Chapter 1

Working in Autodesk Maya

The Autodesk® Maya® working environment has evolved to accommodate the individual artist as well as a team of artists working in a production pipeline. The interface presents tools, controls, and data in an organized fashion to allow you to bring your fantastic creations to life easily.

Understanding the way Maya organizes data about the objects, animations, textures, lights, dynamics, and all the other elements contained within the 3D environment of a scene is essential to understanding how the interface is organized. Maya uses what's known as the Dependency Graph to keep track of the various packets of data, called nodes, and how they affect each other. Any single element of a Maya scene consists of multiple nodes connected in a web, and each one of these nodes is dependent on another. The Maya interface consists of editing windows that allow you to connect these nodes in an intuitive way and edit the information contained within each node.

There is usually more than one way to accomplish a task in Maya. As you grow comfortable with the interface, you'll discover which editing windows best suit your working style.

This chapter is a brief overview of what professionals need to understand when working in Maya. You'll learn what types of nodes you'll be working with and how they can be created and edited in Maya. You'll also learn how to work with projects and scene data as well as the various windows, panels, and controls that make up the interface. This will help you, whether you are working alone or as part of a team of artists.

This chapter is about working with nodes, but it is not meant to be a comprehensive guide to each and every control in Maya. You will find that information in the Maya documentation. If you've never used Maya before, I strongly encourage you to read the Maya documentation as well as *Introducing Autodesk Maya 2015*, by Dariush Derakhshani (Sybex, 2014).

In this chapter, you will learn to:

- Understand transform and shape nodes
- Create a project

Creating and Editing Nodes

A Maya *scene* is a system of interconnected nodes that are packets of data. The data within a node tells the software what exists within the world of a Maya scene. The nodes are the building blocks that you, as the artist, put together to create the 3D scene and animation that will finally be rendered for the world to see. So if you can think of the objects in your scene, their motion, and their appearance as nodes, think of the Maya interface as the tools and controls you use to connect those nodes. The relationship between these nodes is organized by the *Dependency Graph (DG)*, which describes the hierarchical relationship between connected nodes. The interface provides many ways to view the graph, and these methods are described in this chapter.

Any given workflow in Maya is much like a route on a city map. There are usually many ways to get to your destination, and some of them make more sense than others depending on where you're going. In Maya, the best workflow depends on what you're trying to achieve, and there is typically more than one possible ideal workflow.

There are many types of nodes in Maya that serve any number of different functions. All the nodes in Maya are considered DG nodes. Let's say you have a simple cube and you subdivide it once, thus quadrupling the number of faces that make up the cube. The information concerning how the cube has been subdivided is contained within a DG node that is connected to the cube node.

A special type of DG node is the *directed acyclic graph (DAG)* node. These nodes are made of two specific types of connected nodes: transform and shape. The arrangement of DAG nodes consists of a hierarchy in which the shape node is a child of the transform node. Most of the objects you work with in the Maya viewport, such as surface geometry (cubes, spheres, planes, and so on), are DAG nodes.

To understand the difference between the transform and shape node types, think of a transform node as describing where an object is located and a shape node as describing what an object is.

The simple polygon cube in Figure 1.1 consists of six flat squares attached at the edges to form a box. Each side of the cube is subdivided twice, creating four polygons per side. That basically describes what the object is, and the description of the object would be contained in the shape node. This simple polygon cube may be 4.174018 centimeters above the grid, rotated 35 degrees on the x-axis, and scaled four times its original size based on the cube's local x- and y-axes and six times its original size in the cube's local z-axis. That description would be in the transform node.

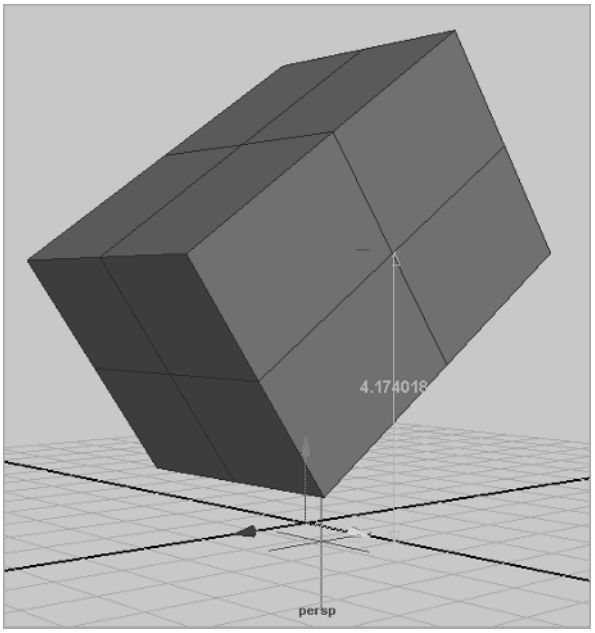


Figure 1.1
A shape node describes the shape of an object and how it has been constructed; a transform node describes where the object is located in the scene.

Maya has a number of workspaces that enable you to visualize and work with the nodes and their connections. The following sections describe how these workspaces work together when building a node network in a Maya scene.

Using the Hypergraph

The *Hypergraph* is a visual representation of the nodes and their connections in Maya. A complex scene can look like an intricate web of these connections. When you need to know how a network of nodes is connected, the Hypergraph gives you the most detailed view. There are two ways to view the Hypergraph:

- The *hierarchy view* shows the relationships between nodes as a tree structure.
- The *connections view* shows how the nodes are connected as a web.

You can have more than one Hypergraph window open at the same time, but you are still looking at the same scene with the same nodes and connections.

This short exercise gives you a sense of how you would typically use the Hypergraph:

1. Create a new Maya scene.
2. Create a polygon cube by choosing Create ➢ Polygon Primitives ➢ Cube.
3. You will be prompted to draw a polygon on the grid by dragging on the grid (see Figure 1.2). Drag a square on the grid, release the cursor, and then drag upward on the square to turn it into a three-dimensional cube. Release the mouse button to complete the cube. At this point, feel free to make your own decisions about the size and position of the cube on the grid.

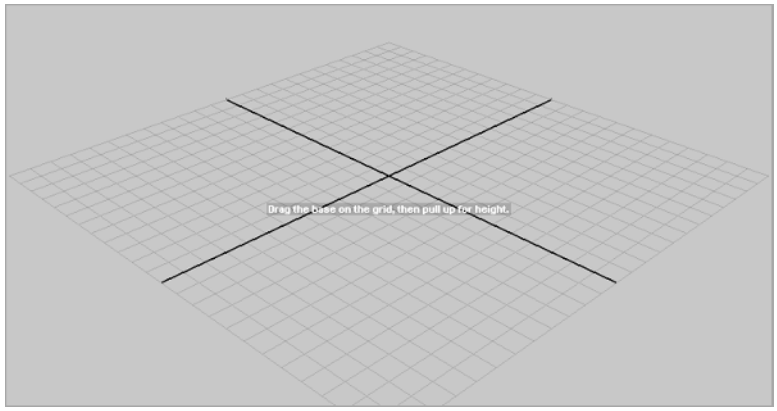


Figure 1.2
Maya prompts you to draw the base of the cube on the grid in the scene.

4. Select the cube in the viewport, and choose Window ➢ Hypergraph: Hierarchy to open the Hypergraph in hierarchy mode. You'll see a yellow rectangle on a black field labeled pCube1. The rectangle turns gray when deselected.
5. Move the mouse over the rectangle labeled pCube and then right-click. Choose Rename from the pop-up window. Rename the cube **myCube**.

Interactive Creation

By default Maya creates objects using the *Interactive Creation method*, which allows you to draw on the canvas as you create your geometry. To turn this feature off, choose Create ➢ Polygon Primitives, and deselect the Interactive Creation option at the bottom of the menu. This feature is turned off for all the projects in this book to ensure precise placement of primitive objects.

While the Interactive Creation mode is on, you can deselect the Exit On Completion method; this means that each time you draw on the grid, you will continue to create cubes until you switch to another tool.

6. Select myCube and, from the Hypergraph menu, choose Graph ➢ Input And Output Connections. This switches the view to the connections view just as if you had originally opened the Hypergraph by choosing Window ➢ Hypergraph: Connections. It's the same Hypergraph, but the view mode has changed, allowing you to see more of the scene.

When you graph the input and output connections, you see the connected nodes that make up an object and how the object appears in the scene. In the current view, you should see the myCube node next to a stack of connected nodes labeled polyCube1, myCubeShape, and initialShadingGroup, as shown in Figure 1.3. (The nodes may also be arranged in a line; the actual position of the nodes in the Hypergraph does not affect the nodes themselves.)

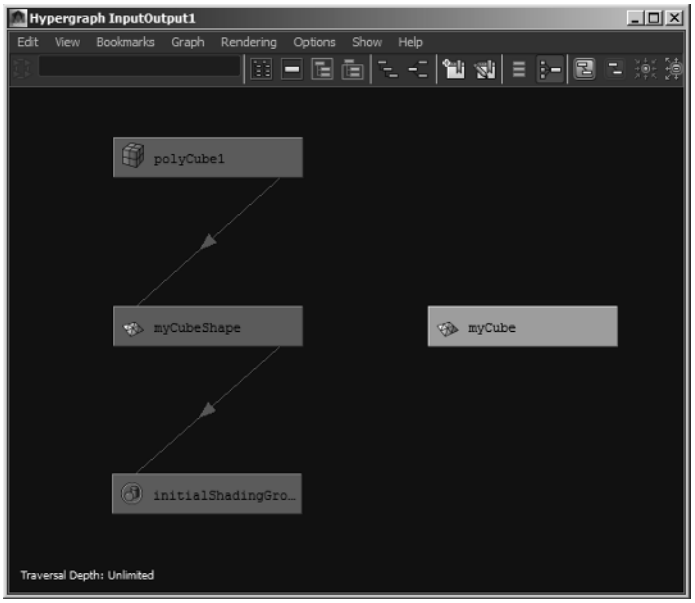


Figure 1.3
The node network appears in the Hypergraph. This shape node (myCubeShape) is connected to two other nodes, whereas the transform node (myCube) appears off to the side.

Navigating the Hypergraph

You can navigate the Hypergraph by using the same hot-key combination you use in the viewport: Alt+MMB-drag/Option+MMB-drag pans through the Hypergraph workspace, and Alt+RMB-drag/Option+RMB-drag zooms in and out. (MMB means clicking with the middle mouse button, and RMB means clicking with the right mouse button.) Selecting a node and pressing the **f** hot key focuses the view on the currently selected node. It is also possible to zoom in using the scroll wheel on your mouse.

The myCube node is the transform node. The myCubeShape node is the shape node. In the Hypergraph, the shape and transform nodes are depicted as unconnected; however, there is an implied connection, as you'll see later. This is demonstrated when you rename the myCube node; the shape node is renamed as well.

