

Autodesk®
Maya®
2012



Autodesk®

What's New in Subscription Advantage Pack

Autodesk[®] Maya[®] 2012

©2011Autodesk, Inc. All rights reserved. Except as otherwise permitted by Autodesk, Inc., this publication, or parts thereof, may not be reproduced in any form, by any method, for any purpose.

Certain materials included in this publication are reprinted with the permission of the copyright holder.

The following are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and other countries: 3DEC (design/logo), 3December, 3December.com, 3ds Max, Algor, Alias, Alias (swirl design/logo), AliasStudio, AliasWavefront (design/logo), ATC, AUGI, AutoCAD, AutoCAD Learning Assistance, AutoCAD LT, AutoCAD Simulator, AutoCAD SQL Extension, AutoCAD SQL Interface, Autodesk, Autodesk Intent, Autodesk Inventor, Autodesk MapGuide, Autodesk Streamline, AutoLISP, AutoSnap, AutoSketch, AutoTrack, Backburner, Backdraft, Beast, Built with ObjectARX (logo), Burn, Buzzsaw, CaiCE, Civil 3D, Cleaner, Cleaner Central, ClearScale, Colour Warper, Combustion, Communication Specification, Constructware, Content Explorer, Dancing Baby (image), DesignCenter, Design Doctor, Designer's Toolkit, DesignKids, DesignProf, DesignServer, DesignStudio, Design Web Format, Discreet, DWF, DWG, DWG (logo), DWG Extreme, DWG TrueConvert, DWG TrueView, DXF, Ecotect, Exposure, Extending the Design Team, Face Robot, FBX, Fempro, Fire, Flame, Flare, Flint, FMDesktop, Freewheel, GDX Driver, Green Building Studio, Heads-up Design, Heidi, HumanIK, IDEA Server, i-drop, Illuminate Labs AB (design/logo), ImageModeler, iMOUT, Incinerator, Inferno, Inventor, Inventor LT, Kynapse, Kynogon, LandXplorer, LiquidLight, LiquidLight (design/logo), Lustre, MatchMover, Maya, Mechanical Desktop, Moldflow, Moldflow Plastics Advisers, MPI, Moldflow Plastics Insight, Moldflow Plastics Xpert, Moondust, MotionBuilder, Movimento, MPA, MPA (design/logo), MPX, MPX (design/logo), Mudbox, Multi-Master Editing, Navisworks, ObjectARX, ObjectDBX, Opticore, Pipeplus, PolarSnap, PortfolioWall, Powered with Autodesk Technology, Productstream, ProMaterials, RasterDWG, RealDWG, Real-time Roto, Recognize, Render Queue, Retimer, Reveal, Revit, RiverCAD, Robot, Showcase, Show Me, ShowMotion, SketchBook, Smoke, Softimage, SoftimageXSI (design/logo), Sparks, SteeringWheels, Stitcher, Stone, StormNET, StudioTools, ToolClip, Topobase, Toxik, TrustedDWG, U-Vis, ViewCube, Visual, Visual LISP, Volo, Vtour, WaterNetworks, Wire, Wiretap, WiretapCentral, XSI.

ACE™, TAO™, CIAO™, and CoSMIC™ are copyrighted by Douglas C. Schmidt and his research group at Washington University, University of California, Irvine, and Vanderbilt University, Copyright (c) 1993-2009, all rights reserved.

mental ray is a registered trademark of mental images GmbH licensed for use by Autodesk, Inc.

Intel is a registered trademark or trademark of Intel Corporation or its subsidiaries in the United States and other countries.

OpenGL is a trademark of Silicon Graphics, Inc. in the United States and other countries.

Python is a registered trademark of Python Software Foundation.

All other brand names, product names or trademarks belong to their respective holders.

Disclaimer

THIS PUBLICATION AND THE INFORMATION CONTAINED HEREIN IS MADE AVAILABLE BY AUTODESK, INC. "AS IS." AUTODESK, INC. DISCLAIMS ALL WARRANTIES, EITHER EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE REGARDING THESE MATERIALS.

Contents

Chapter 1	What's New in General	1
	Import and export Alembic files	2
	Alembic > Import Alembic	3
	Alembic > Export	4
	Run Maya with a Japanese or Simplified Chinese user interface	7
Chapter 2	What's New in Basics	9
	Using the Node Editor	10
	Displaying connections	10
	Create a node	12
	Creating 2D Textures and selecting a mapping method	14
	Locking the Node Editor view	14
	Changing view mode for nodes	15
	Resizing nodes	16
	Renaming Nodes	17
	Selection in the Node Editor	18
	Pinning a node to the Node Editor view	20
	Zooming in and out of the Node Editor	21
	Disabling swatch rendering	23
	Connect nodes	24
	Connect nodes by dragging connection lines	24
	Connect nodes using their default connections	29

	Connect nodes using the Connection Editor	29
	Connect default output to a specific input attribute	30
	3	0
	Change existing node attribute connections	30
	Delete a connection	32
	Connection colors in the Node Editor	32
	Displaying a node's Attribute Editor	34
	Create customizable marking menus	34
	Creating and using bookmarks in the Node Editor	35
	Assign materials to surfaces	37
	Node Editor	38
	Node Editor menu and toolbar	44
	Node Editor marking menus	53
	Node Editor Preferences	55
Chapter 3	What's New in Rendering and Render Setup	59
	UV multi-render pass workflow	60
Chapter 4	What's New in Dynamics	63
	Load the MayaBullet plug-in	64
	Index	65

What's New in General

1

Alembic file format support



You can now load, save, and play back Maya scenes as Alembic-based cache files. The Alembic file format is an open-source format developed for exchanging complex 3D geometry data. Alembic files are highly portable and application independent so they can be shared, processed, and played back by a number of content creations applications.

In Maya, Alembic caches provide a number of performance improvements, including quicker loading time of large scenes, faster play back of complex character animations, and real-time play back of geometry data with topology changes.

Alembic files also let you easily share large scenes between various areas of your production pipeline without the large memory overhead of fully editable scene files.

Options in a new **Alembic** menu, located in the Maya main menu bar, let you configure settings for loading and saving Alembic files.

- [Import and export Alembic files](#) (page 2)

-  Watch: [Alembic file import and export](#)

Run Maya with a Simplified Chinese interface



Maya is now available with a Simplified Chinese interface. For instructions on how to run Maya with a Simplified Chinese interface on a different operating systems, see [Run Maya with a Japanese or Simplified Chinese user interface](#) (page 7).

Import and export Alembic files



Overview

You can save and load Maya scene files as Alembic cache files. The Alembic file format is an open-source format developed for exchanging complex 3D geometry data. Alembic files are highly portable and application independent so they can be shared, processed, and played back by a number of content creations applications.

In Maya, Alembic caches provide a number of performance improvements, including quicker loading time of large scenes, faster play back of complex character animations, and real-time play back of geometry data with topology changes. Complex scenes and animations can be exported as an Alembic file, and then re-imported into Maya to improve play back performance and reduce memory use. This workflow lets you easily pass complex scene data to various areas of your production pipeline, such as between animation and simulation or animation and lighting, without the large overhead of fully editable scenes.

For example, you can export a polygon mesh created from an nParticle effect to an Alembic file, then play the simulation back in Maya solely as a polygon object.

You can use the **Alembic Import** and **Alembic Export** windows to configure settings for loading and saving Alembic files. You can access the **Alembic Import** and **Alembic Export** windows from the **Alembic** menu located on Maya main menu bar. You can also use MEL commands to import and export Alembic files.

See [Alembic > Alembic Import](#) (page 3) and [Alembic > Export](#) (page 4).

To import and export Maya scenes as Alembic files, the **AlembicImport.mll** and **AlembicExport.mll** plug-ins must be loaded. Open to the **Plug-in Manager (Windows > Settings/Preferences > Plug-in Manager)** to ensure these plug-ins are loaded into Maya.

For information about loading plug-ins see [Load or unload Maya plug-ins](#).

Use MEL commands to import and export Alembic files

You can use the `AbcImport` and `AbcExport` MEL commands to import and export Alembic files. For example, use the following to export a polygon torus:

```
AbcExport-j "-frameRange 1 120 -root pTorus1 -file c:/temp/test.abc";
```

NOTE When specifying a file path for the exported file in the `-f` argument, the path cannot contain spaces. For example, you cannot specify:

```
C:\My Documents\maya\2012\Alembic Files.
```

You can instead use `C:\maya\2012\AlembicFiles.`

Help for the commands is available in the **Script Editor** using either

```
AbcExport -h; and AbcImport -h;.
```

Related topics

- [Alembic > Alembic Import](#) (page 3)
- [Alembic > Export](#) (page 4)

Alembic > Import Alembic



Loads Alembic files into the existing scene.

Alembic > Import Alembic >

Opens the **Alembic Import** window from which you can set options for loading an Alembic file.

Mode

Open Opens the selected Alembic file.

Import Imports the selected Alembic file into the current scene.

Replace Replaces the selected polygon and NURBS object(s) with the Alembic-based object(s) contained in the imported file. Only one instance of each Alembic-based object is imported, even when multiple objects are selected in the scene.

Fit Time Range Updates the timeslider range of the current scene to match the timeslider range of the imported Alembic file.

Set Current Time to Start Frame Sets the current time of the scene to the start frame of the imported Alembic file.

Insert Under Current Selection Parents an object selected in the scene to the imported Alembic-based object(s). If multiple objects are selected, the Alembic-based objects become children of the object that is selected first.

Debug Prints debug information to the **Script Editor** window.

Related topics

- [Import and export Alembic files](#) (page 2)
- [Alembic > Export](#) (page 4)

Alembic > Export



Exports either all objects or selected objects to an Alembic cache file.

Alembic > Export All in Alembic > and **Alembic > Export Selection in Alembic >**

Opens the **Alembic Export** window from which you can set options for exporting Alembic files. Selecting **Export All in Alembic** exports all polygon, NURBS, and subdivision objects in the scene to an Alembic file. Selecting **Export Selection in Alembic** exports selected objects to an Alembic file.

General Options

Cache time range

Render Settings Specifies that your current **Render Settings** determine the **Cache Time Range**.

Time Slider Specifies that the current playback range in your Time Slider determines the **Cache Time Range**.

