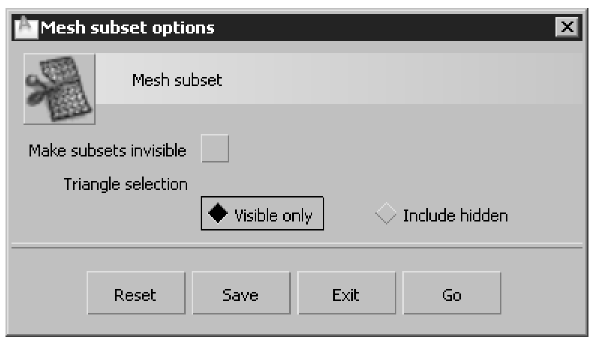


Figure 11-74 *The **Mesh subset options** dialog box*

The options in this dialog box are discussed next.

Make subsets invisible

Select this check box to turn the visibility of the subsets off created in the mesh.

Triangle selection

This area is used to specify whether or the triangles of the defined region on the hidden side of the mesh will be selected or not. This area is significant in case of three dimensional meshes. The radio buttons in this area are discussed next.

Visible only

This radio button is selected by default and enables you to select the triangles on the visible face of the mesh, that is the front face only.

Include hidden

Select this radio button to select the triangles on the visible as well as the hidden face of the mesh, that is the front and back faces of the mesh.

Merging Meshes

Palette:	Mesh > Mesh subset > Mesh merge

You can merge small meshes into a large mesh. To do so, choose the **Mesh merge** button from the **Mesh** tab in the **Palette**; you will be prompted to select the meshes to be merged. Also, the **Merge** button in the inactive state will be displayed at the lower right corner of the active window. Select the meshes to be merged; the **Merge** button will get activated. Choose the **Merge** button; the meshes will be merged into single mesh. Exit the **Merge** tool; else you will be prompted to merge more meshes.

Reversing the Orientation of Mesh

Palette:	Mesh > Reverse mesh orientation

Like surfaces, you can reverse the orientation of meshes. By reversing the orientation, you can reverse the direction of normals of the entire mesh or the selected mesh component. To reverse the orientation, choose the **Reverse mesh orientation** button from the **Mesh** tab in the **Palette**; the default orientation of the mesh will be displayed, as shown in Figure 11-75. Also, you will be prompted to click on a mesh component and two buttons, **Reverse Component**, and **Reverse Mesh** in the inactive state will be displayed at the lower right corner of the active window. Click on the mesh component whose orientation needs to be reversed; the **Reverse Mesh** button will get activated. Choose the **Reverse Mesh** button; the orientation of the selected mesh component will be reversed. If you have cut a mesh into different components, the **Reverse Component** button also will get activated. Select the required mesh component and then choose the **Reverse Component** button; the orientation of the selected mesh component will be reversed, as shown in Figure 11-76. When you reverse the orientation of a mesh, the order of vertices of the component will also be reversed, thereby reversing the direction of the normal. If you reverse entire mesh, only the flag value (boolean value) will be changed.

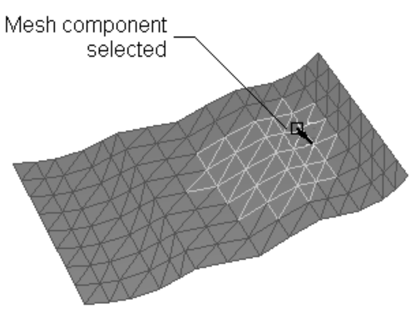


Figure 11-75 *Mesh component selected for reversing the orientation*

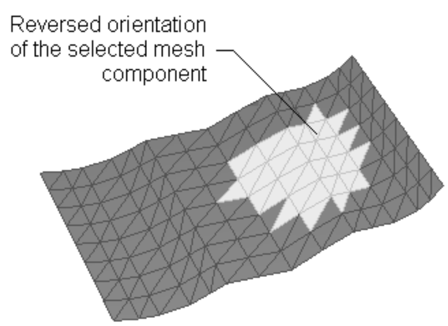


Figure 11-76 *Mesh component after reversing the orientation*

DEFECTS IN MESHES

Some of the operations performed on meshes may cause defects in them. These defects make it difficult to work with meshes by using other meshing tools. Different defects in meshes are discussed next.

Degenerate

This defect arises out when the mesh contains duplicate triangles or triangles with two or three overlapping sides.

Non-Manifold

In this defect, a vertex is shared by more than two boundary edges. Also, an edge is shared by more than two triangles.

Non-Oriented

In this defect, the winding (order) of vertices around triangles is done in such a way that the orientation of normals on a mesh is different.

Self-Intersecting

In this defect, the mesh intersects itself.

Folded-Edges

In this defect, the angle between the adjacent pair of triangles is less than the specified angle.

REPAIRING MESHES

Palette:	Mesh > Mesh Repair

The meshes with the above defects make it difficult to work with other meshing tools. You need to repair these meshes by repairing or removing defective triangles. To repair meshes, choose the **Mesh Repair** button from the **Mesh** tab in the **Palette**; you will be prompted to select the mesh to repair. Also, the **Repair** and **Repair All** buttons in the inactive state will be displayed at the lower right corner of the active window. Select the mesh to be repaired; defects in mesh, if present, will be displayed, as shown in Figure 11-77. The boundaries of the mesh will be highlighted in red and the arrows will point to the defects in the mesh. Also, the two buttons, **Repair** and **Repair All** will be activated. The promptline also displays the number of defects in the mesh and prompts you to repair them. Choose the **Repair** button successively till all defects in mesh get repaired, as shown in Figure 11-78. Alternatively, you can repair all defects in one attempt by choosing the **Repair All** button.