In Maya, the construction history feature stores a record of the changes used to create a particular node. The polyCube1 node is the construction history node for the myCubeShape node. When you first create a piece of geometry, you can set options to the number of subdivisions, spans, width, height, depth, and many other features that are stored as a record in this history node. Additional history nodes are added as you make changes to the node. You can go back and change these settings as long as the history node still exists. Deleting a history node makes all the previous changes to the node permanent (however, deleting history is undoable). Use the following steps to guide you through the process of modifying history nodes:

1. Keep the Hypergraph open, but select the cube in the viewport.
2. Change the menu set in the upper left of the main interface to Polygons.
3. Press the 5 key on the keyboard to switch to shaded mode. Choose Mesh ➢ Smooth. The cube will be subdivided and smoothed in the viewport.

   In the Hypergraph you'll see a new polySmoothFace1 node between the polyCube1 node and the myCubeShape node (see Figure 1.4). This new node is part of the history of the cube.

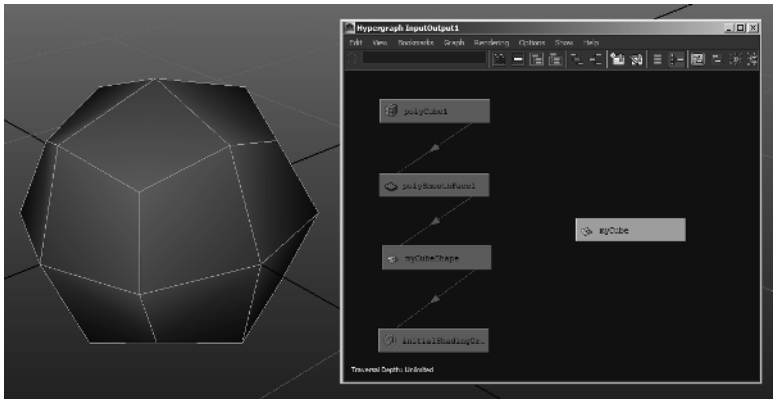


Figure 1.4
Performing a smooth operation on the cube when construction history is activated causes a new polySmoothFace1 node to be inserted into the node network.

4. Select the polySmoothFace1 node, and delete it by pressing the Backspace key on the keyboard. The cube will return to its unsmoothed state.

5. Select the transform node (myCube), and press the **s** hot key. This creates a keyframe on all the channels of the transform node. A keyframe stores the current attribute values at a particular time on the timeline. Animation is created by interpolating between keyframed values.

   You'll see a new node icon appear for each keyframed channel with a connection to the transform node (see Figure 1.5).

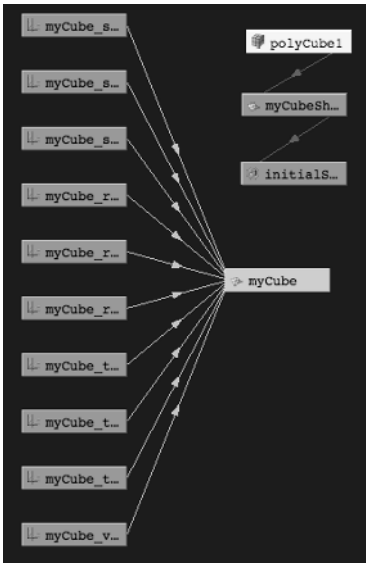


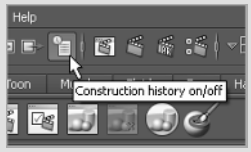
Figure 1.5
The attributes of myCube's transform node have been keyframed. The keyframe nodes appear in the Hypergraph.

6. Hold the cursor over any line that connects one node to another. A label appears describing the output and input attributes indicated by the connection line.

Working with History

Over the course of a modeling session, the history for any given object can become quite long and complex. This can slow down performance. It's a good idea to delete history periodically on an object by selecting the object and choosing Edit ➢ Delete By Type ➢ History. You can also choose to delete the history of all the objects in the scene at once by choosing Edit ➢ Delete All By Type ➢ History. Once you start animating a scene using deformers and joints, you can use the Delete By Type ➢ Non-Deformer History option, which will remove the construction history nodes while preserving connections to animation nodes such as deformers.

You can turn off the history globally by clicking the Construction History toggle switch on the status line, as shown here:



Connecting Nodes with the Node Editor

Connections between nodes can be added, deleted, or changed using the Hypergraph and the Connection Editor. Introduced in Maya 2013, the Node Editor combines the features of the Hypergraph, Hypershade, and Connection Editor into a single graphical interface. Maya 2015 brings numerous changes and enhancements.

When you open the Node Editor you are presented with an empty, grid-lined space. To view a selected node, you must choose the type of connections you wish to graph: input, output, or both. After establishing a graph, you can add additional nodes by choosing the icon with three nodes and a red plus symbol from the Node Editor toolbar.

Every node has a series of ports for connecting attributes between nodes. By default, the nodes are shown in Simple mode, meaning none of their attributes or other ports are shown. In Simple mode, you can click on the dot on either side of the node to access a pop-up menu for a node's input or output connections. When unconnected, the superport is white. After connecting, the port takes on the color of the connected attribute. You do not always have to use the superport and subsequent pop-up menu to make connections. You can expose the lesser ports by changing the nodes' display from their simple, default display to Connected, exposing the connected attributes, or to Full mode. Click the icon in the lower-right corner of each node to change its display. You can also press 1, 2, or 3 on the keyboard, with the node selected, to change its mode. To change all of the nodes' modes at once, use the Edit menu at the top left of the Node Editor. The Full mode allows you to see all of the connectable ports (see Figure 1.6).

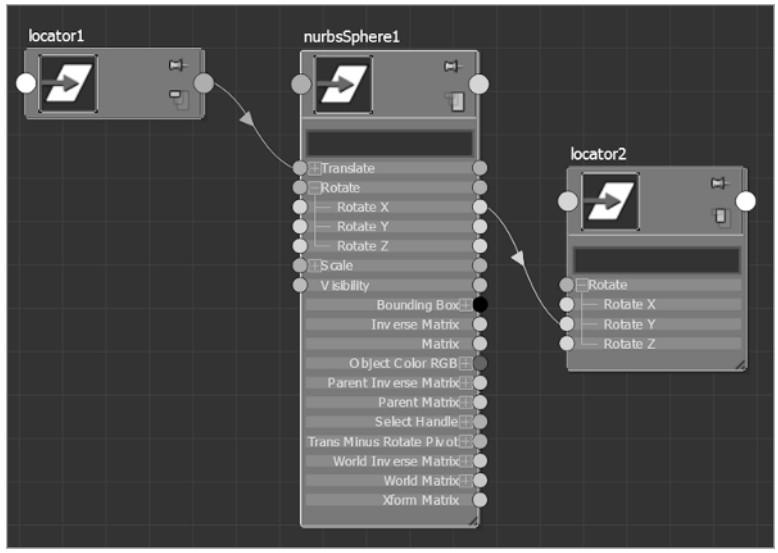


Figure 1.6
The various display modes, starting with locator1 in default, nurbsSphere1 in full, and locator2 in Connected mode

Like with all editors in Maya, you can customize the colors of the Node Editor using the Windows ➢ Settings/Preferences ➢ Color Settings window. The Attribute Type rollout under the Node Editor rollout allows you to change the color of the various types of connections.

The following steps walk you through the basic uses of the Node Editor and how to make connections:

1. Start a new Maya scene.
2. Create a locator in the scene by choosing Create ➢ Locator. A simple cross appears at the center of the grid in the viewport. This locator is a simple nonrendering null that indicates a point in space. Locators are handy tools that can be used for a wide variety of things in Maya.
3. Press the **w** hot key to switch to the Move tool; select the locator at the center of the grid, and move it out of the way.
4. Press the **g** hot key to create another locator. The **g** hot key repeats the last action you performed, in this case the creation of the locator.
5. Create a NURBS sphere in the viewport by choosing Create ➢ NURBS Primitives ➢ Sphere. If you have Interactive Creation selected, you'll be prompted to drag on the grid in the viewport to create the sphere; otherwise, the sphere will be created at the center of the grid based on its default settings.
6. Move the sphere away from the center of the grid so that you can clearly see both locators and the sphere.
7. Use the Select tool (hot key = **q**) to drag a selection marquee around all three objects.
8. Open the Node Editor by choosing Window ➢ Node Editor. A grid is drawn in the workspace. The grid can be toggled on or off by choosing the grid-visibility button (circled in Figure 1.7) from the Node Editor's toolbar. Choose the Input and Output Connections button, also circled in Figure 1.7, to graph your selection.

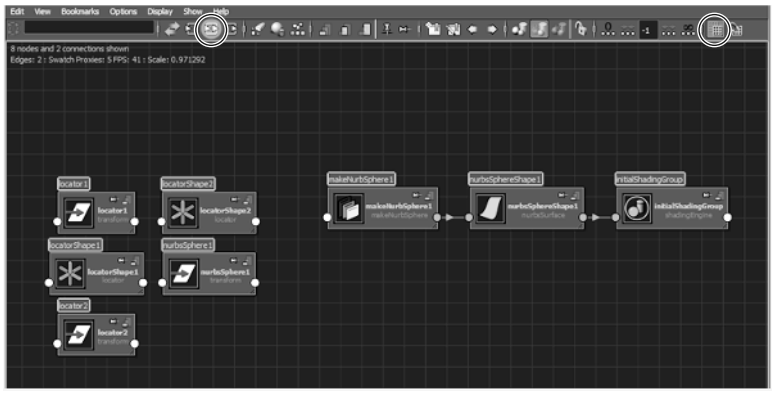


FIGURE 1.7
The input and output connections of the two locators and the sphere are graphed in the Node Editor.

At the bottom of the screen are the two transform nodes for locator1 and locator2. locatorShape1 and locatorShape2 are the shape nodes for the locators. nurbsSphere1 is the transform node for the NURBS sphere. And nurbsSphereShape1 is the shape node for the sphere; it's connected to makeNurbSphere1, which is the history node, and to initialShadingGroup. The initialShadingGroup node is the default shading group that is applied to all geometry; without this node, the geometry can't be shaded or rendered. When you

apply a new shader to an object, the connection to initialShadingGroup is replaced with a connection to the new shader. A *shader* is a node that determines how a surface appears in Maya, as well as how it reacts to virtual lights.

9. In the Node Editor, use Alt+RMB to zoom in and out. Notice how the name bar that sits on top of each node scales with the camera, enabling you to view long names regardless of your camera view.

10. Select the locator1, locator2, and nurbsSphere1 transform nodes, and drag them away from the other nodes so that you can work on them in their own space. To keep your graph neat, you can use snap to grid to align your nodes with the grid.

11. Click the white superport on the right side of the locator1 node. This is the output.

12. From the pop-up menu, choose Translate ➢ Translate. A yellow wire extends from the translate node (see Figure 1.8). The wire can be connected to a white port on another node.