Start/End Specifies that the range of frames between (and including) the specified **Start** and **End** times determines the **Cache Time Range**.

Start/End These fields let you specify a **Start** and **End** time for the **Cache Time Range**.

Evaluate every When creating an Alembic file, one sample of your object's transformations is taken every frame and saved to the Alembic file by default. **Evaluate every** specifies how often samples are taken during file creation. For example, a value of 2 caches the transformations of the current object at every other frame of the **Cache Time Range**.

Pre Roll Start Frame Specifies at which frame to start the scene evaluation. Use this option to set the starting frame for time dependent translations that require run-ups

Advanced Options

Don't Skip Unwritten Frames Evaluates every frame regardless of whether or not there are changes in object translation values between frames.

Verbose Outputs frame number information to the **Script Editor** or output window.

No Normals When on, normal data from the original polygon objects is not included in the exported Alembic cache file.

Renderable When on, any non-renderable nodes or hierarchy, such as hidden objects, are not included in the Alembic file.

Strip Namespaces When on, any namespaces associated with the exported objects are removed from the Alembic file. For example, an object with the namespace `taco:foo:bar` appears as `bar` in the Alembic file.

UV Write When on, UV data from polygon meshes and subdivision objects are written to the Alembic file. Only the current UV map is included.

Whole Frame Geo When on, geometry data at whole frames is sampled and written to the file. When off, geometry data is sampled at sub-frames and written to the file. This is the default setting.

World Space When on, the top node in the node hierarchy is stored as world space. By default, these nodes are stored as local space.

Write Visibility When on, the visibility state of objects is stored in the Alembic file. Otherwise, all objects are considered visible.

Callbacks

A recommended way to use callbacks with the Alembic export, is to first define a global procedure. For example, in the **Script Editor**, define the following:

```
global proc perFrameCallback(int $frame)
{
    print $frame;
}
```

The full MEL callback is then:

```
perFrameCallback(#FRAME#)
```

NOTE

For correct syntax, callback commands cannot contain the following:

- Spaces between characters. For example, ().
 - Single quotes. For example, (').
 - Double quotes. For example, (").
-

Per Frame Callback MEL When each frame is evaluated, the string specified is evaluated as a Mel command. For example, `print("#FRAME#")`.

Post Job Callback MEL When the translation has finished, the string specified is evaluated as a Mel command. For example, `print("Done!")`.

Per Frame Callback Python When each frame is evaluated, the string specified is evaluated as a python command. For example, `print("#FRAME#")`.

Post Job Callback Python When the translation has finished, the string specified is evaluated as a python command. For example, `print("Done!")`.

Related topics

- [Import and export Alembic files](#) (page 2)
- [Alembic > Alembic Import](#) (page 3)

Run Maya with a Japanese or Simplified Chinese user interface



The Maya user interface automatically appears in Japanese/Simplified Chinese when launched within a Japanese/Simplified Chinese operating system environment. If you wish to run Maya with a Japanese or Simplified Chinese UI on a system that is enabled to run in multiple languages, you must set the operating system preferences for that operating system. The steps for doing this differ depending on the operating system:

To run Maya in Japanese/Simplified Chinese on a Windows operating system (Windows XP)

- 1 From the **Start** menu, select **Settings > Control Panel**, and double-click the **Regional and Language Options**.
- 2 In the **Regional and Language Options** window, click the **Advanced** tab and set the **Language for non-Unicode programs** setting to **Japanese** or **Chinese (PRC)**.
- 3 Click **OK**.

The Windows operating system will indicate it needs to be restarted for the changes to take effect.

- 4 Once Windows restarts, you can launch Maya and the Maya user interface will appear in Japanese or Simplified Chinese, depending on your setting.

To run Maya in Japanese/Simplified Chinese on a Windows operating system (Windows 7)

- 1 From the **Start** menu, select **Control Panel**, and select the **Clock, Language, and Region** option; then select **Region and Language**.
- 2 From the **Regional and Language** window, click the **Administrative** tab and select **Change system locale**. Change the **Current system locale** setting to **Japanese** or **Chinese (Simplified, PRC)**.
- 3 Click **OK**.

The Windows operating system will indicate it needs to be restarted for the changes to take effect.

- 4 Once Windows restarts, you can launch Maya and the Maya user interface will appear in Japanese or Simplified Chinese, depending on your setting.

To run Maya in Japanese/Simplified Chinese on a Mac OS X operating system

- 1 From the Apple menu, select **System Preferences**.
- 2 In the Personal section of the **System Preferences**, click the Language and Text icon.
- 3 In the **Languages** list, click-drag the text titled `Japanese` or `Simplified Chinese` upwards so it appears at the top of the list.

The next time you launch Maya, the user interface will appear as Japanese or Simplified Chinese, depending on your setting.

NOTE

If you want to run the English language user interface of Maya when operating on Mac OS X, you can click-drag the text titled `English` that appears in the list of languages up to the top of the Languages list. The next time Maya is launched; the English language user interface will appear.

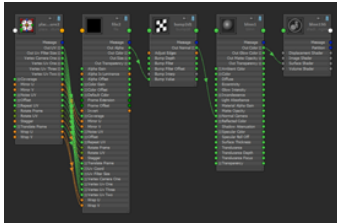
Related topics

- Run Maya in English within a Japanese environment
- Japanese Maya Overview
- Prepare Maya files containing Japanese text

What's New in Basics

2

Node Editor



The **Node Editor** presents an editable schematic of the dependency graph, displaying nodes and the connections between their attributes. It allows you to view, modify and create new node connections. You can access the **Node Editor** from the **Windows** menu. (**Window > Node Editor**).

You can graph the network of nodes of your choice, and build networks by using the Tab key and dragging lines from one node to another. Using this method, you can easily edit existing networks. All attribute ports and connection lines are represented by different colors so that connections can be easily read. In addition, you can create bookmarks to return to previous graph layouts for easy navigation.

- [Node Editor](#) (page 38)

- [Using the Node Editor](#) (page 10)

-  [Watch: Node Editor: Creating a shading network](#)

-  [Watch: Node Editor: Sample rigging workflow](#)

-  [Watch: Node Editor: Synchronizing selection and creation](#)

Using the Node Editor


Displaying connections



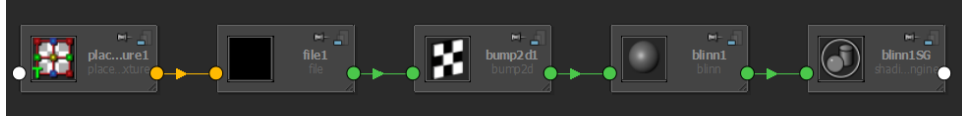
When you first open the **Node Editor**, it displays empty. To graph the node network of your choice, see the following sections.

Displaying node connections

To display node connections, do as follows:


- 1 Select any node or object in Maya, and click the  icon in the **Node Editor** toolbar to display the input and output node connections.

For a legend of what each connection color represents, see [Connection colors in the Node Editor](#) (page 32).

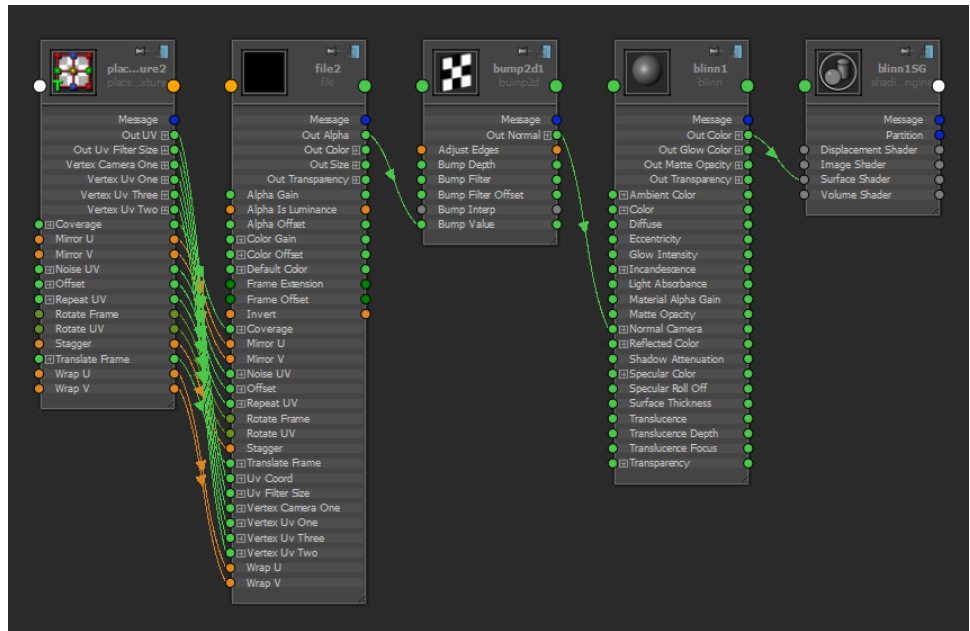


Displaying node attribute connections

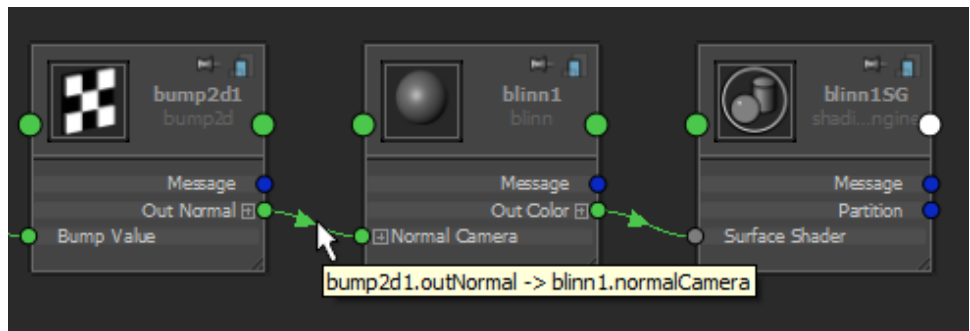
To display the attribute connections between nodes, do as follows:

- 1 Without selecting any nodes, click the  icon on the toolbar to expand each node to its full mode.