Note

By default, the arrows pointing to defects will not be displayed on boundaries. You need to select the ***Show boundaries with arrows*** *check box in the* ***Display*** *area of the* ***Mesh Repair Control*** *dialog box that will be discussed in the next topic.*

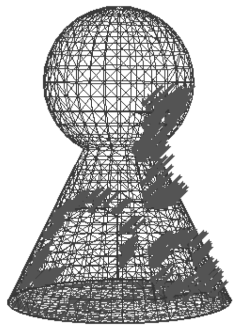


Figure 11-77 *Defective mesh*

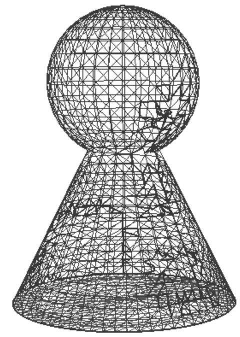


Figure 11-78 *Repaired mesh*

To view possible defects in a mesh and the status of repairing, double-click on the **Mesh Repair** button; the **Mesh Repair Control** dialog box will be displayed, as shown in Figure 11-79. This dialog box provides information about meshes and the status of the tests carried out by the **Mesh Repair** tool.

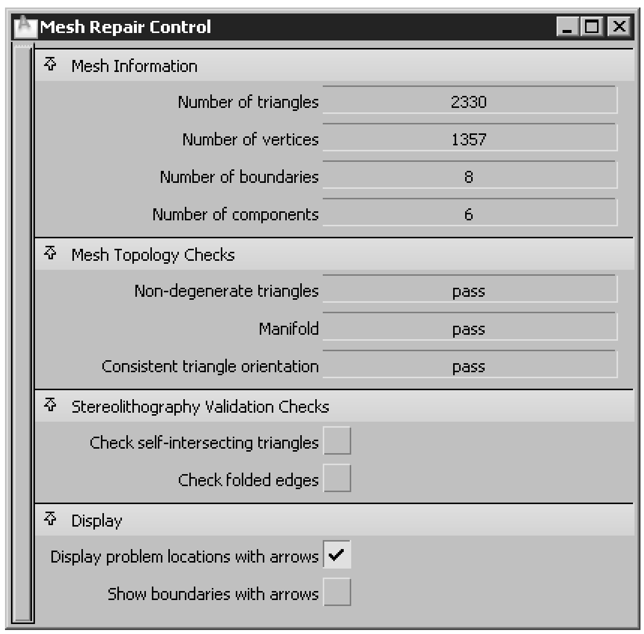


Figure 11-79 *The* ***Mesh Repair Control*** *dialog box*

The options in this dialog box are discussed next.

Mesh Information

This area informs you about the count of different parts of the mesh such as triangles, vertices, boundaries, and so on.

Mesh Topology Checks

This area informs you about the status of the tests carried out by the **Mesh Repair** tool for repairing mesh. These tests need to be passed by the mesh in order to be worked upon by other meshing tools. The **Mesh Repair** tool analyzes the defects in mesh and then repairs them one after the other. The different defects to pass though these tests are degenerated triangles, non-manifold vertices or edges, and non-oriented normals. The status of these tests, whether **'pass'** or **'fail'** will be displayed in the display boxes. If the **Mesh repair** tool fails to diagnose the status, it will display '**unknown**' in the display box.

Stereolithography Validation Checks

The checks/tests in this area need to be passed only for creating the dataset required for stereolithographic processing. These tests include checking self-intersecting triangles, folded edges, and minimum edge that are discussed next.

Check self-intersecting triangles

Select this check box to know the status of the self-intersecting triangles test. The status of this test will be displayed in the **No self-intersecting triangles** display box.

Check folded edges

Select this check box to know the status of the folded edges test. The status of this test will be displayed in the **No folded edges** display box.

Minimum edge angle

The **Minimum edge angle** edit box will be displayed only when you select the **Check folded edges** check box. The **Minimum edge angle edit** box is used to specify the minimum angle that is allowed between the planes of the two triangles sharing a common edge. By default, 15 is displayed in this edit box. You can change this angle by entering a new value in the edit box or by using the slider bar given on the right of this edit box. If the angle between the planes of the two triangles is less than the angle specified in this edit box, the triangles will be considered to be folded.

Display

This area is used to control the display of defects in the mesh. The options in this area are discussed next.

Display problem locations with arrows

This check box is selected by default and is used to display arrows in the active window that point toward the troublesome or problematic triangles of the mesh.

Show boundaries with arrows

Select this check box to display the arrows in each boundary at the end of each repairing process. This enables you to identify the boundaries that are small and hard to detect.

TUTORIALS

Tutorial 1

In this tutorial, you will modify the model created in Tutorial 2 of Chapter 6. The model before and after modification is shown in Figures 11-80 and 11-81. After modifying the model, you will create the mesh of the modified model and save it in the STL format.