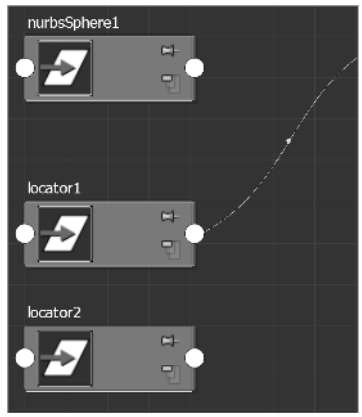


FIGURE 1.8
Use the wire to connect the output of one node to the input of another.

13. Connect the yellow wire to the left side of the nurbsSphere1 node by clicking on its white superport and choosing Translate from the pop-up menu. You can connect the yellow wire to either side of a node. The connection will be the same. A green wire shows the finished connection.

    You can also choose Other from the pop-up menu. Doing so brings up the Input Selection window. The window lists the attributes of the node. Any of the attributes that have a plus sign next to them can be expanded to reveal nested attributes. For instance, find the Translate attribute and expand it by clicking the plus sign. (The term *translate* in Maya refers to an object's position. When you use the Move tool to change the position of an object in 3D space, you are "translating" the object.) You'll see that Translate has Translate X, Translate Y, and Translate Z. This means that you can choose either to select the Translate attribute, which will automatically use all three nested attributes as the input connection, or to expand Translate and choose one or more of the nested Translate X, Y, or Z attributes as the input connection. In some situations, a connection becomes unavailable (grayed out), indicating that the connection between the two attributes cannot be made, usually because the connection is not appropriate for the selected attributes (see Figure 1.9).

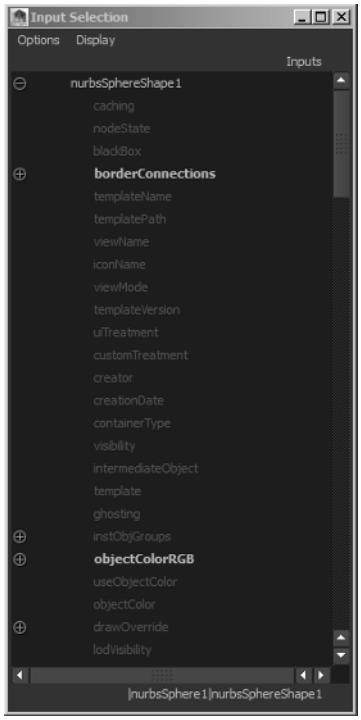


Figure 1.9
The Input Selection window specifies which attributes can be connected between nodes.

14. In the viewport, switch to wireframe mode if you are not already in it. You can do this by pressing **4** on the keyboard or by clicking the wireframe icon on the icon bar at the top of the viewport window; the wireframe icon is the wireframe cube.

15. In the viewport, you'll notice that the sphere has snapped to the same position as the locator. Select the sphere, and try to move it using the Move tool (hot key = **w**). The sphere is locked to the locator, so it cannot be moved. Select the locator, and try to move it; the sphere moves with the locator. The output of the locator's Translate attributes is the input for the sphere's Translate.

Incoming Connections

In wireframe view, an object will be highlighted in purple if it has an incoming connection from the selected object.

16. Click on the nurbsSphere1 node and press 3 to display the node's attributes in full. Click the plus sign next to Rotate to expand the Rotate rollout. Repeat the procedure for locator2.

17. Click on the right-side port or output for RotateX on nurbsSphere1.

18. Drag the yellow wire to the left side (the input side) of locator2 and connect it to RotateY. The yellow wire turns cyan to match the color of its port of origin and the connection is made. The difference in color indicates rotational values as opposed to numeric values, like those used by the translate and scale attributes.
19. In the viewport, select the sphere and switch to the Rotate tool (hot key = **e**).
20. Drag up and down on the red circle of the tool to rotate the sphere around its x-axis. The locator rotates around its y-axis.

Using the Node Editor to Make Simple Connections

The Node Editor is perfect for making one-to-one relationships between attributes on two nodes. In other words, the value of the output connection needs to equal exactly the value of the input connection. You can also create nodes from inside the editor by using the RMB and subsequent marking menu.

The Node Editor can get cluttered quickly. To combat a messy graph, you can pin your nodes to their current position and size regardless of regraphing. The pushpin icon in the upper-left quadrant of the node (top-right corner on Mac) allows you to toggle the pinning feature. You can also select Options ➢ Pin All Nodes By Default or RMB-click in the Node Editor to access the option through the marking menu.

New! It is also possible to reduce the number of attributes displayed in each node to help simplify your view. To activate Edit Custom Attribute List, RMB-click on a port of a node you wish to customize. In edit mode, the node is divided by a yellow bar. The attributes above the yellow bar are the ones that display. Those below the yellow bar are hidden. You can click on an attribute to hide or unhide it. Attributes can also be rearranged by MMB-dragging to a new location within the node. To accept your changes, RMB-click on the node's graphic and deselect Edit Custom Attribute List. You can then press **4** to see your edited node. To revert the node to its original settings, RMB-click on any attribute in edit mode and choose Revert from the context menu.

Experiment with making connections between the various attributes with the Node Editor. You can break a connection by selecting and dragging the arrow along the wire into empty space. Notice the changes in the port colors when making or breaking connections.

Creating Node Hierarchies in the Outliner

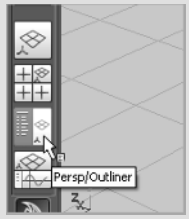
CERT OBJECTIVE

The Outliner shows a hierarchical list of the nodes in the scene in a form similar to the outline of a book. It is another way to view the transform and shape nodes in a scene and a way to create hierarchical relationships between nodes through parenting. The Outliner does not show the connections between nodes like the Hypergraph does; rather, it shows the hierarchy of the nodes in the scene. To see how this works, try the following exercise:

1. Open `miniGun_v01.ma` from the `Chapter1/scenes` directory at the book's web page (`www.sybex.com/go/masteringmaya2015`). The scene consists of a minigun model in three parts.
2. Open the Outliner by choosing Window ➢ Outliner.

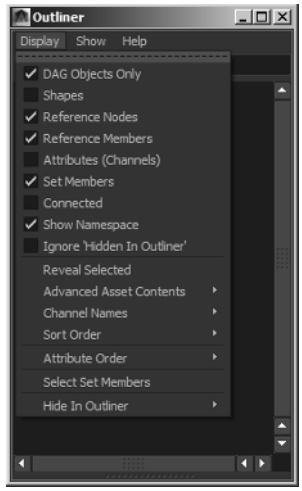
Outliner Layout Presets

The Outliner can be opened as a separate panel or, like many of the panels in Maya, it can be opened in a viewport. A popular window arrangement is to split the viewports into two views, with the left view set to the Outliner and the right view set to the perspective view. You can open this arrangement by going to the menu bar in a viewport window and choosing Panels ➢ Saved Layouts ➢ Persp/Outliner. You can also click the third layout button on the left side of the interface just below the toolbox (as shown here).



3. At the top of the Outliner is a menu bar. In the Display menu, make sure DAG Objects Only is selected and Shapes is deselected (see Figure 1.10).

Figure 1.10
The Display menu at the top of the Outliner



In the Outliner, you'll see three nodes listed—gunBarrels, housing, and mount—in addition to the four default cameras and several set nodes (don't worry about the set nodes). These are the three transform nodes for the pieces of the minigun. Select each node, and you'll see the corresponding part highlighted in the perspective view. At the moment, each piece is completely separate and unconnected.

4. Select the housing node, and switch to the Rotate tool (hot key = **e**).

5. Rotate the objects; nothing else is affected. Try translating housing using the Move tool (hot key = **w**); again, nothing else is affected.

6. Use Undo (hot key = **Ctrl/Cmd+z**) a few times until the housing node returns to its original location and orientation.
7. In the Outliner, select the gunBarrels object. Then Ctrl/Cmd+click the housing object, and choose Edit ➢ Parent (hot key = **p**) from the main Maya menu at the top.

   Parenting one object to another means that you have made one transform node the child of the second. When an object is a child node, it inherits its position, rotation, scale, and visibility from the parent node. When you have multiple objects selected, the last object selected becomes the parent. In the Outliner, you'll notice that the housing node has a plus sign beside it and the gunBarrels node is not visible. The plus sign indicates that the node has a child node.
8. Click the plus sign next to the housing node to expand this two-node hierarchy. The gunBarrels node is now visible as the child of the housing node.
9. Select the housing node, and try rotating and translating it. The gunBarrels node follows the rotation and translation of the housing node (see Figure 1.11).

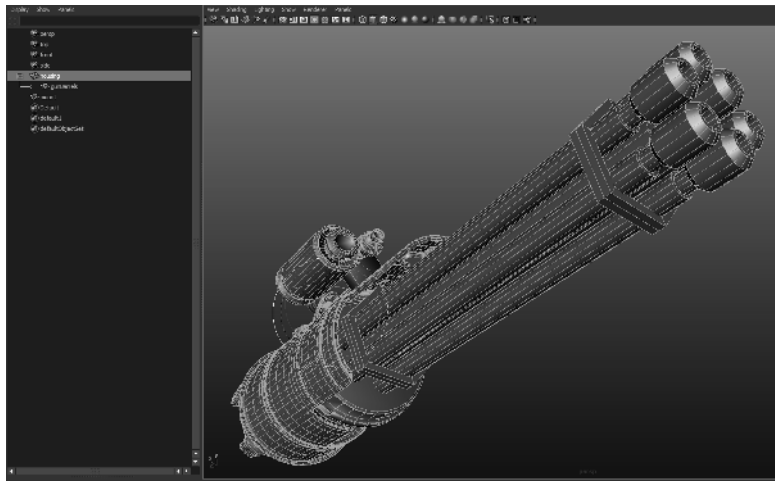


Figure 1.11
When the gunBarrels node is made a child of the housing object, it inherits changes made to the housing object's transform node.

   Unlike the situation presented in the "Connecting Nodes with the Node Editor" section earlier in this chapter, the rotation and translation of the gunBarrels object are not locked to the rotation and translation of the housing node; rather, as a child, its rotation, translation, scale, and visibility are all relative to that of its parent.
10. Select the gunBarrels node, and try rotating and translating the object; then rotate and translate the housing node. You'll see the gun barrels maintain their position relative to the housing node. You could create an animation in which the gun barrels rotate on their own z-axis to spin around while firing, at the same time the housing node is animated, rotating on all three axes in order to aim.
11. Press Undo a few times (hot key = **Ctrl/Cmd+z**) until both the housing and gunBarrels objects are back to their original positions.
12. In the Outliner, select the housing node, and MMB-drag it on top of the mount node. This is a way to parent objects quickly in the Outliner.

13. Click the plus signs next to the mount and housing nodes in the Outliner to expand the hierarchy. The lines indicate the organization of the hierarchy; the gunBarrels node is parented to the housing node, which is parented to the mount node.