You can now see all the attributes of each node and how they are connected to each other.



TIP You can also mouse over a connection to see its details.



Order of the attributes


The attributes for each node are ordered as follows:

- 1 Attributes that can act as both input and output.
- 2 Attributes that can act only as output.
- 3 Attributes that can act only as input.

Graph shading network for selected object(s)

You can graph the shading network for the object(s) selected in the scene view.

- 1 In the scene view, select the object(s) for which you want to graph the shading network.

- 2  on any empty area in the **Node Editor** to access the work area marking menu and select **Graph Materials on Viewport Selection**.

Related topics

- [Node Editor menu and toolbar](#) (page 44)
- [Node Editor marking menus](#) (page 53)

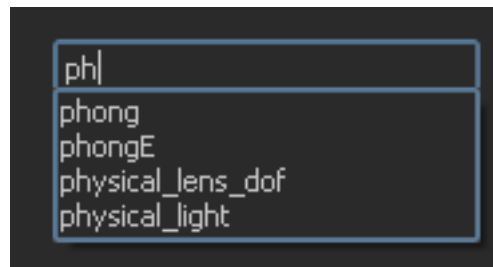
Create a node



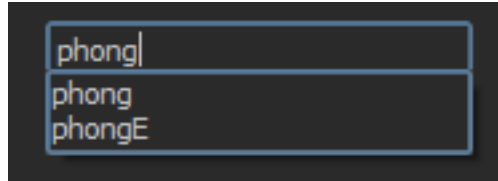
To create a node, do as follows:

Create a node (Method 1):

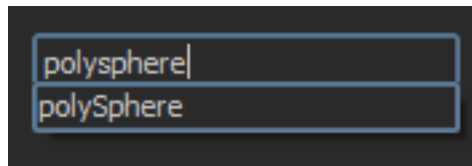
- 1 Click Tab and enter the type of your node.
You can enter the keyword for the node that you want to create. As you enter a keyword, the **Node Editor** updates in real time and only nodes with names that match what you entered are displayed.



TIP Click Tab to select the first node in the list of available nodes. In the example above, clicking Tab selects phong. Click Tab once again to create the phong node.




TIP If there is only one node that matches what you have entered, then clicking Tab automatically creates the node. In this example, a polySphere node is created automatically when you click Tab.



NOTE If you attempt to create a node type that does not exist in Maya, an error message appears informing you that the node cannot be created. The Tab creation UI then remains open for you to correct the error.

Create a node (Method 2):

- 1  any empty area in the **Node Editor** to access the work area marking menu and select **Create Node**.
Alternatively, you can select **Window > Create Node** to open the **Create Node** window.
- 2 Middle-drag a node into the **Node Editor**, and drop it in the desired location. Alternatively, you can click the node that you want and it will appear in the **Node Editor**.

Create a node (Method 3):

- 1 Select **Window > Rendering Editors > Hypershade** to open the **Hypershade**.
- 2 From the **Create** bar in the **Hypershade**, middle-drag a node into the **Node Editor**, and drop it in the desired location. Alternatively, you can click the node that you want and it will appear in the **Node Editor**.

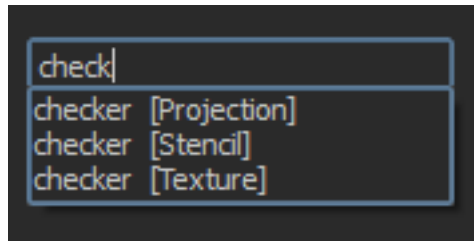
NOTE You can copy multiple nodes from the **Hypershade** to the **Node Editor** by selecting them in the **Hypershade** and then dragging and dropping them to the **Node Editor**.

Creating 2D Textures and selecting a mapping method



When creating a 2D texture, you can select among one of three mapping methods: create normally (create texture), create as projection, and create as stencil.

When creating a node in the **Node Editor** using the Tab key, as you enter the name of your desired 2D texture type, three variations of the texture appear (Projection, Stencil, Texture), allowing you to select among the three mapping methods. See [Create a node](#) (page 12) for more information on how to create a node using the Tab key.



Related topics


- Mapping methods

Locking the Node Editor view





You can lock your **Node Editor** view so that any new nodes you create do not appear in the graph.



Click the lock icon  to lock your view. The **Node Editor** view is unlocked by default.



When in locked mode , you can still use the following methods to create new nodes, and they will appear in the **Node Editor**:

- Tab key. See [Create a node](#) (page 12) for more information.
-  a node from the **Hypershade** or **Create Node** window or **Outliner** to the **Node Editor** (drag and drop creation).

NOTE You can still perform delete and undo in locked mode.

However, nodes created in the scene view, or by clicking in the **Create** Bar in the **Hypershade** and **Create Node** window (in other words, *not* drag and drop creation), do not appear in the **Node Editor**.




TIP You can also lock the view by disabling the **Options > Add to Graph on Create** option under the **Node Editor Options** menu or from the **Node Editor** work area marking menu. See [Node Editor Options menu](#) (page 46) and [Node Editor marking menus](#) (page 53) for more information.

Changing view mode for nodes









There are three ways to view nodes within the **Node Editor**: simple, connected and full. See [Node Editor view modes](#) (page 38) for more information on the view modes.

Changing the view mode for individual nodes

You can change the view mode for each node by clicking the , , or  icons on the top right corner of each node.

Changing the view mode for selected nodes

Select the nodes for which you want to change the view mode and click the , , or  icons on the **Node Editor** toolbar.

Alternatively, you can select the **Edit > Simple Mode** (+S), **Edit > Connected Mode** (+C), or **Edit > Full Mode** (+F) options from the **Node Editor** menu bar. See [Node Editor menu bar](#) (page 45) for more information.

Changing the view mode for all nodes

When none of the nodes are highlighted (active) in the **Node Editor**, clicking the icons in the toolbar, or selecting the same options from the **Node Editor Edit** menu, or using the **Edit** menu shortcuts, affects all nodes.


NOTE

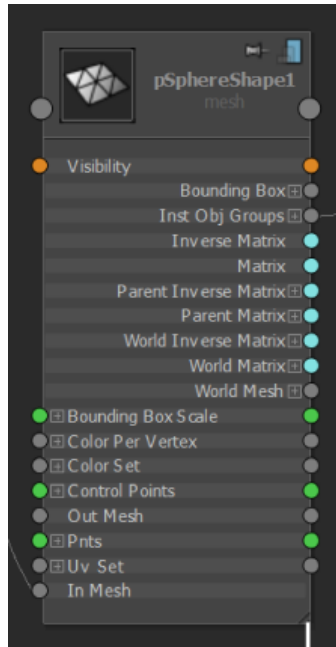
You can also customize this option in the **Preferences** window, and your nodes are displayed in the view mode you selected the next time you create a **Node Editor** window. See [Node Editor Preferences](#) (page 55) for more information.

Resizing nodes



You can resize any node in any of the view modes: simple, connected or full.

The resize corner  on the bottom right corner of a node denotes that it can be resized.



indicates node can be resized

Renaming Nodes



You can rename the nodes in the **Node Editor** using either of these two methods:

Method 1

- Double-click on the node name in the node to rename it.

Method 2

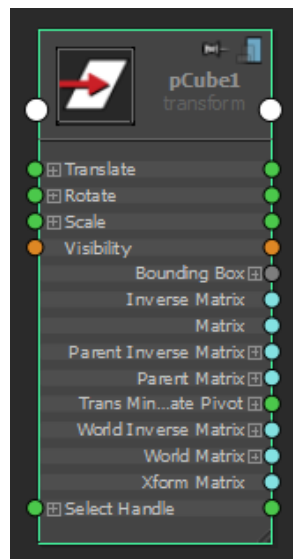
- Right-click on the node to access the node marking menu and select **Rename**.

Selection in the Node Editor

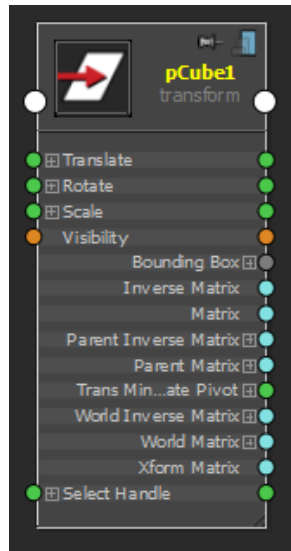


There are four types of selection states in the **Node Editor**. A node can be:

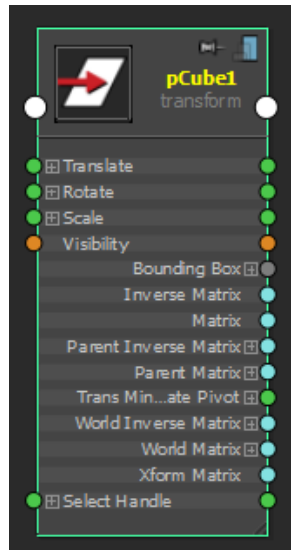
- selected in the scene view but not in the **Node Editor**



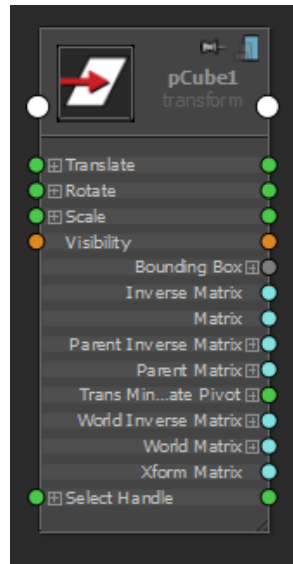
- highlighted in the **Node Editor**, but not selected in the scene view



- selected in the scene view and highlighted in the **Node Editor** (in sync)



- unselected in the scene view and not highlighted in the **Node Editor** (in sync)






NOTE Selected states are always outlined in green. An outline of white indicates that the node is active in the **Node Editor** but not selected in the scene.

You can customize whether or a not a node is also selected in the scene view when it is highlighted in the **Node Editor**, and vice versa. Select **Options > Sync Node Editor/Model Selection** so that, if you click on a node in the **Node Editor**, the same node is also selected in the scene view and vice versa. You can also select this option via the work area marking menu. For more information about marking menus, see [Node Editor marking menus](#) (page 53).