(Expected time: 30 min)

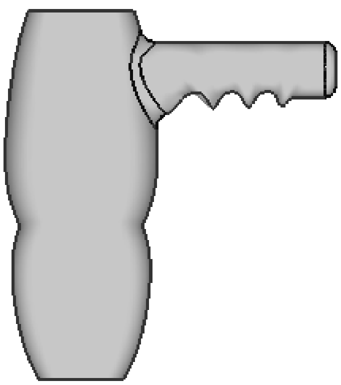


Figure 11-80 *Model before modification*

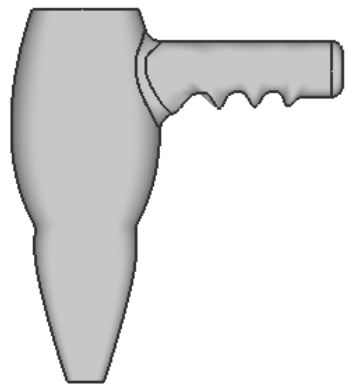


Figure 11-81 *Model after modification*

The following steps are required to complete this tutorial:

a. Open the model of Tutorial 2 in the *c06_tutorials* folder.
b. Set the target for transform rig by using the **Transform Rig** tool.
c. Add the modifier to the target.
d. Add the constraint to the target.
e. Add the clamper to the target.
f. Modify the model by using the **Scale** tool.
g. Create the mesh by using the **Nurbs to mesh** tool.
h. Save the mesh in the STL format.
i. Save the model and close the wire file.

Opening the Model

1. Choose **File > Open** from the menu bar; the **Open** dialog box is displayed. From the **Look in** drop-down list, navigate to the *c06_tutorials* folder.

2. Double-click on this folder to display the files in it and then select *c06_tut2.wire*.

3. Next, choose the **Open** button from the **Open** dialog box; the **confirm** message box is displayed.

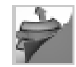
4. Choose the **Yes** button from this message box; the wire file of Tutorial 2 opens.

5. Next, choose **Layouts > Left** from the menu bar; the **Left** window expands to fill the entire windows area.

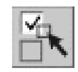
Setting the Target for Transformer Rig

For modifying the model, you need to set the target for the **Transformer Rig** tool.

1. Choose the **Transformer Rig** button from the **Object Edit** tab in the **Palette**; the **Transformer Rig Toolbox** is displayed and you are prompted to select the geometry. Also, the **Accept Targets** button in the inactive state is displayed at the lower right corner of the active window.



2. Choose the **Show Pick Mask** button from the **Transformer Rig Toolbox**; the **Transformer Rig Pick Mask : Add Target Mode** dialog box is displayed.



3. Clear the **Curve** check box and close the dialog box. Clearing this check box ensures that curves are not selected.

4. Next, select the entire model; the model is highlighted, as shown in Figure 11-82 and the **Accept Targets** button gets activated.

5. Choose the **Accept Target** button; the **confirm** message box is displayed.

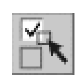
6. Choose the **Yes** button from the **confirm** message box; you are prompted to add free or predefined modifiers. Also, the **Go** button in the inactive state is displayed at the lower right corner of the active window.

Adding the Modifier

After setting the target for transform rig, you need to add modifiers to the target.

1. Choose the **Add Free Modifiers** button from the **Transformer Rig Toolbox**; you are prompted to select the geometry. Also, two buttons, **Cancel** in the active state and **Accept Modifiers** in the inactive state are displayed at the lower right corner of the active window.

2. Next, choose the **Show Pick Mask** button from the **Transformer Rig Toolbox**; the **Transformer Rig Pick Mask : Add Modifier Mode** dialog box is displayed.



3. Clear the **Surface** check box and close this dialog box. Clearing this check box ensures that surfaces are excluded from the selection.

4. Select the required curve, as shown in Figure 11-83; the **Accept Modifiers** button is activated.

5. Choose the **Accept Modifiers** button; the selected modifier is highlighted in blue, as shown in Figure 11-84. Also, the **Go** button is displayed at the lower right corner of the active window.

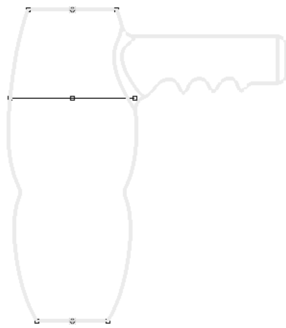


Figure 11-82 Target for transform rig

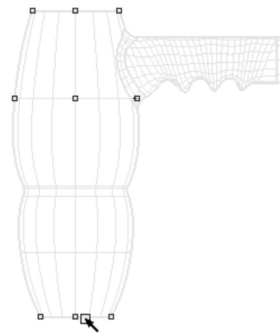


Figure 11-83 Curve to be selected as modifier

6. Choose the **Go** button; the **confirm** message box is displayed.

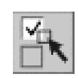
7. Choose the **OK** button; the modifier is added to the target.

Adding the Constraints

After adding modifier, you need to add constraints to the target. It is necessary to limit the modification of the target.

1. Choose the **Add Constraints** button from the **Transformer Rig Toolbox**; you are prompted to select the geometry. Also, the four buttons, **Tan**, **Pos**, **Cancel**, and **Accept Constraints** are displayed at the lower right corner of the active window.