Shift+Click to Expand the Hierarchy

You can expand an entire hierarchy with one click in the Outliner. Just Shift+click the plus sign for the hierarchy you want to expand.

14. Select the mount node, and choose Edit ➢ Duplicate (hot key = **Ctrl/Cmd+d**). This makes a copy of the entire hierarchy. The duplicated mount node is named mount1.
15. Select the mount1 node, and switch to the Move tool (hot key = **w**). Pull on the tool's red arrow to move the duplicate along the x-axis about two units.
16. Select the mount node, and then Ctrl/Cmd+click the mount1 node in the Outliner.
17. Choose Edit ➢ Group (hot key = **Ctrl/Cmd+g**) to group these two nodes under a single parent node.

    A group node is a transform node that has no shape node. It's just a location in space used to organize a hierarchy. Like a parent node, its children inherit its rotation, translation, scale, and visibility.
18. Select the group1 node, and Shift+click the plus sign next to it in the Outliner to expand the group and all its children.
19. Double-click the label for the group1 node in the Outliner to rename it; rename the group **guns**.

Renaming Nodes

You'll notice that the duplicate mount node has been renamed mount1 automatically. Nodes on the same level of the hierarchy can't have the same name. The child nodes do have the same name, and this is usually a bad idea. It can confuse Maya when more complex connections are made between nodes. Whenever you encounter this situation, you should take the time to rename the child nodes so that everything in the scene has a unique name.

20. Select the mount1 node in the guns hierarchy, and choose Modify ➢ Prefix Hierarchy Names.
21. In the pop-up window, type **right_**. This renames the top node and all its children so that "right_" precedes the name. Do the same with the other mount node, but change the prefix to **left_**.

CERT OBJECTIVE

22. Select the guns group, and choose Modify ➢ Center Pivot. This places the pivot at the center of the group. Try rotating the guns group, and you'll see both guns rotate together (see Figure 1.12).

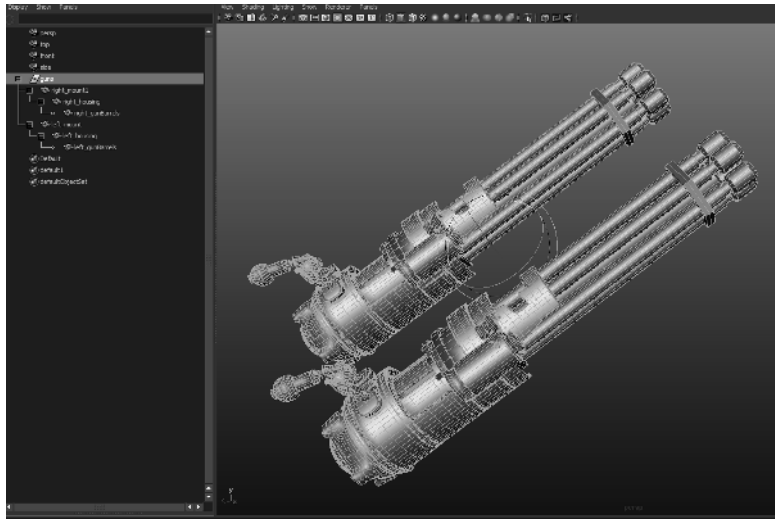


Figure 1.12
The guns group is rotated as a single unit.

Each member of the hierarchy can have its own animation, so both gun barrels can rotate around their z-axes as they fire, the two housing nodes could be animated to aim in different directions, and the two guns could rotate as one unit, all at the same time. The entire group can be parented to another node that is part of a vehicle.

Displaying Options in the Outliner

There are several options in the Outliner for displaying nodes and their hierarchical arrangements. You can see that the default perspective, top, side, and front cameras are visible as nodes at the top of the Outliner. Also, a number of sets, such as the defaultLightSet, appear at the bottom of the Outliner. These sets are mainly used for organization of data by Maya and are usually not directly edited or altered.

1. In the Display menu of the Outliner, select the Shapes option to display the shape nodes of the objects. The shape nodes appear parented to their respective transform node. You can select either the transform node or the shape node in the Outliner to select the object.

Accessing Outliner Options

You can right-click in the Outliner to access the Outliner's display options quickly, rather than use the menu at the top of the Outliner.

2. In the Display menu, activate the visibility of attributes by selecting the Attributes (Channels) option.

   Each node now has an expandable list of attributes. Most of the time you may want this option off because it clutters the Outliner and there are other ways to get to these attributes. Ultimately, how you use these options is up to you.

3. Turn off the Attributes display, and turn off the DAG Objects Only option. This allows you to see all the nodes in the scene in the Outliner list, as opposed to just the DAG nodes.

   DAG stands for "directed acyclic graph," and DAG objects are those that have both a shape and a transform node. It's not crucial to understand exactly what directed acyclic graph means as long as you understand that it is an arrangement in which a shape node is parented to a transform node. When you turn off DAG Objects Only in the Outliner, you'll see all the nodes in the Maya scene appear. Many of these are default utility nodes required to make Maya function, such as the layerManager node or the dynController1 node. Many other nodes appear when you create a new node or connection. An example of this is a keyframe or an expression node.

   When you turn off DAG Objects Only, the list can get quite long. To find a node quickly, type the node's name in the field at the very top of the Outliner. This hides all nodes except the named node. Clearing the field restores the visibility of all nodes in the Outliner (see Figure 1.13).

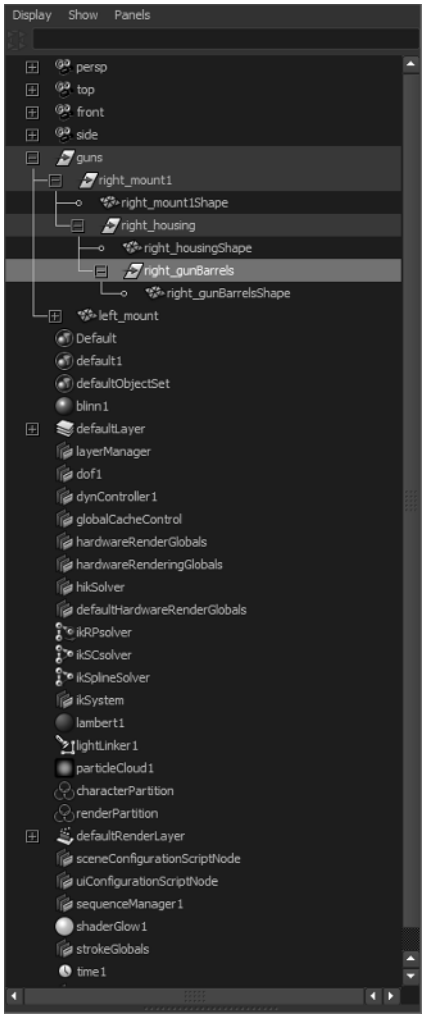


FIGURE 1.13
The Outliner can display shape nodes as well as other types of nodes in the scene.

Additional viewing options are available in the Show menu, which contains options for displaying only nodes of a certain type. Throughout this book, the Outliner will be used extensively, so you'll have lots of practice working with this panel.

4. In the Display menu of the Outliner, turn the DAG Objects Only option back on. Save the scene as **`minGun_v02.ma`**.

To see a finished version of the scene, open `miniGun_v02.ma` from the `chapter1\scenes` directory on the book's web page.

Searching Features in the Outliner

A complex scene in Maya can easily have hundreds of nodes. Just one character often has associated geometry, dozens of animation controls, joints, textures, and so on. Add another character to the scene with its own set of node networks, and the Outliner can become very cluttered very quickly. Establishing an organized naming system for the nodes in your scenes has many benefits, one of which is that you can use the search feature in the Outliner to filter what is displayed, thus making it easy to access the nodes you need. Take the time to name your nodes in such a way as to make searching easy. The following illustrates how the search feature can be used in a complex scene.

Let's say you have a scene with two complex characters, one named Larry and the other named Cecil. Both characters have similar rigs that use NURBS curves to control their animation rigs, and both have geometry, joints, shaders, and so on. When naming the nodes associated with each character, you make sure that all Larry nodes start with the name "larry." So, Larry's skin geometry might be named `larry_SKIN_GEO`, whereas his clothes would use names like `larry_PANTS_GEO`. Using capital letters in this case is purely a personal preference; the important thing is that the name of each node starts with `larry_`. Cecil would use the same convention; his skin geometry would be `cecil_SKIN_GEO`, and his pants would be `cecil_PANTS_GEO`. You end the names using `GEO` so that you know that this is a geometry node.

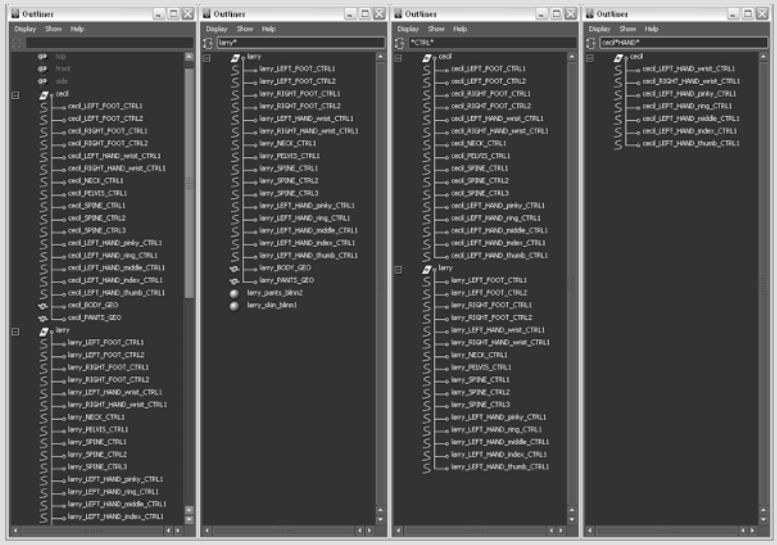
The controls for the animation rig use names like `larry_LEFT_HAND_wrist_CTRL1`, `larry_SPINE_CTRL1`, and `larry_NECK_CTRL1`. You get the idea. You can see that each of these nodes belongs to Larry, nodes for the left side of the body are clearly identified, the associated body part is identified, and they end with the letters `CTRL`. The same goes for Cecil.

Now here's where this type of organization, or something similar, is helpful in the Outliner. At the top of the Outliner is a blank field. To filter the nodes listed in the Outliner, you need to type some text and either precede or follow the text with an asterisk (*). The asterisk tells Maya to show all nodes that use the text before or after the asterisk in the name. So if you want to see all nodes associated with Larry, type **`larry*`**. If you want to see all the control nodes for both Cecil and Larry, type **`*CTRL*`**. In this case, there may be text before and after the CTRL letters, so use an asterisk before and after CTRL. If you want to see the controls associated with Cecil's hands, type **`cecil*HAND*`**, and so on.