You can also set this option as a preference via **Windows > Settings/Preferences > Preferences > Settings > Node Editor**. If you select this option under the **Preferences** window, the changes are applied the next time you open the **Node Editor**.



Pinning a node to the Node Editor view





You can click the pin icon  to pin a node so that it stays within the **Node Editor** view and retains its position even after a regraph.  indicates that the node is pinned and  indicates that it is unpinned.

Pinning/unpinning multiple nodes

You can select multiple nodes and pin or unpin them all at once. Select

multiple nodes and then click the  icon in the toolbar to pin or the  icon to unpin.

TIP If none of the nodes are highlighted (active) in the **Node Editor**, then clicking



the  and  icons affects all of the nodes.

Zooming in and out of the Node Editor



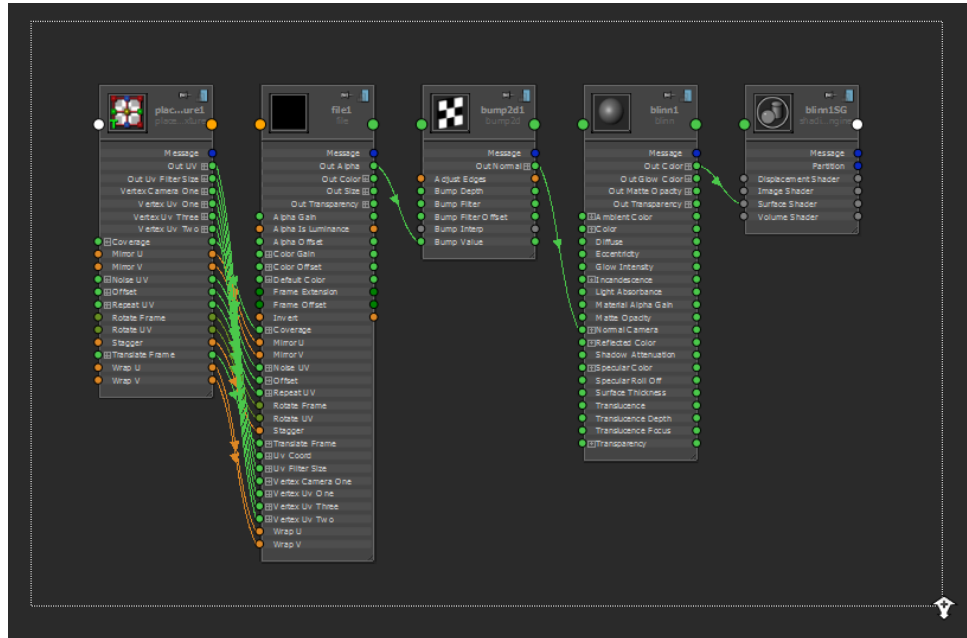
Zooming in

You can zoom into the **Node Editor** in one of the following ways:

- Marquee select zoom:  +  +left mouse drag (from left to right) to create a marquee box to zoom into.



As you drag a box shape, the zoom in icon appears at the bottom right corner to denote that you are zooming in.





- Selecting **View > Zoom In**

- Using the hotkey +

Zooming out

You can zoom out of the **Node Editor** in one of the following ways:

- Marquee select zoom:  +  + left mouse drag (from right to left) to create a marquee box to zoom out of.



As you drag a box shape, the zoom out icon appears at the top left corner to denote that you are zooming out.

- Selecting **View > Zoom Out**
- Using the hotkey -

Disabling swatch rendering



You can disable swatch rendering in the **Node Editor**, **Hypershade** and **Attribute Editor** to help improve performance slowdowns caused by swatch refresh.

To disable swatch rendering in the Node Editor, Hypershade and Attribute Editor for the startup state of Maya

- 1 Select **Window > Settings/Preferences > Preferences** to open the **Preferences** window. Under **Display**, disable **Render swatches (startup)**.

To disable swatch rendering in the Node Editor, Hypershade and Attribute Editor for the current session of Maya

- 1 Disable **Options > Render Swatches** from the **Node Editor Options** menu.

NOTE As soon as you disable swatch rendering, all existing swatches should stop updating. Swatches that are already drawn remain visible but are not updated.

TIP You can disable swatch rendering via the **Preferences** window so that swatch rendering is disabled upon startup, then override this option for your session by enabling this option in the **Node Editor Options** menu.

Connect nodes


Connect nodes by dragging connection lines



Method 1

To connect attributes using connection lines

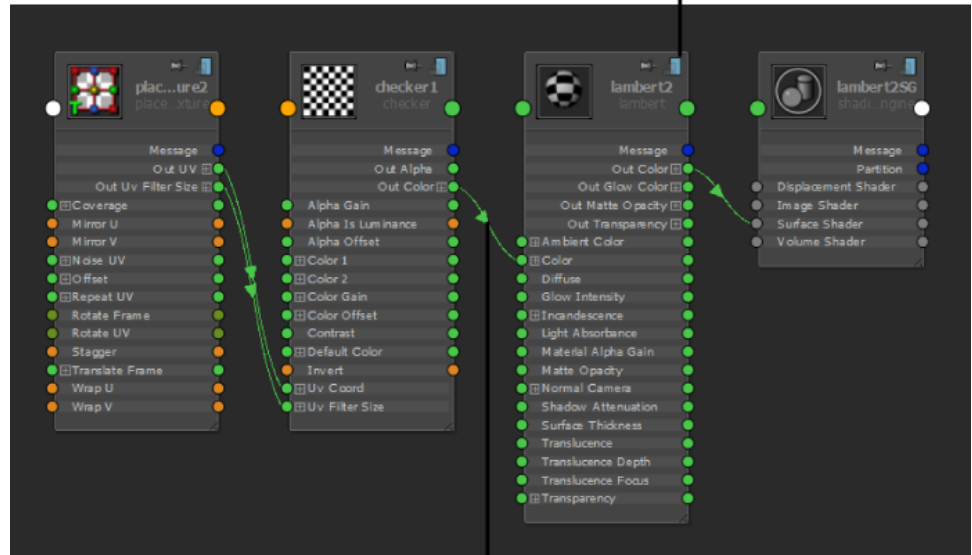


- 1 In the **Node Editor**, click on the  icon on the top right hand corner of each node to expand it to its full mode.
- 2 Middle-mouse or left-mouse drag to create connection lines between your output and input attributes.

If you mouse over a compound attribute, the **Node Editor** automatically expands to expose its children to which you can make a connection.

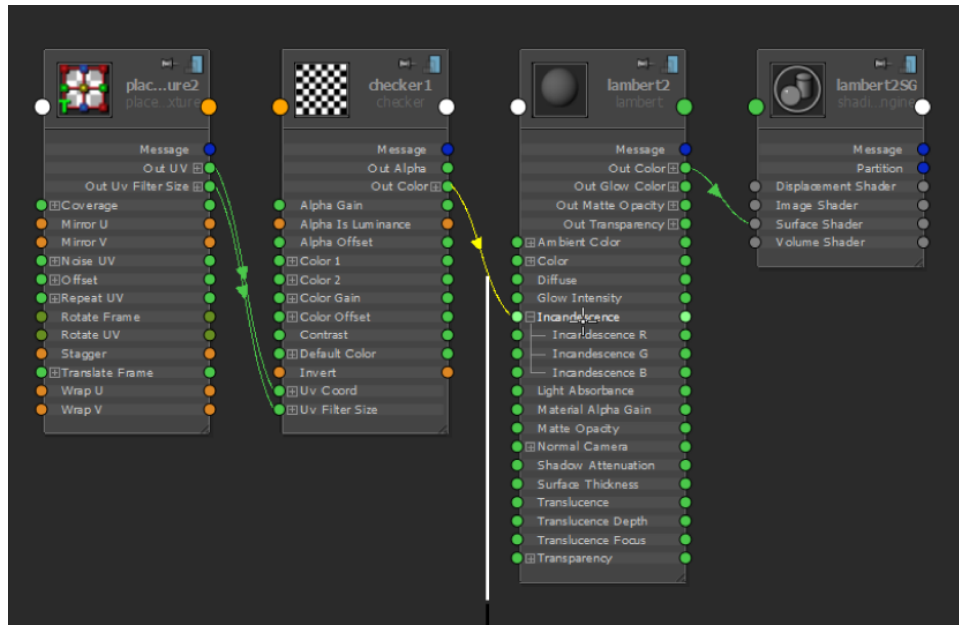
In addition, it also snaps to the ports to which you can make a connection. This allows you to easily identify the attributes that you can connect to.

Click to change to the full mode to expose all attributes



Use the left or middle-mouse button to create a connection line between the output and input attributes

NOTE As you mouse over the available inputs, all inputs that can receive the connection are highlighted. This indicates that it is possible to make a connection with this input.



Incandescence is highlighted when you mouse over it because a connection is possible between Out Color and Incandescence. As you mouse over Incandescence, it automatically expands itself to expose its children so that you can make a connection.

TIP When connecting compound attributes (for example, a parent attribute such as Color, with Color R, G, B child attributes), if you try to make a connection to a parent attribute (Color) when the child attribute (for example, Color R) is already connected, the connection to the child attribute will become disconnected.

Method 2

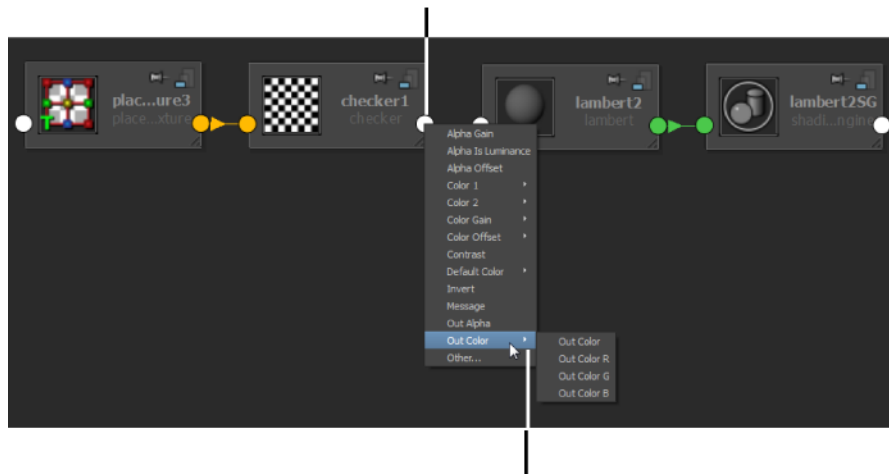
To connect attributes without expanding the nodes

- 1 Click the output master port of the node to display the menu of available output attributes. A list of common output attributes is displayed; or, select **Other** to display the **Connection Editor, Output Selection** window. From this window, you can select the output of your choice.
- 2 Drag the connection line to the input master port of the node receiving the input connection. A list of common input attributes is displayed; or,

select **Other** to display the **Input Selection** window. From this window, you can select the input of your choice.