2. Next, choose the **Show Pick Mask** button from the **Transformer Rig Toolbox**; the **Transformer Rig Pick Mask : Add Constraints Mode** dialog box is displayed.



3. Clear all check boxes, except the **Curve** check box in this dialog box, if they are selected. Clearing these check boxes ensures that only curves are included in the selection.

4. Select the curves, as shown in Figure 11-85; the **Accept Constraints** button is activated.

5. Choose the **Accept Constraints** button; the selected constraints are highlighted in red with the continuity label displayed on them, as shown in Figure 11-86. Also, the **Go** button is displayed at the lower right corner of the active window.

6. Choose the **Go** button; the **confirm** message box is displayed again.

7. Choose the **OK** button; the constraints are added to the target.

Adding the Clampers

Even after adding constraints, the **Transformer Rig** tool may not be able to constraint the modification of the target completely. To overcome this limitation, you need to add clampers.

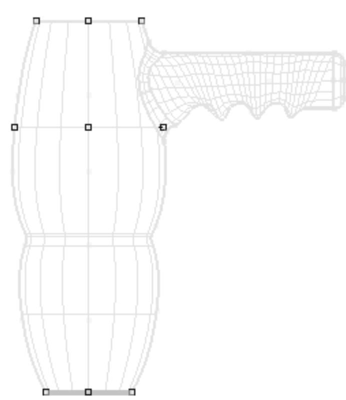


Figure 11-84 *Highlighted modifier*

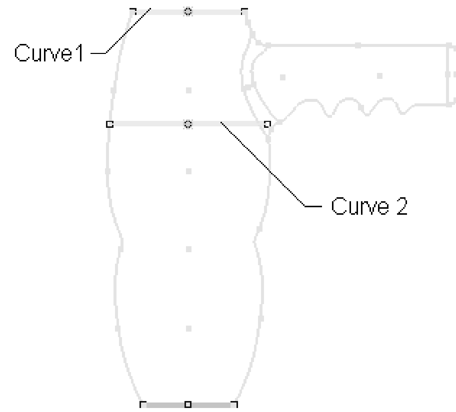


Figure 11-85 *Curves selected as constraints*

1. Choose the **Add Clampers** button from the **Transformer Rig Toolbox**; you are prompted to click on any geometry to add clamper. Also, the **Done** button is displayed at the lower right corner of the active window.

2. Select the n-sided surface from the **Pick chooser** menu that is displayed when you click on the right of the handle of the model; the clamper represented by pink indicator is displayed on the n-sided surface.

3. Add the second clamper to the n-sided surface. The handle of the model after adding two clampers is displayed, as shown in Figure 11-87.

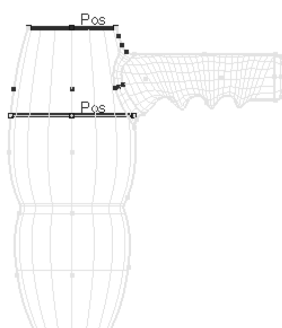


Figure 11-86 *Highlighted constraints with the continuity labels*

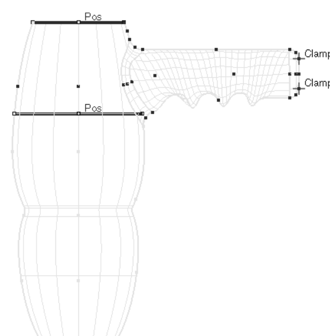


Figure 11-87 *Clampers added to the n-sided surface*

Tip: *The colors of the dots, which are displayed after adding modifiers or constraints, show the region to be modified dynamically. The region with green dots can be modified, whereas the region with red dots cannot be modified. Click on any dot with the middle mouse button to highlight these dots properly.*

4. Next, choose the **Done** button; the **Go** button is displayed.

5. Choose the **Go** button; the **confirm** message box is displayed.

6. Choose the **OK** button from the **confirm** message box; clampers are added to the target.

Modifying the Model Dynamically

After adding modifier, constraints, and clampers to the target, you need to modify it to the required shape by scaling it.

1. Choose the **Scale** button from the **Transform** tab in the **Palette** and then select the arc that was added as modifier; you are prompted to enter the scale factor.

2. Drag the cursor toward the model with the left mouse button to scale down the modifier geometry; the lower portion of the model is updated accordingly, as shown in Figure 11-88.

3. Select the modified model and then clear the **Isoparm U** and **V** check boxes; the display of the isoparametric curves is turned off.

4. Choose **WindowDisplay > Hardware Shade** from the menu bar; the model is shaded, as shown in Figure 11-89.

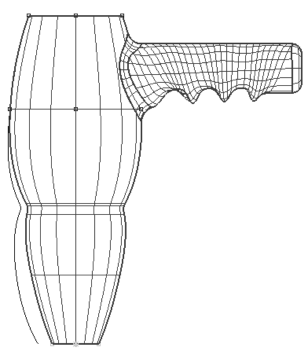


Figure 11-88 *Model after dynamic modification*

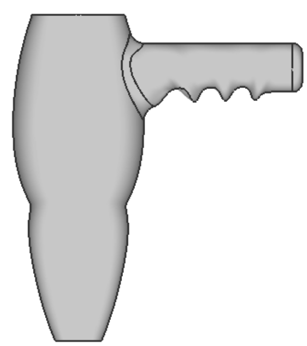


Figure 11-89 *Model after turning the display of isoparametric curves off*

Generating the Mesh on the Modified Model

After modifying the model dynamically, you can generate mesh on it. Before generating mesh, you need to change the parameters of the mesh.