The following images show variations on how to search through the Outliner with this method. If nothing appears in the Outliner when you type some text, check to see whether the asterisk is in the right place. To find one specific node, type its full name without the asterisk.

continues

(continued)



The Channel Box

CERT OBJECTIVE

The term *channel* is, for the most part, interchangeable with *attribute*. You can think of a channel as a container that holds the attribute's value. The *Channel Box* is an editor that lists a node's attributes for quick access. The Channel Box displays the node's attributes, which are most frequently keyframed for animation.

The Channel Box is located on the right side of the screen when the view mode at the end of the status bar is set to show the Channel Box/Layer Editor (see Figure 1.14).

FIGURE 1.14
The icon in the upper right of the interface toggles the visibility of the Channel Box/Layer Editor.

Two tabs on the very right side of the screen allow you to switch quickly between the Channel Box and the Attribute Editor, as shown in Figure 1.15. (The Attribute Editor is discussed in detail later in this chapter.) These tabs are visible when both the Attribute Editor icon and the Channel Box icon are activated on the status bar in the upper-right corner of the interface.

This exercise gives a quick tour of how to work in the Channel Box:

1. Create a new scene in Maya, and create a NURBS sphere on the grid (Create ➢ NURBS Primitives ➢ Sphere). You'll be prompted to draw the sphere on the grid if the Interactive Creation mode is on; if not, the sphere will appear at the center of the grid. Either option is fine.

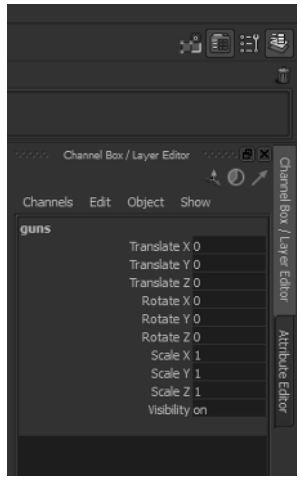


Figure 1.15
The two tabs on the right side of the screen allow you to switch quickly between the Channel Box and the Attribute Editor.

2. Make sure that the Channel Box is visible on the right side of the screen. To do this, click the icon at the farthest right of the status bar (shown earlier in Figure 1.14). This is a toggle to display the Channel Box. Click it until the Channel Box appears, as shown in Figure 1.16.

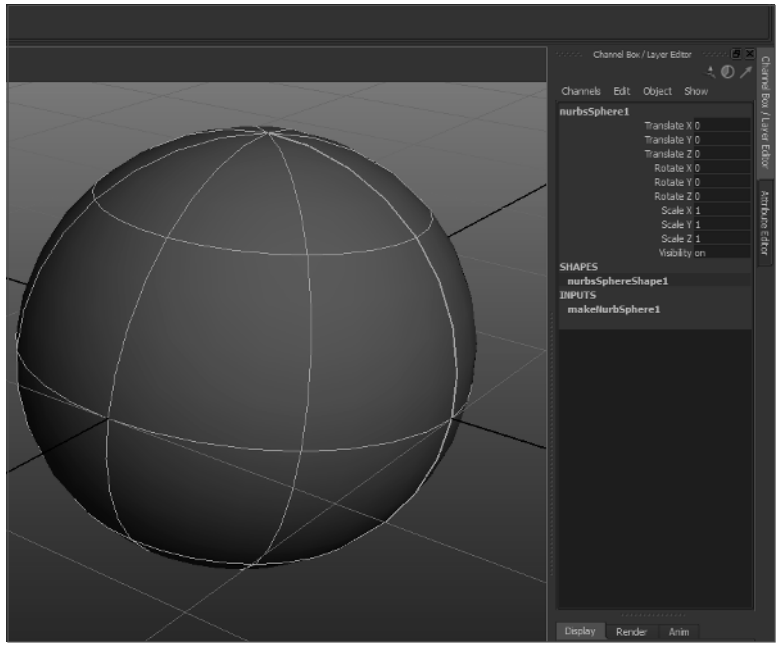


Figure 1.16
The Channel Box displays the channels for the currently selected object.

3. The Channel Box will list the currently selected object. Select the sphere, and you'll see nurbsSphere1 appear. The list below it shows the attributes for the nurbsSphere1's transform node.

   The lower half of the Channel Box lists the connections to this node. You'll see the name of the associated shape node under SHAPES, and below this a section for the inputs. In this case, the input is the history node, named makeNurbSphere1, which contains the original settings used to create the sphere. If you delete history on the sphere, these attributes will no longer be accessible.

4. In the upper section of the Channel Box, under nurbsSphere1, try selecting the fields and inputting different values for Translate, Scale, and Rotate. The sphere updates its position, size, and orientation.

5. In the Visibility channel, select the word On in the field, and type **0**. The sphere disappears. Input the value **1**, and it reappears. Visibility is a Boolean, meaning it is either on or off, 1 or 0.

6. Select the Translate X field so that it is highlighted. Shift+click the Rotate Z value, and all the values in between are also selected.

7. Type **0** in the Translate X field while they are selected, and press the Enter key. Doing so sets all the Translate and Rotate values to the same value, places the sphere at the center of the grid, and returns it to its original orientation (see Figure 1.17).

FIGURE 1.17
You can quickly "zero out" the Translate and Rotate channels by Shift+clicking their fields and entering 0.

8. In the makeNurbSphere section under INPUTS, highlight the Start Sweep channel. Enter a value of **90**, and the sphere opens up. If this is hard to see, switch to shaded mode by pressing **5** on the keyboard. You're altering the construction history of the sphere so that it is no longer a closed surface.

9. Select the word Sections so that it is highlighted in blue. MMB-drag back and forth in the viewport. Doing so creates a virtual slider so that you can change the value of the field interactively instead of numerically. This should work for all the channels (at least, most of the time).

10. Set the timeline to frame 1 by clicking on the far left of the time slider where it is labeled 1, and press the **s** hot key. You'll see all the channels turn orange, indicating that they have been keyframed. The **s** hot key keyframes all the available channels.

11. Move the timeline to frame 24, and change some settings both on the transform node (the upper half of the Channel Box) and under makeNurbSphere1 in the INPUTS section.

12. Press the **s** hot key again to set another key. Play the animation, and you'll see the sphere update based on the keyframed changes.

    If the animation seems to play too quickly, you need to change the preferences so that playback matches the frame speed of the animation. To do so, choose Windows ➢ Settings/Preferences ➢ Preferences. In the Preferences window, choose Time Slider on the left column and set PlayBack Speed to Real-Time [24 FPS].

    The **s** hot key keyframes everything, even those channels you may not need to keyframe. You can use the Channel Box to keyframe specific channels.

13. Rewind the timeline, and choose Edit ➢ Keys ➢ Delete Keys to remove all the keyframes on the sphere.

14. Highlight Translate X and Shift+click Translate Z so that the translation channels are all selected.

15. Right-click these values, and choose Key Selected (see Figure 1.18).

Figure 1.18 Right-click the selected channels, and choose Key Selected to keyframe just those specific channels.

16. Move to frame 24, and enter different values in the Translate fields.

17. Shift+click the Translate fields in the Channel Box, right-click, and choose Key Selected. This places a keyframe on just the selected channels—often a cleaner and more efficient way to work because you're placing keyframes only on the channels you need to animate and not on every keyable channel, which is what happens when you use the **s** hot key. Now try playing the animation.

Be Thrifty with Keyframes

Creating extra, unnecessary keys leads to a lot of problems, especially when you start to refine the animation on the Graph Editor (discussed in Chapter 2, "Introduction to Animation." Keyframes also can increase the scene size (the amount of storage space the scene uses on disk). Be cheap with your keyframes, and use the Key Selected feature to keyframe only the channels you need. Avoid using the **s** hot key to create keys on everything.

18. To remove keys, you can highlight the channels, right-click, and choose Break Connections. This removes any inputs to those channels. The values for the current keyframe will remain in the channels.

The channels are color-coded to show what kind of input drives the channel:

- Pink indicates a keyframe.
- Purple indicates an expression.
- Yellow indicates a connection (as in a connection from another node or channel, made in the Connection Editor or Node Editor).
- Brown indicates a muted channel.
- Gray means the channel is locked.

Locking and Muting Channels

You can mute a channel by right-clicking it and choosing Mute Selected from the context menu. When you mute a channel, the keyframes on that channel are temporarily disabled; as long as the channel is muted, the animation will not update. This is useful when you want to disable the keyframes in a channel so that you can focus on other aspects of the animation. Locking a channel is another option available when you right-click selected channels in the Channel Box. A locked channel prevents you from adding keyframes to a channel regardless of whether it has been animated. Creating animation is examined further in Chapter 2.

The Channel Box will be explored throughout the book and used frequently, particularly in the chapters concerning animation.

The Attribute Editor

The Attribute Editor is a tabbed panel that gives detailed information and access to a node's attributes. The tabs at the top of the editor allow you to move between the attributes of all the upstream (input) and downstream (output) connected nodes. This exercise gives a brief tour of how to use the Attribute Editor:

1. Create a new scene in Maya. Create a polygon cube on the grid (Create ➢ Polygon Primitives ➢ Cube).
2. Select the cube, and open its Attribute Editor. There are several ways to do this:
   - Right-click and hold the right mouse button over the cube, and choose pCube1 from the marking menu.
   - Select the cube, and choose Window ➢ Attribute Editor.
   - Click the Show/Hide Attribute Editor icon in the upper right of the Maya interface (Figure 1.19).

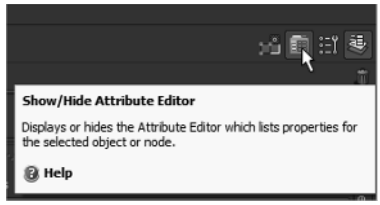


Figure 1.19
The Show/Hide Attribute Editor icon resides in the upper-right corner of the Maya interface.

- Press Ctrl+a on the keyboard. This toggles between Channel Box and Attribute Editor if they're docked.

3. With the Attribute Editor open, choose the pCube1 tab at the top (Figure 1.20). The panel that opens contains the attributes for the cube's transform node, much like the upper section of the Channel Box described in the previous section. It also contains options for setting limits on the transform attributes.

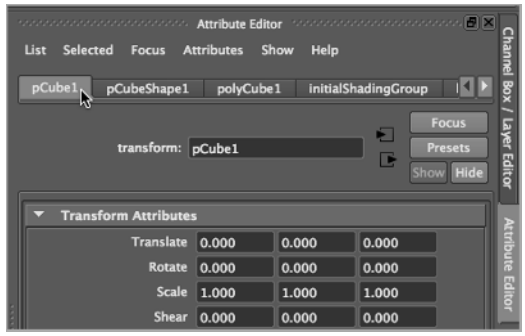


Figure 1.20
The Attribute Editor contains tabs that allow you to move through the connected nodes of a network.