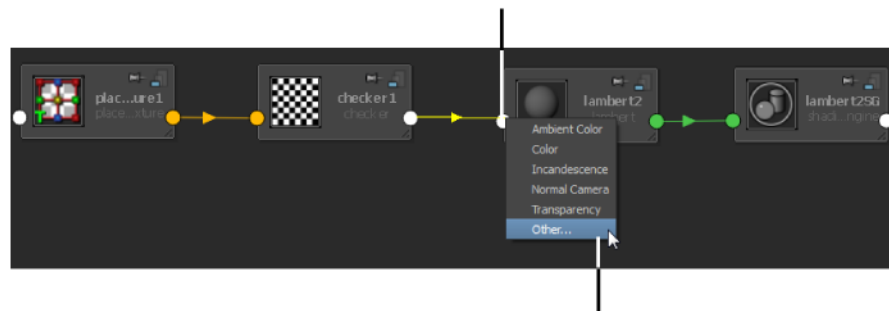
NOTE The **Input Selection** window only lets you select an attribute that is compatible with the previously selected output attribute.

Click to display a list of available output attributes.



A list of common output attributes appear; or select Other to display the Output Selection window.

Drag connection line to this port to display a list of available input attributes.



A list of common input attributes appear; or select Other to display the Input Selection window.

Related topics


For more information regarding the different views of the nodes, see [Node Characteristics](#) (page 38).

For more information regarding the connection lines in the **Node Editor**, and what each color represents, see [Connection colors in the Node Editor](#) (page 32).


Connect nodes using their default connections



To connect nodes using their default connections (Method 1):

- 1 Ctrl+  a node onto another in the **Node Editor**. The default connection is created.


To connect nodes using their default connections (Method 2):

- 1 Ctrl+  a node from the **Node Editor** onto a node in the **Hypershade**. The default connection is created.


Connect nodes using the Connection Editor



Connect nodes using the Connection Editor (Method 1):

- 1 Shift+  a node onto another in the **Node Editor**. The **Connection Editor** appears so you can select your output and input attributes.


Connect nodes using the Connection Editor (Method 2):

- 1 Shift+  a node from the **Node Editor** onto a node in the **Hypershade**. The **Connection Editor** appears so you can select your output and input attributes.


Connect default output to a specific input attribute



Method 1:

-  a node from the **Node Editor** into a field in the **Attribute Editor**.

Method 2:

-  a node from the **Node Editor** onto a node in the **Hypershade**.
Select the attribute you want to connect to from the pop-up menu that appears.

Change existing node attribute connections

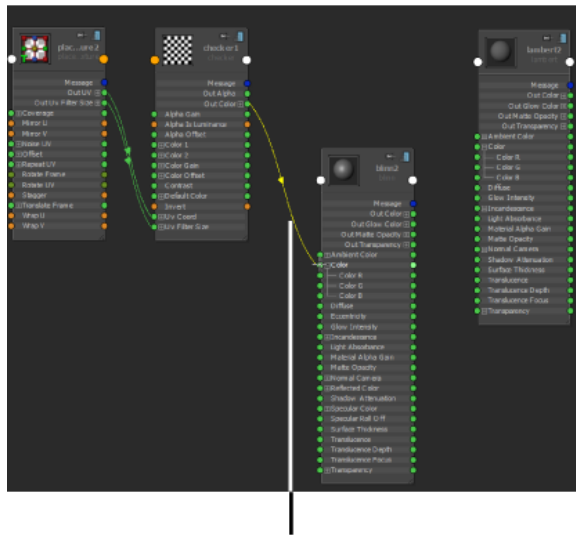
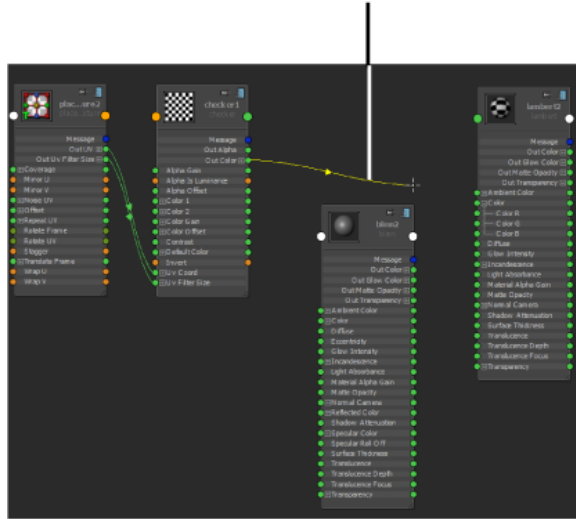


You can drag on a connection line to break and recreate a connection.

Reconnecting an input to a different output

- 1 Drag on a connection line from the output side to disconnect it, and reconnect it to a different output.

Drag to disconnect...



and then reconnect.

Reconnecting an output to a different input

- 1 Drag on a connection line from the input side to disconnect it, and reconnect it to a different input.

Delete a connection



To delete a connection (Method 1)

- 1 Drag on a connection line and release the mouse in the background to delete a connection.

To delete a connection (Method 2)

- 1 Click the connection line(s) you want to delete (marquee-select to select more than one).
The lines are selected.
- 2 Press Backspace (Windows and Linux) or Delete (Mac OS X) to delete the connections.

The lines are deleted and the swatches in the **Node Editor** update to show the changes.

To delete a connection (Method 3)


- 1 Press  ,  ,  to draw a line in the **Node Editor**.
Release to delete the connections crossed by the line.


Connection colors in the Node Editor



All ports and connection lines are represented by different colors to denote the attribute type for the input or output, for example, integer, string, boolean and so forth. The connection line takes on the color of the output.


The following is a legend of the various connection colors in the **Node Editor**.

Numeric (double, float: for example, 2 values of double, 3 values of float, and so forth) 

Integer (for example, 2 values of 16-bit integers or 3 values of 32-bit integers and so forth) 

Boolean 

String 

Multi-ambiguous (for example, compound or enum attributes) 

Curve 

Rotation 

Matrix 

Color 

Array



Displaying a node's Attribute Editor



You can display a node's **Attribute Editor** by double-clicking on it in the **Node Editor**.

Create customizable marking menus



You can customize the node marking menus. The following is an example using Python to add a custom node marking menu item to the plusMinusAverage node.

Create the plusMinusAverage node in the **Node Editor**, then run the following script in the **Script Editor**.

```
from maya.app.general import nodeEditorMenus
from maya import cmds

def addPMAMenuItems(ned, node):
    """
    Check for plusMinusAverage node and create necessary
    menu items
    """
    type = cmds.nodeType(node)
    if type == 'plusMinusAverage':
        def doSomething(*args):
            cmds.confirmDialog(title="Something",
                message="Ok?")
            cmds.menuItem(label="PlusMinusAverage! Do
                something!", c = doSomething)
```

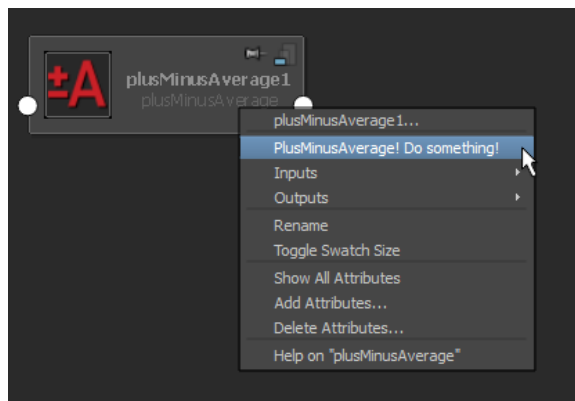
```

        return True
    else:
        return False

# register the custom callback with nodeEditor
#
nodeEditorMenus.customInclusiveNodeItemMenuCallbacks.append(addPMAMenuItems)

```

The plusMinusAverage node now includes the custom menu item "PlusMinusAverage! Do something!".



Related topics

- [Node Editor marking menu](#) (page 53)

Creating and using bookmarks in the Node Editor

You can create bookmarks in the **Node Editor** to store the current node graph as a quick reference.

Creating a bookmark

Creating a bookmark (Method 1)

- 1 Select **Bookmarks > Create Bookmark**.

The **Name Bookmark** window appears. You can provide a custom name for your bookmark; otherwise, it is assigned a default name.

Creating a bookmark (Method 2)

- 1 Click  on the toolbar.





The **Name Bookmark** window appears. You can provide a custom name for your bookmark; otherwise, it is assigned a default name.

Loading an existing bookmark

Loading an existing bookmark (Method 1)

- 1 Select **Bookmarks** > <bookmark name>.

Loading an existing bookmark (Method 2)

- 1 Click the  and  buttons in the toolbar to step through the bookmarks. Click  to load the previous and to  load the next bookmark.

Editing bookmarks

You can edit bookmarks by using the **Bookmark Editor**. Select **Bookmarks**

> **Bookmark Editor** or click  on the toolbar to open the **Bookmark Editor**.

Deleting a bookmark

- 1 Right-click the bookmark that you want to delete in the **Bookmark Editor** and select **Delete Bookmark**.


NOTE You can also delete all existing bookmarks by right-clicking and selecting **Delete All Bookmarks**.

Renaming a bookmark (Method 1)

- 1 Right-click the bookmark that you want to rename in the **Bookmark Editor** and select **Rename Bookmark**.

NOTE The **Name Bookmark** window appears and allows you to rename your bookmark.

Renaming a bookmark (Method 2)

- 1 Under the **Bookmarks** menu, click  beside the bookmark name.