1. Select the entire model; the model is highlighted.

2. Double-click on the **Nurbs to mesh** button from the **Mesh** tab in the **Palette**; the **Nurbs to Mesh Options** dialog box is displayed.

3. Enter **0.5** in the **Max edge length** edit box and then choose the **Go** button; the mesh is generated on the model, as shown in Figure 11-90.

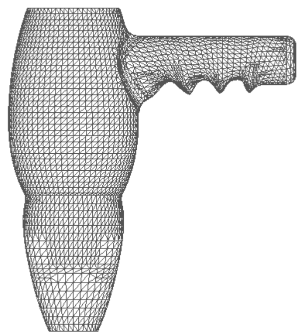


Figure 11-90 *Mesh generated on the model*

Note
All surfaces of the model in Figure 11-90 have been hidden to get a clear view of the generated meshes.

Saving the Mesh in the STL Format

After generating meshes, you need to save them in the STL format.

1. Choose **File > Export > STL** from the menu bar; the **Accept** button in the inactive state is displayed at the lower right corner of the active window.

2. Select all meshes from the active window; the **Accept** button is activated.

3. Choose the **Accept** button; the **Export STL** dialog box is displayed.

4. Enter *mesh_tutorial1* in the **Object name** edit box and then choose the **Save** button; the mesh is saved in the STL format.

Saving the File

1. Save the model with the name and location given below:

\aliasstudio_2009\c11_tutorials\c11_tut01.wire

Tutorial 2

In this tutorial, you will create the model of a bottle, as shown in Figure 11-91 and then modify it dynamically to the shape shown in Figure 11-92. **(Expected time: 30 min)**

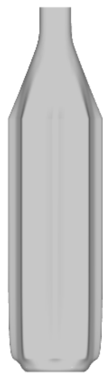


Figure 11-91 *Model before modification*

Figure 11-92 *Model after modification*

The following steps are required to complete this tutorial:

a. Start a new wire file.
b. Create the body, neck, and mouth of the bottle by using the **Skin surface** tool.
c. Create the base of the bottle by using the **Skin surface** and **Set planar** tools.
d. Attach the body to the base and neck by using the **Attach** tool.
e. Create the round on the base by using the **Round** tool.
f. Modify the bottle dynamically by using the **Twist Rig** tool.
g. Save the model and close the application.

Starting a New Wire File

1. Choose **File > New** from the menu bar; a new Studio wire file gets started and four windows are displayed on the screen. You can change the window display by choosing **Layouts** from the menu bar and then choosing the required option.

Creating the Body

The body of the bottle is created with the **Skin surface** tool. You need to turn the grid display on to create the bottle.

1. Double-click on the **Circle** button of the **Curves** tab in the **Palette**; the **Circle Options** dialog box is displayed.

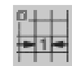
2. Enter **32** in the **Sections** edit box and then choose the **Go** button; you are prompted to specify the position of the circle.

3. Enter **0** in the promptline; the circle is created.

4. Choose the **Scale** button; you are prompted to enter the scale factor.

5. Enter **5** in the promptline; the circle is scaled five times as compared to its original size, as shown in Figure 11-93.

6. If grids are not displayed by default, choose **WindowDisplay > Toggles > Grid** from the menu bar to display grids in view windows.



7. Double-click on the **Grid preset** button of the **Construction** tab in the **Palette**; the **Preset Grid Options** dialog box is displayed.

8. Enter **5** in the **Subdivisions** edit box and then choose the **Go** button; grids with the modified setting are displayed, as shown in Figure 11-94.

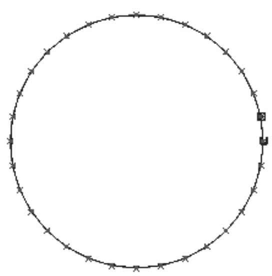


Figure 11-93 *Circle after scaling*

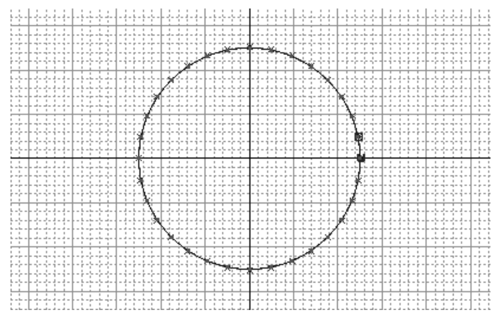


Figure 11-94 *Grids displayed in the window*

9. Next, choose the **Pick CV** button from the **Pick** tab in the **Palette**; you are prompted to select the CV.

10. Select the CV from the active window, as shown in Figure 11-95.

11. Choose the **Move** button from the **Transform** tab in the **Palette**; you are prompted to enter the move amounts.