Many of the settings can be accessed through the Attribute Editor's rollout panels. These are collapsible sections of grouped settings.

4. In the Attribute Editor, on the pCube1 tab, click the triangle next to mental ray. This reveals mental ray–specific settings related to the cube. Note that there are subsections under mental ray that are also collapsible.

5. Choose the pCubeShape1 tab at the top of the Attribute Editor. This tab contains settings related to the shape node. For example, expand the Render Stats section and you'll see a list of settings that control how the shape will appear in a render.

6. Choose the polyCube1 tab, and you'll see the construction history settings. If you delete history on the cube, this tab will no longer appear.

7. Expand the Poly Cube History rollout. If you right-click any of the fields, you get a menu that offers options such as expressions, key setting, or locking, much like the fields in the Channel Box (Figure 1.21).

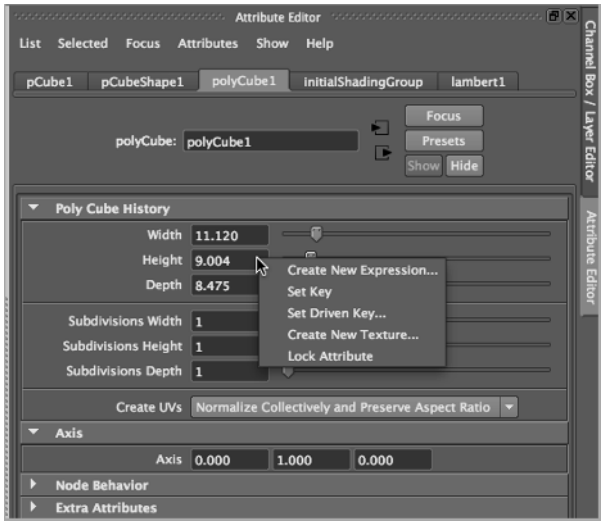


Figure 1.21
Right-clicking an attribute field reveals a menu with options for animating the attribute value.

8. In the Subdivisions Width field, highlight the text and type **=**. Doing so lets Maya know that you want to add an expression to the field.

Customizing the Attribute Editor

There are several ways to customize the Attribute Editor. The menu displayed by right-clicking on any field contains the Hide Attribute Control option. Choosing this removes the attribute from view. To display it, choose Show ➢ Show Attributes, and then choose the hidden attribute you want to show. For greater control, you can create an XML-based template file. Maya automatically reads XML files placed in the `scripts\AETemplates` folder of your Maya install directory. You can then choose the template from the Attribute Editor's menu Show ➢ Set Current View or Show ➢ Set Global View.

9. Complete the expression by typing **9*2** after the equals sign (see Figure 1.22); then press the Enter key. Doing so adds an expression to this attribute that makes the Subdivisions Width value equal to 18. Note that the field turns purple and the slider can no longer be moved.

Figure 1.22
You can enter simple mathematical expressions directly into a field in the Attribute Editor.

Note that a new tab called expression1 is added to the top of the Attribute Editor; this is a new expression node that is now part of the cube's node network.

If the number of connected nodes is too large to fit within the tab listing at the top, you can use the two arrow buttons to the right of the tabs to move back and forth between the tab listings. Likewise, if not all connections are visible, you can use the Go To Input and Go To Output Connection buttons to the right of the field indicating the node name.

The Notes field at the bottom is useful for typing your own notes if you need to keep track of particular settings or leave a message for yourself or other users (see Figure 1.23). You can collapse this section by dragging the bar above it downward, thus making more room for the other settings in the Attribute Editor.

Figure 1.23
Messages can be entered in the Notes section at the bottom of the Attribute Editor.

Load Attributes

You can use the Load Attributes button at the bottom of the Attribute Editor if the attribute display needs to be refreshed. Maya automatically updates the editor when new attributes are added, but occasionally it misses an update and needs to be refreshed.

The Attribute Editor is the workhorse panel of Maya. Throughout this book we will use it constantly. Make sure you are comfortable with the core concepts of how to switch between node settings using the tabs as well as how to change the available values.

Working with Shader Nodes in the Hypershade

The *Hypershade*, as the name suggests, is similar in function to the Hypergraph. It gives a visual display of how nodes in a Maya scene are connected. The Hypershade is mostly concerned with shaders—nodes used to define the color and material properties of renderable objects in a scene. These include materials (also known as shaders), textures, lights, cameras, and shading utilities. However, it is not unusual to use the Hypershade Work Area to make connections between other types of nodes as well. In this exercise, you'll use the Hypershade to connect several types of nodes.

1. Create a new scene in Maya. Create a NURBS cone on the grid. You'll be prompted to draw the cone on the grid if Interactive Creation mode is on; if it is not, the cone will appear at the center of the grid. Either option is fine.
2. Switch to smooth-shaded mode by pressing **6** on the keyboard, or click the Smooth Shade All and Textured icons on the viewport's menu bar (see Figure 1.24).

FIGURE 1.24
The Maya viewport menu bar allows you to choose between shading modes by toggling buttons.

3. Open Hypershade by choosing Window ➢ Rendering Editors ➢ Hypershade.

   The Hypershade Editor is made up of several frames. On the left side is a list and a visual menu of the nodes you can create in Hypershade. The list is divided into sections for the Maya nodes, mental ray nodes, and a customizable list for your own favorites at the very top. Clicking a category in the list filters the node-creation buttons to the right of the list, which helps to cut down on the amount of time you need to hunt for specific nodes. To add a node to the Favorites list, MMB-drag the node button from the right on top of the Favorites menu. You can also search through the nodes by typing in the field at the very top of the list. For example, typing **mia** in this field filters the node creation buttons so that all the mia (mental images architectural) nodes are displayed (see Figure 1.25).

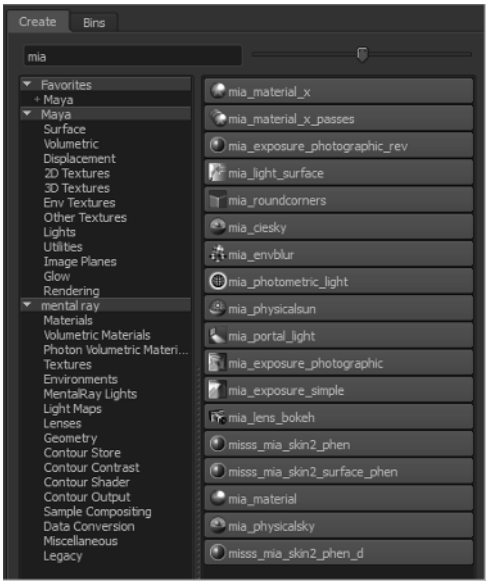


FIGURE 1.25
The text field at the top of the Create tab allows you to filter the list of buttons. MMB-dragging a button on top of the Favorites section adds the node to the Favorites.

   The right side of the Hypershade contains a visual display of the nodes in the scene at the top and the Work Area at the bottom (see Figure 1.26). The upper section is organized by tabs named Materials, Textures, Utilities, Rendering, Lights, Cameras, Shading Groups, Bake Sets, Projects, and Asset Nodes. If you want to access all the file textures used in the scene, you can choose the Textures tab to see them listed with preview icons.

4. On the left side of the Hypershade, click Surface in the list of Maya nodes. Click the Blinn button to create a new Blinn material.

You can see the new blinn1 material listed on the Materials tab; it also appears in the Work Area.

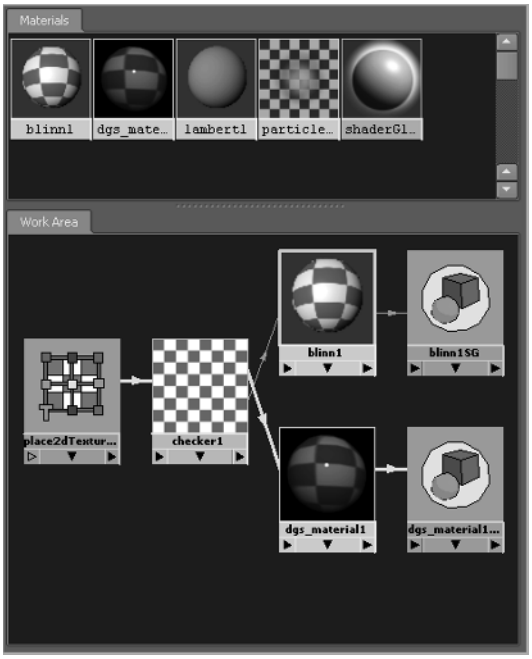
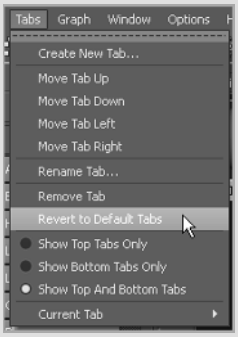


Figure 1.26
The Hypershade organizes render nodes and offers a workspace for constructing render node networks. This image shows an example of a shader network graphed in the Work Area.

Hypershade Tabs

If tabs appear to be missing, you can revert to the default tab layout by choosing Tabs ➢ Revert To Default Tabs, as shown here. This clears the current tabs and replaces them with the default Materials, Textures, Utilities, Rendering, Lights, Cameras, and other default tabs. You can use the Tabs menu to create your own custom tabs and determine which tabs you want visible and in what order.



Default Scene Materials

All Maya scenes start with three materials already created: lambert1, particleCloud1, and shaderGlow1. The lambert1 material is the default material applied to all newly created geometry, the particleCloud1 material is a special material reserved for particle-cloud objects, and the shaderGlow1 node sets the glow options for all shaders in the scene.

5. Select the blinn1 material in the Work Area and, from the menu at the top of the Hypershade, choose Graph ➤ Input And Output Connections. This displays all the upstream and downstream nodes connected to blinn1. Upstream nodes are nodes that plug into a node and affect its attributes; downstream nodes are ones that are affected by the node.

   The blinn1SG node is a downstream node known as a *shader group*, connected to blinn1. All materials have a shader group node connected to them. This node is a required part of the network that defines how the shader is applied to a surface, and it is often used when creating complex mental ray shader networks and overrides (see Figure 1.27).

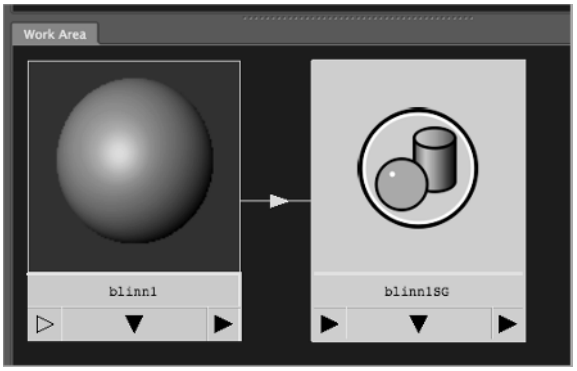


Figure 1.27
Shaders all have shading group nodes attached, which define how the shader is applied to the geometry.