NOTE The **Name Bookmark** window appears and allows you to rename your bookmark.

Updating a bookmark

You can replace an existing bookmark with your current graph layout as follows:

- 1 Right-click the bookmark that you want to update in the **Bookmark Editor** and select **Replace Bookmark**.

NOTE Now, when you select this bookmark via **Bookmarks** > <bookmark name>, the updated graph network is displayed.


Assign materials to surfaces



To assign a material to surfaces (Method 1)

- 1 In the scene view, select the object(s) that you want to assign the material to.
- 2 In the **Node Editor**, right-click the material to expose its marking menu and select **Assign Material to Viewport Selection**.

To assign a material to surfaces (Method 2)

- 1 From the **Node Editor**,  the material node over the surface(s) to which you want to assign the material.

Related topics

- [Node Editor marking menus](#) (page 53)

Node Editor




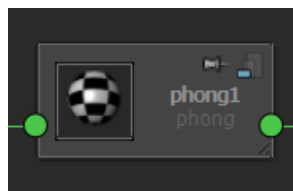
The **Node Editor** presents an editable schematic of the dependency graph, displaying nodes and the connections between their attributes. It allows you to view, modify and create new node connections. You can access the **Node Editor** from the **Windows** menu. (**Window** > **Node Editor**).

Node view modes


There are three ways to view nodes within the **Node Editor**: simple, connected and full.

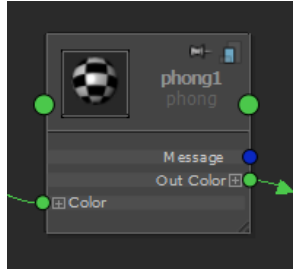
The simple mode shows one input and one output master port, represented

as circles. The  icon also denotes that you are in the simple mode.



The connected mode shows the input and output master ports, as well as any connected attributes. These attributes have their own ports, which are displayed

as small circles. The  icon also denotes that you are in the connected mode.




The full mode shows the input and output master ports, as well as primary node attributes. The set of primary attributes for a given node is based on the properties of the attributes such as whether they are keyable or dynamic. The



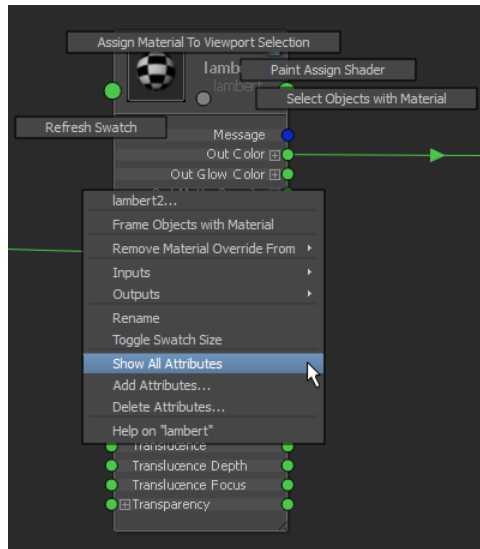
icon also denotes that you are in the full mode.

Input attributes are displayed on the left side of a node in full mode. Output attributes are displayed on the right side. To expand a node's attribute list,

click the  located next to the attribute. If an attribute can be connected as either an input or an output, a port is available on both the left and right sides of the node. For more information on node connections, see [Displaying connections](#) (page 10).



You can select the **Show All Attributes** option from the node marking menus to temporarily replace the full mode for a node with a listing of all of the attributes. The displaying of all of its attributes is only valid for this draw instance. If you regraph the node, or if you change the view mode of the node, you must re-select this option to see all of its attributes again. See [Node Editor Marking Menu](#) (page 53) for more information about the **Node Editor** marking menus.




Changing view modes for multiple nodes

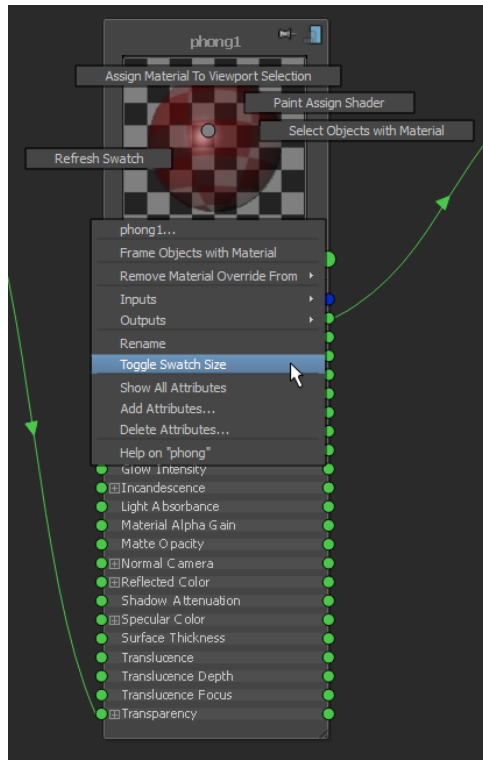
You can change the view mode for multiple nodes in one step by selecting the nodes and selecting the desired mode icon in the toolbar. You can also do the same via the menu options (**Edit > Simple Mode, Connected Mode, Full Mode**) or their shortcuts. See [Node Editor Edit menu](#) (page 45) and [Node Editor Toolbar](#) (page 48) for more information.

NOTE When none of the nodes are highlighted (active) in the **Node Editor**, clicking the icons in the toolbar, or selecting the same options from the **Node Editor Edit** menu, or using the **Edit** menu shortcuts, affects all nodes.

Toggling swatch size



In any view mode, you can toggle between a larger and a smaller version of









the node swatch by  the node and selecting **Toggle Swatch Size** from the node marking menu.







Functionality

The following table lists the functionality of the **Node Editor** at a glance.

To...	Do this
Display connections.	Select a node and click the  icon in the toolbar to display the input and output node connections.
Create a node.	Click Tab and enter the type of your node. Alternatively, you can  drag and drop from the Create bar in the Hypershade or the Create Node window.

To...	Do this
Copy a node into the Node Editor .	You can  a node from the Outliner to bring it into the Node Editor .
Create a connection using connection lines.	Expand the node to its full mode by clicking  . Middle-mouse or left-mouse drag to create connection lines between your output and input attributes.
Create a default node connection.	 +  a node onto another in the Node Editor .
Connect nodes using the Connection Editor .	 +  a node onto another in the Node Editor .
Delete a connection.	Drag on a connection line and release the mouse in the background area to delete a connection. Or, you can click the connection line(s)/marquee-select and press Backspace (Windows and Linux) or Delete (Mac OS X).
Change an existing connection.	Drag on a connection line to disconnect and then reconnect with a new input/output.
Lock the view.	Click the  icon to lock your view so that any new nodes you create do not appear in the graph. Or, you can disable the Add to Graph on Create option from the Node Editor work area marking menu.
Resize nodes.	You can resize any node in any of the view modes: simple, connected, or full. The resize corner  on the bottom right corner of a node denotes that it can be resized.

To...	Do this
Pinning a node to the view.	You can pin a node so that it stays within the Node Editor view and retains its position even after a regraph. Click  to pin and  to unpin. You can also pin or unpin multiple nodes in one step via the toolbar icons.
Connect a node to an attribute in the Attribute Editor .	 a node from the Node Editor onto a field in the Attribute Editor .
Assign a material.	In the scene view, select the object(s) that you want to assign the material to. Then, in the Node Editor , right-click the material to expose its marking menu and select Assign Material to Viewport Selection . Or, you can  a shader or material from the Node Editor onto an object in a model pane.

Related topics

- [Using the Node Editor](#) (page 10)
- [Connect Nodes](#) (page 24)
- [Node Editor Preferences](#) (page 55)
- Hypergraph
- Hypershade


Node Editor menu and toolbar



Node Editor menu bar

Edit menu

Simple Mode Change the view mode of the selected nodes to simple mode so that only the input and output master ports are shown. See [Node view modes](#) (page 38) for more information about the **Node Editor** view modes.


You can also do the same via the toolbar icons or via the  + S shortcut. You can also customize this option in the **Preferences** window, and your nodes are displayed in simple mode the next time you create a **Node Editor** window. See [Node Editor Preferences](#) (page 55) for more information.

Connected Mode Change the view mode of the selected nodes to connected mode so that the input and output master ports, as well as any connected attributes, are shown. See [Node view modes](#) (page 38) for more information about the **Node Editor** view modes. You can also do the same via the toolbar

icons or via the  + C shortcut.

You can also customize this option in the **Preferences** window, and your nodes are displayed in connected mode the next time you create a **Node Editor** window. See [Node Editor Preferences](#) (page 55) for more information.

Full Mode Change the view mode of the selected nodes to full mode so that the input and output master ports, as well as primary node attributes, are shown. The set of primary attributes for a given node is based on the properties of the attributes such as whether they are keyable or dynamic. See [Node view modes](#) (page 38) for more information about the **Node Editor** view modes.

You can also do the same via the toolbar icons or via the  + F shortcut. You can also customize this option in the **Preferences** window, and your nodes are displayed in full mode the next time you create a **Node Editor** window. See [Node Editor Preferences](#) (page 55) for more information.

NOTE When none of the nodes are highlighted (active) in the **Node Editor**, selecting any of these options or using their corresponding shortcuts will affect all nodes.

Pin When you pin a node, it stays within the **Node Editor** view and retains its position even after a regraph. You can pin multiple nodes in one step by selecting multiple nodes and then selecting this option. You can also do the same via the toolbar icons.

Unpin When you pin a node, it stays within the **Node Editor** view and retains its position even after a regraph. You can unpin multiple nodes in one step by selecting multiple nodes and then selecting this option. You can also do the same via the toolbar icons.

NOTE When none of the nodes are highlighted (active) in the **Node Editor**, selecting either the **Pin** or **Unpin** menu item will affect all nodes.

View menu

Frame All Adjusts the graph view to fit all objects in the editor. You can also use the hotkey A.

Frame Selection Adjusts the graph view to fit all selected objects in the editor. You can also use the hotkey F.

Zoom In Zoom into your graph. You can also use the hotkey +.

Zoom Out Zoom out of your graph. You can also use the hotkey -.