12. Drag the selected CV with the ALT key pressed to the position, as shown in Figure 11-96.

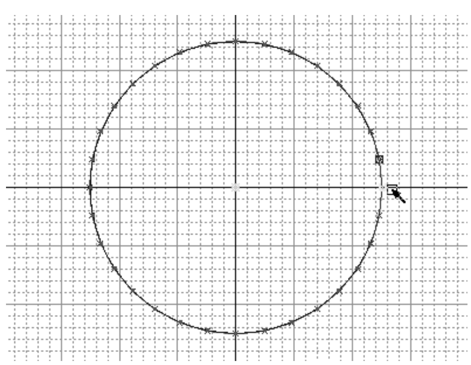


Figure 11-95 *Selected CV*

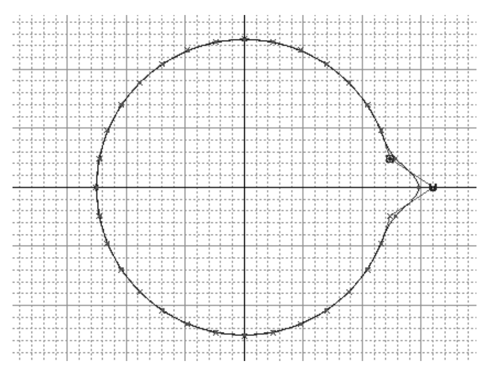


Figure 11-96 *Selected CV moved to the new position*

13. Similarly, move more CVs to the positions, as shown in Figure 11-97.

14. Select the modified circle and then choose **Edit > Duplicate** from the menu bar; a flyout is displayed.

15. Click on the box given on the right of the **Object** option of the flyout; the **Duplicate Object Options** dialog box is displayed.

16. Enter **15** in the Z-axis **Translation** edit box and then choose the **Go** button; the duplicate copy of the modified circle is created, as shown in Figure 11-98.

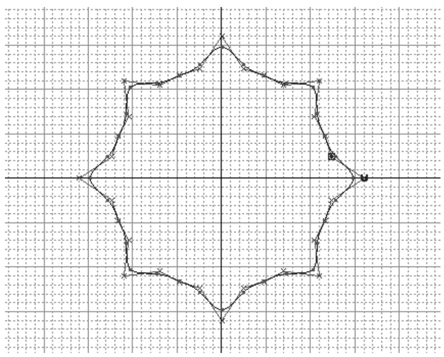


Figure 11-97 *Other CVs moved to new positions*

Figure 11-98 *Duplicate copy of the modified circle*

17. Next, choose the **Skin surface** button from the **Surfaces** tab in the **Palette** and select the modified circle and its duplicate copy; the body of the bottle is created, as shown in Figure 11-99.

Creating the Neck

The neck of the bottle is created by using the **Skin surface** tool. You need to create skin surface with three curves.

1. Choose the **Circle** button from the **Curves** tab in the **Palette** and then create two circles of different sizes at different positions, as shown in Figure 11-100.

2. Double-click on the **Skin surface** button; the **Skinning Options** dialog box is displayed.

3. Select the **Open** radio button in the **Topology** area and then choose the **Go** button to close the **Skinning Options** dialog box.

4. First, select the top edge of the body, and then select the two circles with the SHIFT key pressed; the neck is created, as shown in Figure 11-101.

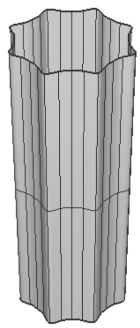


Figure 11-99 Body of the bottle created

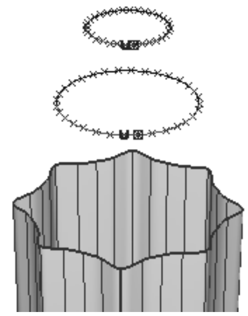


Figure 11-100 Circles created for the neck

Creating the Mouth

The mouth of the bottle is created by using the **Skin surface** tool. You need to attach the neck and the mouth to get a smooth transition between them.

1. Choose the **Circle** button from the **Curves** tab in the **Palette** and then create the circle, as shown in Figure 11-102.

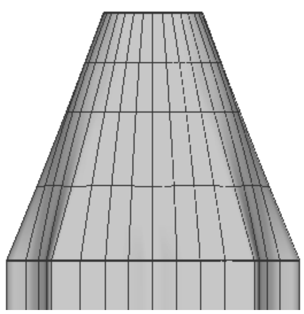


Figure 11-101 Neck of the bottle created

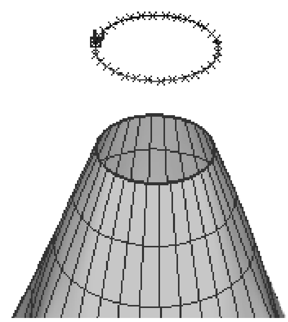


Figure 11-102 Circle created for the mouth

2. Choose the **Skin surface** button from the **Surfaces** tab in the **Palette** and select the top edge of the neck and then the circle; the mouth is created, as shown in Figure 11-103.

3. Double-click on the **Attach** button of the **Object Edit** tab in the **Palette**; the **Attach Options** dialog box is displayed.

4. Enter **0.5** in the **Position** edit box and then choose the **Go** button. Make sure the **Blend** radio button is selected in the **Type** area of the **Attach Options** dialog box.