6. In the viewport, select the cone. You can apply the blinn1 material to the cone in three ways:
   - MMB-drag the material icon from the Hypershade on top of the cone in the viewport window.
   - Select the cone, right-click the blinn1 node in the Hypershade, and choose Assign Material To Selection (see Figure 1.28).
   - Right-click the surface in the viewport, and choose Assign New Shader to create a new shader, or choose Assign Existing Material to assign a shader you've already created.
7. In the Work Area of the Hypershade, select the blinn1 node. This opens the Attribute Editor for the Blinn shader if it's not open already. This is where you can adjust the settings that define the look of the material.

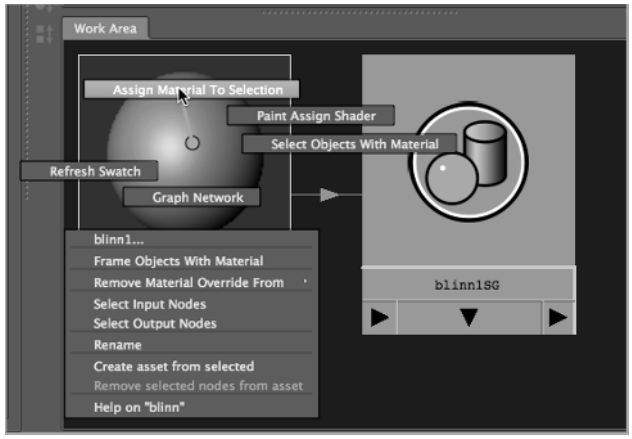


Figure 1.28
Right-click a shader, and drag upward on the marking menu to choose Assign Material To Selection.

Use the Shelf Buttons to Create a New Shader

You can assign a new material to a surface using the buttons in the rendering shelf at the top of the Maya interface (shown here). If you select an object and click one of these buttons, a new shader is created and assigned to selected objects. If no objects are selected, it just creates a new shader, which you'll find on the Materials tab of the Hypershade.

8. In the Attribute Editor for the blinn1 node, rename the material **coneShader** by typing in the field at the top of the editor.
9. Click the checkered box to the right of the Color slider. This opens the Create Render Node window (see Figure 1.29).
10. Select 2D Textures from the list on the left, and click the Grid button on the right side of the panel to create a grid texture; this is applied to the color channel of the cone, and it is visible on the cone in the viewport when textured and smooth-shaded mode is on (hot key = **6**) (see Figure 1.30).
11. Select coneShader in the Work Area of Hypershade, and right-click its icon. Choose Graph Network. You'll see that the coneShader node now has the grid1 texture node as well as the place2dTexture1 node attached (see Figure 1.31).
12. Click the place2dTexture1 node in the Work Area, and its attributes will be displayed in the Attribute Editor.

13. Type **0.5** in the first field next to Coverage and press Enter. This reduces the coverage of the grid texture in U space by one-half.

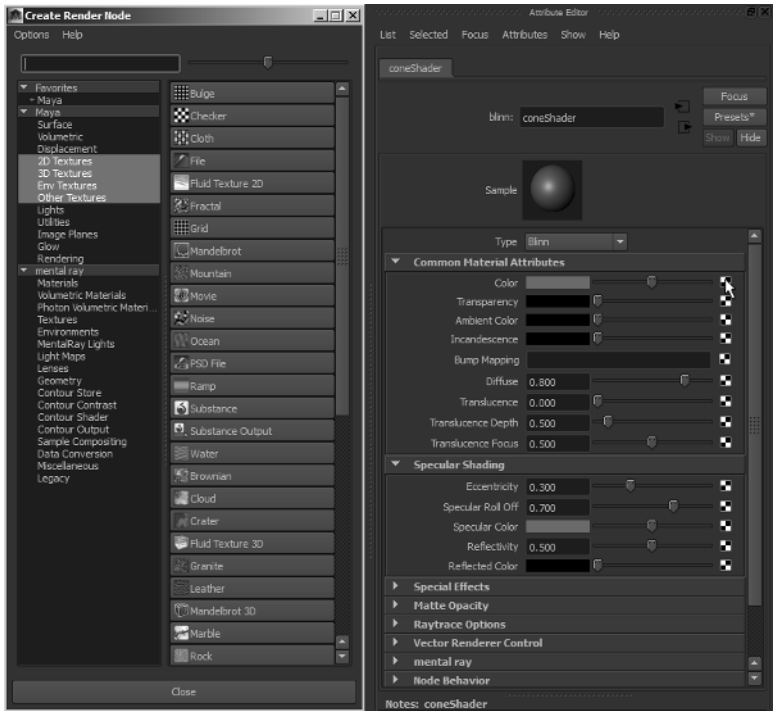


FIGURE 1.29
Click the checkered box next to the Color slider to open the Create Render Node window.

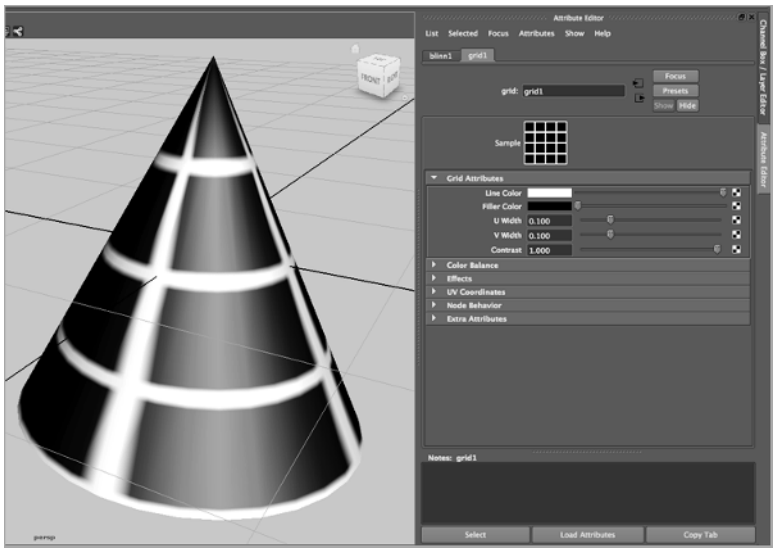


FIGURE 1.30
The grid texture appears on the cone when the perspective view is set to shaded mode.

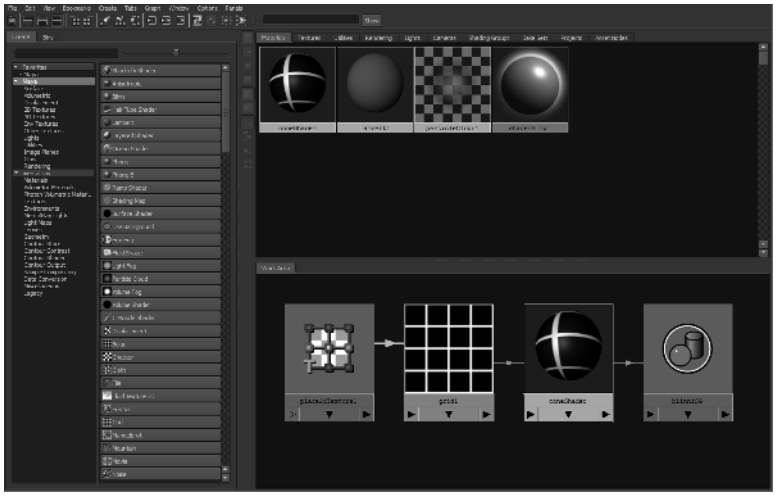


Figure 1.31
Applying the grid texture to the color channel of the coneShader adds two new nodes to the shader network. Graph the input and output connections to see these nodes in the Hypershade.

NURBS UV Texture Coordinates

NURBS surfaces have their U and V texture coordinates based on the parameterization of the surface, unlike polygon meshes, which require defined UV coordinates. You can use the attributes in the place2dTexture1 node to position textures on NURBS surfaces.

14. Select the grid1 node in the Work Area of the Hypershade to open its settings in the Attribute Editor.

15. Expand the Color Balance rollout, and click the color swatch next to Default Color. This opens the Color History. Set the color to **red**.

Default Color

The default color of the texture is the color "behind" the grid texture. Any part of the surface that is not covered by the grid (based on the settings in the place2dTexture1 node) will use the default color.

16. In the left panel of the Hypershade, select 2D Textures in the Maya list, and click the Ramp button to create a ramp node. At the moment, it is not connected to any part of the coneShader network. This is another way to create render nodes in the Hypershade.

Navigating the Hypershade Work Area

You can zoom in or out while in the Work Area of the Hypershade by holding down the Alt button while RMB-dragging; likewise, you can pan by holding the Alt button down while MMB-dragging.

17. Select the grid1 texture to open its settings in the Attribute Editor.

18. In the Work Area of the Hypershade, MMB-drag the ramp texture from the Work Area all the way to the color swatch next to the Filler Color in the grid's Attribute Editor. Hold the MMB while dragging; otherwise, you'll select the ramp texture, and the Attribute Editor will no longer display the grid1 texture attributes (see Figure 1.32). If the Connection Editor pops up as a result of dragging, you can just click Close.

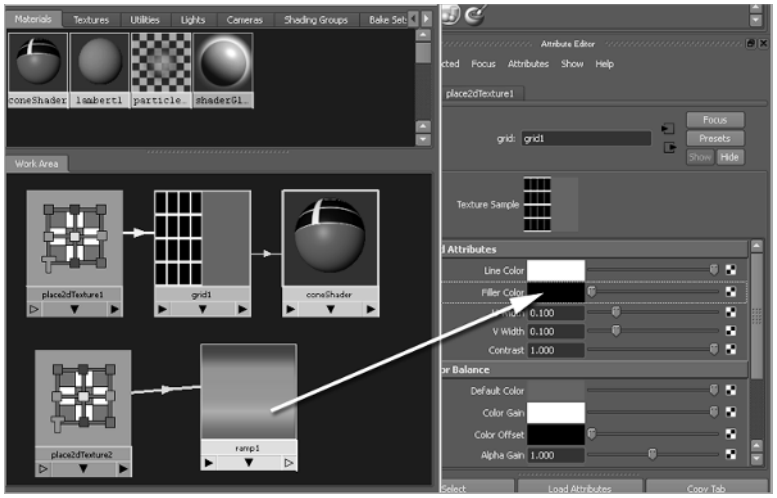


FIGURE 1.32
A texture node can be MMB-dragged from the Hypershade into an attribute slot in the Attribute Editor.

19. Select the coneShader node in the Work Area of the Hypershade, and choose Graph ➢ Input And Output Connections from the Hypershade menu. In the Work Area, you can see that the ramp texture is connected to the grid1 texture. The grid1 texture is connected to the coneShader, and the shader is connected to the blinn1SG node (see Figure 1.33).