TIP If you find it difficult to read your node name when you are zoomed out of your graph, mouse over the node and a tool tip with the node name pops up.

Bookmarks menu

Create Bookmark Select this option to create a bookmark to store the current node graph as a quick reference.

Bookmark Editor Select this option to open the **Bookmark Editor** to edit your bookmarks. See [Bookmarks in the Node Editor](#) (page 35) for more information on editing bookmarks.

<bookmark name> A list of existing bookmarks is listed under this menu.

Select the bookmark name to load each bookmark. You can also click the beside each bookmark name to rename your bookmark.

Options menu

Render Swatches Disable this option to disable swatch rendering in the **Node Editor**, **Hypershade**, and **Attribute Editor** for the current session of Maya.

NOTE As soon as you disable swatch rendering, all existing swatches should stop updating. Swatches that are already drawn remain visible but are not updated.

You can also set this option in the **Preferences** window by selecting **Window > Settings/Preferences > Preferences > Display > Render swatches in editors** and your setting will be applied upon the startup of Maya. For

example, you can disable this option via the **Preferences** window to improve performance when loading a scene, then re-enable swatch rendering for your **Node Editor** session by enabling this option via the **Node Editor Options** menu.

Sync Node Editor / Model Selection Enable this option so that if you highlight a node in the **Node Editor**, the same node is also selected in the scene and vice versa. See [Selection in the Node Editor](#) (page 18) for more information.

Add to Graph on Create Enable this option so that when you create a node in the scene, it is also added to the **Node Editor**. If this option is disabled, only nodes that you create using the **Node Editor** (for example, using Tab creation or drag and drop creation; see [Create a Node](#) (page 12) for more information) are shown. You can also disable this option by clicking the lock



button to lock the view. See [Locking the view](#) (page 14) for more information.

NOTE You can also control the **Sync Node Editor / Model Selection** and **Add to Graph on Create** options by selecting **Windows > Settings/Preferences > Preferences > Settings > Node Editor**. If you select these options under the **Preferences** window, the changes are applied the next time you open the **Node Editor**. See [Node Editor Preferences](#) (page 55) for more information.

NOTE You can also select these options via the work area marking menu. See [Node Editor marking menus](#) (page 53) for more information.

Display menu

Customize your graph in the **Node Editor** by selecting any of the provided options.

Shapes Choose among the options to customize which shapes are displayed.

Transforms Select this option to include transform nodes in your graph.

Extend to shapes Select this option to include child shape nodes for selected transforms.

You can also select the **All Shapes**, **All Shapes Except Shading Group Members**, and **No Shapes** options from the **Node Editor** toolbar. See [Node Editor toolbar](#) (page 48) for more information.

You can also select these options via the work area marking menu. See [Node Editor marking menus](#) (page 53) for more information.

You can also control these options by selecting **Windows > Settings/Preferences > Preferences > Settings > Node Editor**. If you select this option under the **Preferences** window, the changes are applied the next time you open the **Node Editor**. See [Node Editor Preferences](#) (page 55) for more information.

Show menu

Objects Select which object types to show in the **Node Editor**. Select **Clear Below** to turn off all filters. You can also select these options via the work area marking menu. See [Node Editor marking menus](#) (page 53) for more information.

Show All Turns off all filters in the **Show** menu.

Show Selected Type(s) Shows only object types of the same type as the current selection. This option lets you create a filter that limits the types of objects displayed in the **Node Editor** to the same type of object as the current selection.

Show Auxiliary Nodes Shows node types that the **Node Editor** normally does not show because they are rarely needed (such as underworld nodes).

Auxiliary Nodes Lets you set which nodes are considered “auxiliary”.

You can customize which nodes are hidden to reduce clutter in the editors. Generally, these are nodes that are not normally useful to see or edit. Select this option to display the **Auxiliary Nodes** editor and customize the node types that are hidden in editors.

Help menu

Help Lists various learning options, including links to the Maya Help and the Learning Path.

Node Editor toolbar

Text Filter Indicator



Click to clear any applied filters (beyond the implicit one) and return the graph to its default contents.

Enter text to filter listing You can enter syntax to limit the node listing in the **Node Editor**. The syntax is case sensitive. For example, P* shows only nodes beginning with the letter P. Every listed child node is displayed with its corresponding parent node even if the parent node would not otherwise be shown because of the filter.

Display All Shapes



Customize your graph by selecting this option to display all shapes. You can click the icon to select this option, or select it from the **Display** menu. The option that you have selected is also reflected in the toolbar: the corresponding icon is highlighted.

You can also customize this option in the **Preferences** window, and your setting is applied the next time you create a **Node Editor** window. See [Node Editor Preferences](#) (page 55) for more information.

Display All Shapes Except Shading Group Members



Customize your graph by selecting this option to display all shapes except shading group members. You can click the icon to select this option, or select it from the **Display** menu. The option that you have selected is also reflected in the toolbar: the corresponding icon is highlighted.

You can also customize this option in the **Preferences** window, and your setting is applied the next time you create a **Node Editor** window. See [Node Editor Preferences](#) (page 55) for more information.

Display No Shapes



Customize your graph by selecting this option to refrain from displaying shapes. You can click the icon to select this option, or select it from the **Display** menu. The option that you have selected is also reflected in the toolbar: the corresponding icon is highlighted.

You can also customize this option in the **Preferences** window, and your setting is applied the next time you create a **Node Editor** window. See [Node Editor Preferences](#) (page 55) for more information.

Create a new bookmark



Select this option to create a bookmark to store the current node graph as a quick reference.



Open the Bookmark Editor window



Select this option to open the **Bookmark Editor** to edit your bookmarks. See [Bookmarks in the Node Editor](#) (page 35) for more information on editing bookmarks.

Load previous/next bookmark



Click  to load the previous and  to load the next bookmark.

Clear the graph



Clears the current **Node Editor** layout.

Remove selected nodes from graph



Customize the layout of your graph by removing the selected nodes. To remove a node from the graph, select it and click this icon.

Layout the graph



Takes the current collection of nodes being displayed in the **Node Editor** and re-arranges them.

Input connections



Displays only the input connections of selected nodes. See also [Connect Nodes](#) (page 24).

Input and output connections



Displays the input and output connections of selected nodes. See also [Connect Nodes](#) (page 24).

Output connections



Displays only the output connections of selected nodes. See also [Connect Nodes](#) (page 24).

Increase/decrease traversal depth




You can customize how far you can see the connections graphed in the **Node Editor**. By default, depth traversal is unlimited (set to -1).

Use the arrows or enter an integer into the numeric field of the menu bar to change the depth traversal value. Click the Unlimited button or set the number to -1 to remove the depth traversal limit.

Simple Mode



Change the view mode of the selected nodes to simple mode so that only the input and output master ports are shown. See [Node view modes](#) (page 38) for more information about the **Node Editor** view modes. You can also do the


same via **Edit > Simple Mode** or via the  + S shortcut.

You can also customize this option in the **Preferences** window, and your nodes are displayed in simple mode the next time you create a **Node Editor** window. See [Node Editor Preferences](#) (page 55) for more information.

Connected Mode



Change the view mode of the selected nodes to connected mode so that the input and output master ports, as well as any connected attributes, are shown. See [Node view modes](#) (page 38) for more information about the **Node Editor** view modes. You can also do the same via **Edit > Connected Mode** or via

the  + C shortcut.

You can also customize this option in the **Preferences** window, and your nodes are displayed in connected mode the next time you create a **Node Editor** window. See [Node Editor Preferences](#) (page 55) for more information.

Full Mode





Change the view mode of the selected nodes to full mode so that the input and output master ports, as well as primary node attributes, are shown. The set of primary attributes for a given node is based on the properties of the attributes such as whether they are keyable or dynamic. See [Node view modes](#) (page 38) for more information about the **Node Editor** view modes. You can

also do the same via **Edit > Full Mode** or via the  + F shortcut.

You can also customize this option in the **Preferences** window, and your nodes are displayed in full mode the next time you create a **Node Editor** window. See [Node Editor Preferences](#) (page 55) for more information.

NOTE When none of the nodes are highlighted (active) in the **Node Editor**, selecting any of the simple, connected or full mode icons will affect all nodes.

Pin selected nodes / Unpin selected nodes  /  You can select multiple nodes and pin or unpin them all at once. When you pin a node, it

stays within the **Node Editor** view and retains its position even after a

regraph. Select  to pin and  to unpin.

NOTE When none of the nodes are highlighted (active) in the **Node Editor**, clicking either the pin or unpin icon will affect all nodes.


Allow/disallow new nodes to be automatically added to the graph



Click this icon to lock your **Node Editor** view so that any new nodes you

create do not appear in the graph.  indicates the unlocked state and



 indicates the locked state. See [Locking the view](#) (page 14) for more information. You can also do the same by disabling the **Add to Graph on Create** option via the work area marking menu or the **Options** menu. See [Node Editor marking menus](#) (page 53) for more information.

Node Editor marking menus



In addition to the **Node Editor** toolbar and menu, the **Node Editor** marking menus provide an additional way for you to customize your **Node Editor** graph.

Work area marking menus

Access the work area marking menus by right-clicking on any empty area in the **Node Editor**.


Create Node Select this option to open the **Create Node** window.

Select All in Graph Select this option to highlight all of your nodes in your **Node Editor** graph.

Graph Materials on Viewport Selection Select this option to graph the shader(s) applied to the object(s) selected in your scene view.

Sync Node Editor/Model Selection Enable this option so that if you highlight a node in the **Node Editor**, the same node is also selected in the scene and vice versa. You can also select this option from the **Node Editor Options** menu. See [Selection in the Node Editor](#) (page 18) for more information.

Add to Graph on Create Enable this option so that when you create a node in the scene, it is also added to the **Node Editor**. If this option is disabled, only nodes that you create using the **Node Editor** (for example, using Tab creation or drag and drop creation; see [Create a Node](#) (page 12) for more information) are shown. You can also select this option from the **Node Editor Options** menu; or, you can also enable/disable this option by clicking the

lock  button to lock/unlock the view. See [Locking the view](#) (page 14) for more information.

Display Customize your graph in the **Node Editor** by selecting any of the provided options. You can also select these options via the **Node Editor Display** menu. For a description of the available options, see [Node Editor Display menu](#) (page 47) for more information.