5. Select the bottom edge of the mouth; the **confirm** message box is displayed.

6. Choose the **Yes** button from the **confirm** message box.

7. Select the top edge of the neck; the neck gets attached to the mouth with a smooth transition, as shown in Figure 11-104.

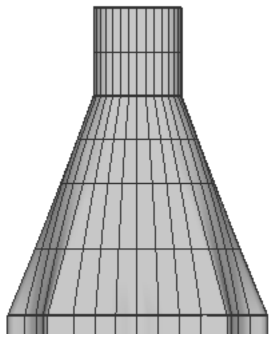


Figure 11-103 *Mouth of the bottle created*

Figure 11-104 *Mouth after attaching it with the neck*

Creating the Base

After creating mouth, you need to create the base of the bottle by using the **Skin surface** tool. You need to cap this skin surface by using the **Set planar** tool.

1. Choose the **Circle** button from the **Curves** tab in the **Palette** and create two circles, as shown in Figure 11-105. Note that these two circles should have different scale factors.

2. Next, choose the **Skin surface** button from the **Surfaces** tab in the **Palette** and select the bottom edge of the body; you are prompted to select the next curve.

3. Select two circles with the SHIFT key pressed; the base of the bottle is created, as shown in Figure 11-106.

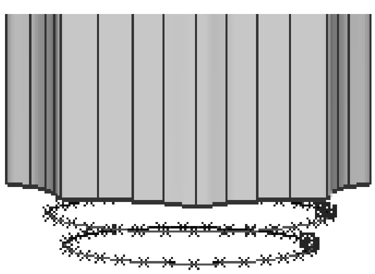


Figure 11-105 *Circles created for the base*

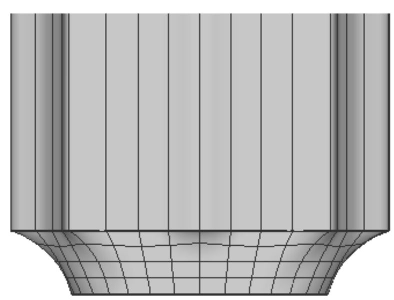


Figure 11-106 *Base of the bottle created*

4. Next, choose the **Set planar** button from the **Surfaces** tab in the **Palette** and select the bottom edge of the base; the **Go** button is displayed at the lower right corner of the active window.

5. Choose the **Go** button; the base gets capped, as shown in Figure 11-107.

6. Select entire model and then clear the **Isoparm U** and **V** check boxes from the **Display** area of the **Control Panel**; the model is displayed, as shown in Figure 11-108.

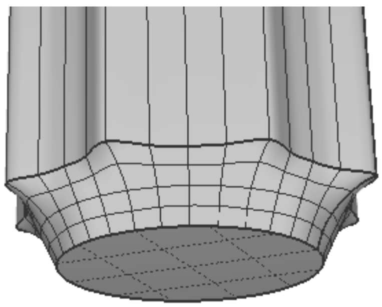


Figure 11-107 *Base of the bottle capped*

Figure 11-108 *Model after clearing the **Isoparm U** and V check boxes*

Attaching the Body to Neck and Base

After creating all surfaces, you need to attach them such that the model becomes a single unit.

1. Double-click on the **Attach** of from the **Object Edit** tab in the **Palette**; the **Attach Options** dialog box is displayed.

2. Select the **Connect** radio button in the **Type** area and then choose the **Go** button; you are prompted to select the first object near the attach location.

3. Select the bottom edge of the neck; you are prompted to select the second object near the attach location.

4. Select the top edge of the body; the body gets attached to the neck.

5. Similarly, attach the base to the body.

Note

The edges of the base or the neck coincide with the edges on the body. To attach the base or the neck to the edges on the body, you need to select the edges from the Pick chooser menu that is displayed when you click on the attach location.

Creating the Round on the Base

After attaching surfaces, you need to create a smooth transition between the base and its planar face by using the **Round** tool.

1. Choose the **Round** button from the **Surfaces** tab in the **Palette**; you are prompted to select a pair of edges. Also, the two buttons, **Built** and **Revert** are displayed at the lower right corner of the active window.
2. Select the common edges between the base and the planar surface; the radius manipulator is displayed on the selected edge.
3. Enter **0.3** in the promptline and choose the **Built** button; the round surface is created, as shown in Figure 11-109.

4. Next, exit the **Round** tool by choosing the **Pick object** button from the **Pick** tab in the **Palette**.
5. Scroll the mouse to zoom out the model to the required size, as shown in Figure 11-110.

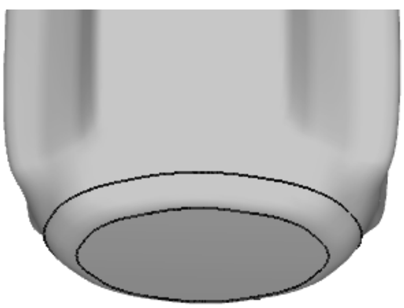


Figure 11-109 *Round created on the base*

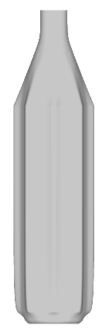


Figure 11-110 *Model after zooming out*

Modifying the Model Dynamically

After creating the entire model, you need to modify it dynamically using the **Twist Rig** tool. Before the modification, you need to create the twist axis using the **Line** tool.