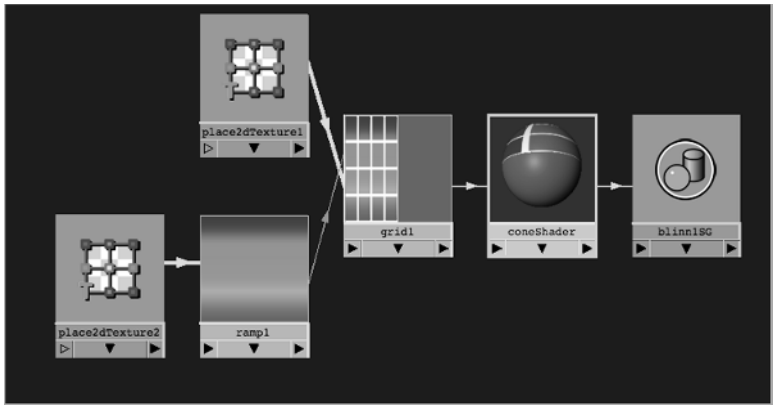


FIGURE 1.33
The coneShader network has grown with the addition of new nodes.

Graphing Shader Networks

You can graph the shader network at any time to refresh the view of the connected nodes in the Work Area. Just right-click the node you want to graph and choose Graph Network, or click the Input And Output Connections button at the top of the Hypershade (shown here). More options for displaying the network are available in the Graph menu at the top of the Hypershade.

20. Select the blinn1SG node, and graph its input and output connections. The cone's shape node appears (if the Bottom Caps option was on in the Creation options for the NURBS cone, you'll see a second shape node for the cone's bottom cap surface). The blinn1SG node is also connected to the render partition and the light linker, which defines the lights used to light the cone (see Figure 1.34).

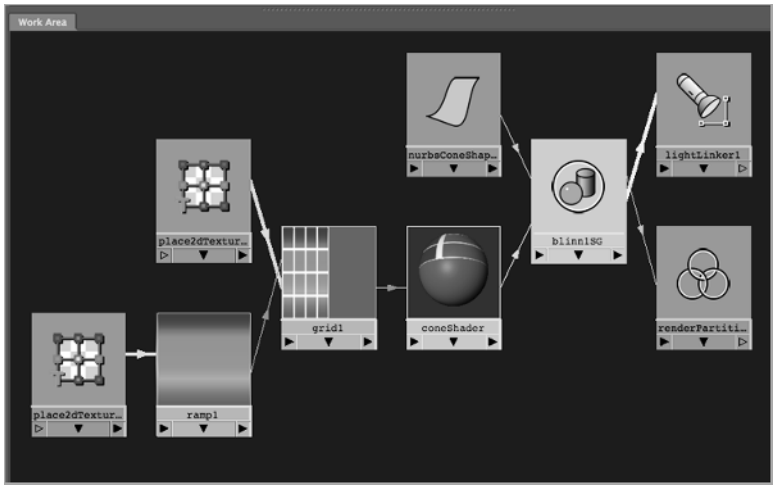


Figure 1.34
The shape nodes for the cone are included in the graph when the input and output connections of the blinn1SG node are graphed.

The Hypershade is a powerful and easy-to-use editor. You can build complex networks of nodes quickly, just like rearranging building blocks. You can see how nodes are connected by holding the mouse pointer over the lines that connect the nodes.

The previous sections of this chapter revealed to you the many ways Maya nodes can be displayed, connected, and edited in a Maya scene. Make sure that you are comfortable with the basics of working with the editors described. You will rely on them heavily throughout the book, and by working through the various exercises, you will gain proficiency in using them.

Creating Maya Projects

Organization is the key to creating a successful animation. Whether you are working by yourself or with others in a production pipeline, you'll want to know where to find every file related to a particular project, whether it's a scene file, a texture map, an image sequence, or a particle disk cache. To help you organize all the files you use to create an animation, Maya offers you the option of easily creating a Maya project, which is simply a directory with subfolders where each file type related to your scenes can be stored.

Creating a New Project

Creating a new project is simple. Projects can be created on your computer's primary hard drive, a secondary drive, or a network drive. The scene files used for each chapter in this book are stored in their own project directories at `www.sybex.com/go/masteringmaya2015`. Maya uses a default project directory when one has not been specified. This is located in your `My Documents\maya\projects` folder in Windows. As an example, you'll create a project directory structure for the examples used in this chapter.

1. Start a new Maya scene. You'll note that an empty scene is created when you start Maya.
2. Choose File ➢ Project Window.
3. The Project Window dialog box opens. Click the New button to the right of the Current Project field. In the Current Project field, type **`Mastering_Maya_Chapter01`** (see Figure 1.35).

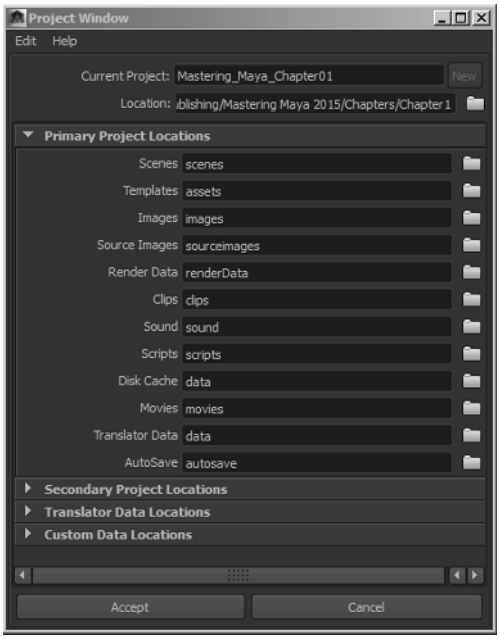


Figure 1.35
The Project Window dialog box lets you set the location of a new project directory and determine its directory structure on your hard drive.

4. To the right of the Location field, click the folder icon to browse your computer. The Select Location window opens and lets you determine where on your computer or network you want the project to be stored, or you can simply type the path to the directory. The project folder can be a subfolder of another folder if you like.

   In the Primary Project Locations section, you'll see a large number of labeled fields. The labels indicate the various types of files a Maya scene may or may not use. The fields indicate the path to the subdirectory where these types of files will be located.

   When you click the New button as mentioned in step 3, Maya automatically fills in all the fields (see the right side of Figure 1.36).

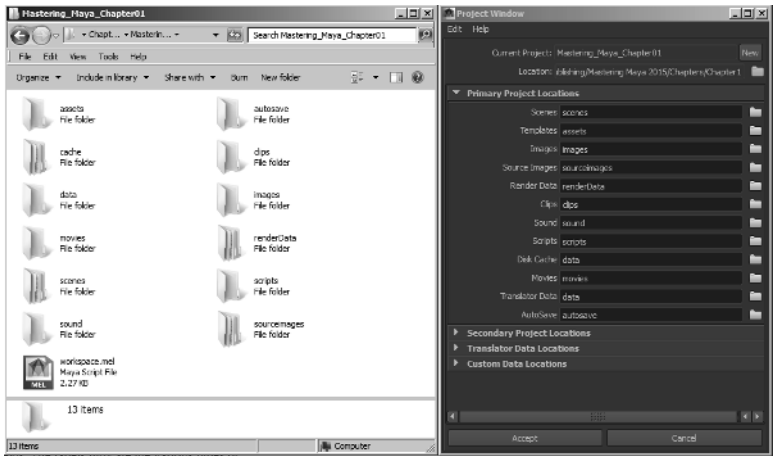


Figure 1.36
Clicking the New button fills in all the fields with the preferred default file structure in Maya (right side of the image). The directory structure is created on the specified drive (left side of the image).

   The fields contain the name of the subdirectory relative to the project file. So when you choose to use the default settings, all Maya scene files (files with the `.mb` or `.ma` file extension) will be stored in a folder labeled `Scenes`. The path to that folder will be, in this example, `Mastering_Maya_Chapter01\Scenes`.

Editing Default File Locations

If you decide you want to store the scene files in a different directory, you can type the path to that directory in the field or click the folder icon to the right of the field to set a different folder path.

5. Click Accept. Maya will take a few moments to create the project directory and all subfolders on the specified drive.

6. Use your computer's file browser to locate the new project; then expand the folder, and you'll see all the subfolders.

Editing and Changing Projects

You may not need to use every folder Maya creates for you, or you may decide to change where Maya looks for elements such as file textures. If you're working on a number of projects, you may need to switch the current project. All of these options are available in the Maya File menu.

1. To edit the current project, choose File ➢ Project Window. The Project Window dialog box opens with all the paths to the project subdirectories. You can type a new directory path in any one of these fields or click the folder icon to browse your computer's directory. Then click the Accept button. If a folder does not already exist, Maya will create it for you.

Relinking Files After Changing Project Settings

If you edit the project, Maya will look in the newly specified folders from this point on, but files used prior to editing the project will not be copied or moved. You'll need to move these files using the computer's file browser if you want Maya to find them easily after editing the project.

2. To switch projects, you can choose File ➢ Set Project or choose a project listed in the Recent Projects menu.

When working on a project with a number of other animators, you can choose to share the same project, which is a bit risky, or each animator can create their own project directory structure within a shared folder. The latter approach is a little safer because it prevents two people from having the same project open or overwriting each other's work. Later in this book, you'll learn how multiple animators can share parts of the scene using file references.

It is possible to work on a scene file outside the current project. This happens usually when you forget to set the project using the File ➢ Set Project option. Make a habit of setting the current project each time you start to work on a scene; otherwise, linked files such as textures, dynamic caches, and file references can become broken, causing the scene to behave unpredictably (which is a nice way of saying that the scene will fall apart and possibly crash Maya).

While working with the project files for this book, you'll want to copy the entire project to your local drive and then use the Project Window dialog box to choose the corresponding chapter project directory as the current project. This way, all the linked files in the scenes should behave correctly and the exercises will function.

Overriding Project Settings

You can choose to override a project setting for an individual scene element. For instance, by default Maya looks to the `sourceimages` directory for file textures. However, when you create a file texture node, you can use the Browse button to reference a file anywhere on your machine or the network. This is usually not a great idea; it defeats the purpose of organizing the files in the first place and can easily lead to broken links between the scene and the texture file. It's a better idea to move all file textures used in the scene to the `sourceimages` directory or whatever directory is specified in your project settings.

The Bottom Line

Understand transform and shape nodes. DAG nodes have both a transform node and a shape node. The transform node tells where an object is located; the shape node describes how it is made. Nodes can be parented to each other to form a hierarchy.

Master It Arrange the nodes in the `miniGun_v03.ma` file in a hierarchical structure so that the barrels of the guns can rotate on their z-axis, the guns can be aimed independently, and the guns rotate with the turret.

Create a project. Creating a project directory structure keeps Maya scene files and connected external files organized to ensure the animation project is efficient.

Master It Create a new project named Test, but make sure the project has only the scene, source images, and data subfolders.