Show Select which object types to show in the **Node Editor**. You can also select these options via the **Node Editor Show** menu. For a description of the available options, see [Node Editor Show menu](#) (page 48) for more information.

Hypershade Select this option to enable the **Hypershade**.

Hypergraph Select this option to enable the **Hypergraph**.

You can also select the **Display** menu options, and the **Sync Node Editor/Model Selection** and **Add to Graph on Create** options by selecting **Windows > Settings/Preferences > Preferences > Settings > Node Editor**. If you select these options under the **Preferences** window, the changes are applied the next time you open the **Node Editor**. See [Node Editor Preferences](#) (page 55) for more information.

Node marking menus

Access the node marking menus by right-clicking on any node in the **Node Editor**.

Inputs Use the menu items provided to select, enable, and disable all inputs.

Outputs Use the menu items provided to select, enable, and disable all outputs.

Rename Select this option to rename your node.

Toggle Swatch Size Select this option to toggle between a larger and a smaller version of the node swatch.

Show All Attributes Select this option to change the node to full mode and to display all of its attributes. The displaying of all of its attributes is only valid for this draw instance. If you regraph the node, or if you change the view mode of the node, you must re-select this option to see all of its attributes again.

Add Attributes Select this option to open the **Add Attribute** window. See Create, edit or delete custom attributes for more information.

Delete Attributes Select this option to open the **Delete Attribute** window to delete a custom attribute.

Related topics

- [Node Editor menu and toolbar](#) (page 44)
- Hypergraph
- Hypershade

Node Editor Preferences



You can customize the following options for the **Node Editor**. They are applied the next time you create a **Node Editor** window.

Display

Customize your graph in the **Node Editor** by selecting any of the provided options.

Shapes Choose among the options to customize which shapes are displayed.

Transforms Select this option to include transform nodes in your graph.

Extend to shapes Select this option to include child shape nodes for selected transforms.

Default node view mode

- **Simple Mode:** Set the view mode of nodes to simple mode so that only the input and output master ports are shown.
- **Connected Mode:** Set the view mode of nodes to connected mode so that the input and output master ports, as well as any connected attributes, are shown.
- **Full Mode:** Set the view mode of nodes to full mode so that the input and output master ports, as well as primary node attributes, are shown. The set of primary attributes for a given node is based on the properties of the attributes such as whether they are keyable or dynamic.

You can also customize these options from within the **Node Editor**.

You can select the **All Shapes**, **All Shapes Except Shading Group Members**, and **No Shapes** options from the **Node Editor** toolbar or from the **Display** menu or from the work area marking menus. See [Node Editor menu bar](#) (page 45), [Node Editor toolbar](#) (page 48), and [Node Editor marking menus](#) (page 53) for more information.

You can select the **Transforms** and **Extend to shapes** options from the **Display** menu or from the work area marking menus. See [Node Editor menu bar](#) (page 45) and [Node Editor marking menus](#) (page 53) for more information.

You can change the view mode (**Simple Mode**, **Connected Mode**, and **Full Mode**) for multiple nodes interactively in the **Node Editor** using one of these methods: from the **Edit** menu, via shortcuts, or from the toolbar. See [Changing view modes for multiple nodes](#) (page 41), [Node Editor menu bar](#) (page 45) and [Node Editor toolbar](#) (page 48) for more information.

Refresh Graph Defaults

Sync Node Editor/Model Selection Enable this option so that if you highlight a node in the **Node Editor**, the same node is also selected in the scene, and vice versa. See [Selection in the Node Editor](#) (page 18) for more information.

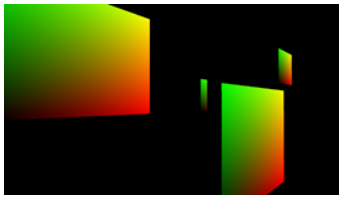
Add to Graph on Create Enable this option so that when you create a node in the scene, it is also added to the **Node Editor**. If this option is disabled, only nodes that you create using the **Node Editor** (for example, using Tab creation or drag and drop creation; see [Create a Node](#) (page 12) for more information) are shown.

NOTE You can also control these options from the **Options** menu within the **Node Editor** or from the **Node Editor** work area marking menus. See [Node Editor marking menus](#) (page 53) for more information.

What's New in Rendering and Render Setup

3

New render passes added



Two new multi-render passes have been added: UV pass and world position pass.

A UV pass converts UV values to R/G values and creates a rasterized version of UV space. Using a UV pass, you can replace textures in 3d renderings as a post-process, without having to worry about tracking new textures in place.

A world position pass converts position (x,y,z) values to R,G,B values. Use the world position pass for relighting workflows in compositing.

NOTE You must save to the openEXR file format when rendering the UV pass and world position pass.

mental ray Satellite rendering support

mental ray satellite rendering can take place on four remote machines, with up to four processors each and an unlimited number of cores.

UV multi-render pass workflow



A UV pass converts UV values to R/G values and creates a rasterized version of UV space. Using a UV pass, you can replace textures in 3d renderings as a post-process, without having to worry about tracking new textures in place.

To create a UV pass for your workflow, do as follows:


- 1 Create a pass contribution map for the object(s) for which you want to retexture.

Select your object, and in your **Render Layer Editor**, select **Contribution > Create and Associate Pass Contribution Map**.

- 2 Open the **Render Settings** window; select **mental ray** as your renderer, and click the **Passes** tab. Click the Create new render pass



button and select **UVPass** from the **Create Render Passes** window and click **Create and Close** to create the UV pass.

- 3 Click the Associate Selected Passes button  to associate the UV pass with your scene, and then with your pass contribution map.
- 4 In the UV pass **Attribute Editor**, select **Hold-Out**. This option blacks out all other objects in your scene other than the object you are retexturing.

Ensure that **Filtering** is disabled.

NOTE In general, it is not advisable to perform filtering on a UV pass.

- 5 Select **Embed object ID in Blue channel**. Selecting this option embeds values from the *miLabel* attribute on shape nodes directly to float values in the empty blue channel in the UV pass.
This option allows you to separate the object(s) in your compositing software in order to make further changes.
- 6 Create and add the *miLabel* attribute to the object's shape node. Click **Select** in the object's shape node **Attribute Editor**. Then select **Attributes > Add Attribute**.


- 7 Enter miLabel as the **Long name** and select **Integer** as your **Data Type**.

NOTE You must use the name miLabel spelled exactly as stated.


You can now see the MI Label attribute under the **Extra Attributes** section of the shape node.

- 8 Enter your MI Label value. To group objects, give all objects an identical MI Label.



- 9 Under the **Passes** tab, click  again and select **Coverage** from the **Create Render Passes** window and click **Create and Close** to create a coverage pass. The coverage pass provides the alpha for the object you want to retexture.



- 10 Click  to associate the coverage pass with your render layer and then with your pass contribution map.

- 11 Render your pass using the `.exr` format. In the **Common** tab, select **OpenEXR (exr)** under **File Output > Image format**.

NOTE Currently, only the OpenEXR file format is supported.

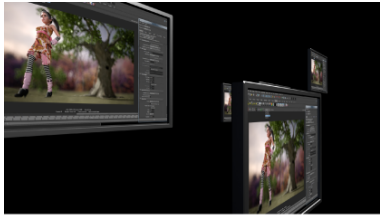
NOTE Under the **Quality** tab, **Framebuffer** section, you must set your **Data type** to 32-bit float or 16-bit half float to avoid your image being clamped.

In your compositing software, you can then do as follows:

- use your UV pass to displace a new image into your scene's texture co-ordinates to produce a re-textured image that can be multiplied by your coverage pass and merged back into your original render
- use expressions or keying to extract the value of your blue channel and then use it to apply changes to your objects

NOTE

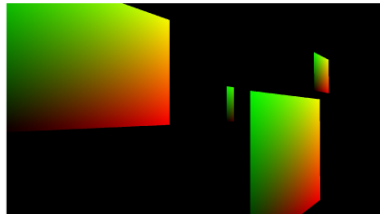
You may want to render textures and lighting as separate passes to facilitate re-lighting in compositing. You can do so by multiplying the Diffuse Material Color pass by the Direct Irradiance pass after using the UV pass to retexture.



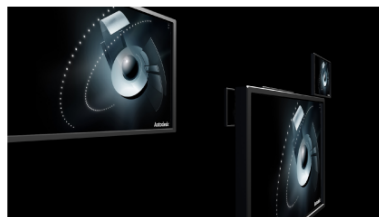
original 3d render



replacement texture



UV pass of original render



texture replaced in 3d render

What's New in Dynamics

4

MayaBullet physics simulation plug-in

Maya now includes the MayaBullet physics simulation plug-in. Built from the Bullet physics library, the plug-in lets you use the Bullet physics engine to create large-scale, highly-realistic dynamic and kinematic simulations. MayaBullet simulations can include interacting soft body and rigid body objects, as well as constrained collision objects, all contained in a single dynamic system within Maya.

MayaBullet automatically installs with Maya on Windows 64-bit, Mac OS, and Linux systems. MayaBullet is not available for Windows 32-bit systems.

For documentation on the MayaBullet plug-in, browse to the `doc` folder located in the `Bullet2012` installation directory. For example:

```
C:\Program Files\Autodesk\Bullet2012\doc.
```

For more information about Bullet Physics, see <http://www.bulletphysics.org>.

■ [Load the MayaBullet plug-in](#) (page 64)

Load the MayaBullet plug-in



The MayaBullet plug-in automatically installs with Maya on Windows 64-bit, Mac OS, and Linux platforms. MayaBullet is not available for Windows 32-bit systems. Before using the MayaBullet for the first time, you need to load the plug-in using the **Plug-in Manager**. To do this:

- 1 Start Maya.
- 2 Select **Window > Settings/Preferences > Plug-in Manager**.
The **Plug-in Manager** opens.
- 3 Locate `bullet.mll` and turn on **Loaded**. To load the plug-in automatically every time you start Maya, turn on **Auto load**.
- 4 Click **Close**.

In Maya, a **Bullet** menu is added to the Maya main menu bar and a **Bullet** item is added to the shelf. Use these items to access MayaBullet.