1. Choose the **Line** button from the **Keypoint Curve Toolbox** and create the line such that it passes through the center of the entire model, as shown in Figure 11-111.

2. Choose the **Twist Rig** button from the **Object Edit** tab in the **Palette**; the **Twist Rig Toolbox** is displayed and you are prompted to select the geometry. Also, the **Accept Targets** button in the inactive state is displayed at the lower right corner of the active window.

3. Select the entire model excluding the line; the **Accept Targets** button gets activated.

4. Choose the **Accept Targets** button; the model is highlighted, refer to Figure 11-111. Also, you are prompted to select the twist axis and the **Accept Axis Curve** button in the inactive state is displayed at the lower right corner of the active window.

5. Select the line; the line is highlighted in yellow and the **Accept Axis Curve** button is activated.

6. Choose the **Accept Axis Curve** button; the twisted bottle with four manipulator handles is displayed, as shown in Figure 11-112.

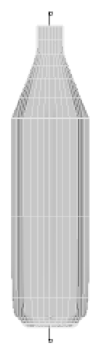


Figure 11-111 *Highlighted model with the line displayed at the center of the model*

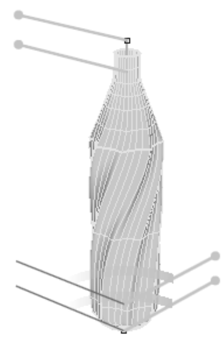


Figure 11-112 *Twisted bottle with manipulator handles*

7. Next, choose the **Commit** button from the **Twist Rig Toolbox**; the **confirm** message box is displayed.

8. Choose the **Yes** button from the **confirm** message box; the bottle gets modified dynamically, as shown in Figure 11-113.

Figure 11-113 *Bottle after dynamic modification*

Saving the File

1. Save the model with the name and location given below:

   \aliasstudio_2009\c11_tutorials\c11_tut02.wire

Self-Evaluation Test

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

1. Clampers are used to limit the modification of the target inside the regions of interest only. (T/F)

2. After adding modifiers and constraints to the target, the region with red colored dots can be modified, whereas the region with green colored dots cannot be modified. (T/F)

3. When you accept the axis for twisting by default, the target gets twisted by 90-degree in the counter-clockwise direction. (T/F)

4. While modifying the model using the **Bend Rig** tool, the length and width of the target is displayed in the promptline and their values keep changing when you scale the target with a mouse button. (T/F)

5. You cannot change the direction of the target while modifying the model using the **Conform Rig** tool. (T/F)

6. You can obtain meshes by converting polysets or __________ surface to meshes.

7. The data generated by 3D scanning is known as __________.

8. Meshes can be saved in wire, STL, or __________ file format.

9. The curves-on-mesh help in separating, merging, or __________ meshes.

10. By default, __________ twist handles are displayed on the selected twist axis.

Review Questions

Answer the following questions:

1. What is the range of sampling density used in the **Transformer Rig** tool?

   (a) 0.05 to 2.0.　　(b) -1 to 1
   (c) 1 to 10　　(d) 0 to 1

2. Which of the following alphabets represents twist angle?

   (a) A (b) T
   (c) W (d) G

3. Which of the following file formats cannot be used to import meshes in AliasStudio?

   (a) Wire (b) STL
   (c) OBJ (d) IGES

4. Which of the following messages is displayed in the **Mesh Repair Control** dialog box when AliasStudio fails to diagnose the status of the tested mesh?

   (a) **Pass** (b) **Fail**
   (c) **Unknown** (d) **OK**

5. The range of the maximum number of samples in the **Sampling and Diagnostics** area of the **Transformer Rig Options** dialog box is 100 to 5000. (T/F)

6. Meshes are built directly in AliasStudio. (T/F)

7. Subsetting a mesh allows you to shade parts of the mesh separately. (T/F)

8. A component is defined as a collection of connected __________ of the mesh.

9. By reversing the orientation of meshes, you can reverse the direction of the __________ of the entire mesh.

10. In __________ defect, a vertex is shared by more than two boundary edges.

Exercises

Exercise 1

In this exercise, you will create a twisted staircase, as shown in Figure 11-114.

(Expected time: 30 min)

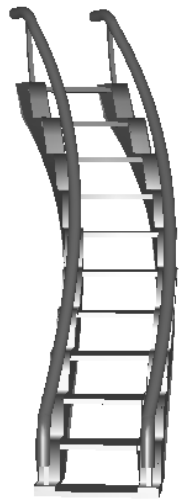


Figure 11-114 *Model for Exercise 1*

Exercise 2

In this exercise, you will create the bent model, as shown in Figure 11-115.

(Expected time: 30 min)

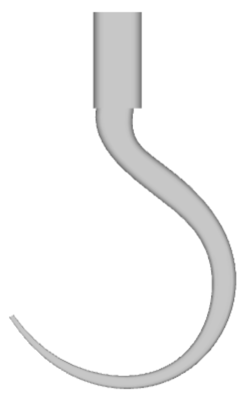


Figure 11-115 *Model for Exercise 2*

Answers to Self-Evaluation Test

1. T, **2.** F, **3.** T, **4.** T, **5.** F, **6.** NURBS, **7.** cloud data, **8.** OBJ, **9.** cut, **10.** four