Autodesk® 3ds Max® 2009

Help: Volume 1

Autodesk®

Trademarks

Disclaimer

1 Introduction

What's New in Autodesk 3ds Max 2009

Autodesk 3ds Max 2009 software introduces time-saving new animation and mapping workflow tools, groundbreaking new rendering technologies, and improved 3ds Max interoperability and compatibility with such industry-standard products as Autodesk® Revit™, Autodesk® Mudbox™, Autodesk® Maya® and Autodesk® MotionBuilder™ software.

New rendering technologies include the Reveal™ rendering toolset for both iterative workflows and faster, finished renderings, and the ProMaterials™ material library for simulating real-world surfaces. The release also delivers numerous biped enhancements, new UV editing tools, along with improved OBJ and FBX® import and export, which enhance interoperability with Autodesk Mudbox, Maya, MotionBuilder, and other third-party applications.

NOTE This topic doesn't comprehensively list all the changes in 3ds Max. As you proceed through the documentation, keep an eye out for the icon, which designates a new feature. You can also use the index in this reference to identify topics that contain information about new features in the program. For topics that describe new program features, check the index entry "new feature". For changes in existing features, check the index entry "changed feature". Also, a few features have changed names, such as “Render Scene dialog” to “Render Setup dialog.” To find these, check the index entry “changed name.”

Following is a list of major new features with brief descriptions and links to the relevant reference topic:

Help Movies

The help now contains links to a number of videos that introduce features and show how to accomplish certain tasks. To find the videos, search for "Watch a movie".

NOTE When you click a link, the movie opens in a Web browser. You will need to click once to activate the window and then, to watch the movie, click the Play button.

General Improvements

- **InfoCenter Capability** In Autodesk 3ds Max 2009, you can access help via Autodesk InfoCenter. InfoCenter lets you search for information through key words (or by entering a phrase), display the Communication Center panel for product updates and announcements, or display the Favorites panel to access saved topics. In 3ds Max, InfoCenter appears on a toolbar. See Find the Information You Need on page 13 for how to use InfoCenter; see also InfoCenter Toolbar on page 7503
- **ViewCube™ and SteeringWheels™** The new ViewCube™ on page 107 makes it easy to use the mouse to orbit viewports and jump quickly to a standard view orientation such as Left or Top. And the SteeringWheels™ on page 114 give you handy mouse-tip controls for zooming, panning, orbiting, and rewinding through a series of view changes. Both of these features will become standard across Autodesk's 3D products, providing users with a consistent navigation experience even as they move among applications.

Revit Interoperability and FBX Support

Autodesk 3ds Max 2009 provides enhanced support for Autodesk Revit® Architecture users.

- In imported FBX files you can browse the Revit user-defined tags category, family, level, and type in the Scene Explorer; see Scene Explorer Columns on page 7385.
- ProMaterials on page 5647 directly correspond to Revit materials, which can simplify exchanging information between the products.
- Photometric lights on page 5005 now correspond more closely to their Revit counterparts, making it easier to share scenes.

- Additional improvements are described in the *3ds Max FBX Plug-in Guide* that you can access from the FBX Importer/Exporter dialog.

Photometric Lights

There are now only two types of photometric lights on page 5005 (aside from the sun and sky lighting options): Target and Free. The user interface for photometric lights has been improved, and you choose the distribution and shape properties of a photometric light on the General Parameters rollout on page 5019 and the Shape/Area Shadows rollout on page 5040, respectively.

Other photometric light improvements include:

- Three new shadow-casting shapes: disc, sphere, and cylinder.
- Far Attenuation controls that let you limit the range of a photometric light. Using this option can dramatically improve rendering time, especially in scenes that contain a large number of light objects.
- A new option, Incandescent Lamp Color Shift When Dimming, that simulates the way an incandescent light becomes more yellow in color as it is dimmed.
- Display of web diagrams when you browse for or choose a photometric web file.
- New sky models for the mr Sky light: Perez All-Weather on page 5179 and CIE on page 5180.

The new interface for photometric lights is closer to the photometric light interface in Revit, which is a convenience if you use Revit as well as 3ds Max.

Materials and Mapping

- **mental ray ProMaterials** ProMaterials™ on page 5647 are mental ray materials that provide a convenient way of creating realistic textures. These materials correspond to Autodesk Revit materials, so users of both Revit and 3ds Max can share surface and material information between the two applications. Also included are presets based on manufacturer-supplied data and professional images.
- **mental ray Utility Materials** The new Utility Bump Combiner on page 5695 and Utility Displace Combiner on page 5697 materials let you combine a base material with multiple bump or displacement maps.

- **Spline Mapping** The Unwrap UVW modifier now lets you define custom mapping with a spline on page 1915. This is useful for quickly mapping such objects as snakes, tentacles, and winding roads.
- **Pelt Map Redesign** The Pelt Map command interface on page 1897 in Unwrap UVW has been redesigned for easier workflow and enhanced functionality.
- **Composite Map** The Composite map on page 5918 is completely revamped. Additional functionality includes the ability to apply masks and use color correction on both maps and masks, and to use blend modes for different methods of combining the layers.
- **Color Correction Map** The Color Correction map on page 5933 provides tools for modifying the colors of an incorporated, underlying map, using a stack-based method. Tools include monochrome, inversion, custom rewiring of color channels, hue shift, and adjustment of saturation and lightness. Color-adjustment controls in many cases mirror those found in Autodesk Toxik and Autodesk Combustion.
- **Enhanced Accessibility of Production Shaders** The mental ray production shaders on page 6017 are now more accessible. For example: A new lens shader enables mental ray to evaluate only those rays that intersect specified objects; a matte/shadow material is now available that can capture indirect illumination; and the new chrome ball shader lets the artist incorporate accurate scene environments (including lighting) from HDR photos.

Rendering

- **Rendered Frame Window Enhancements** As the command center of the Reveal workflow, the Rendered Frame Window on page 6073 offers a significantly expanded feature set for a streamlined rendering workflow, including the ability to render images, set rendering regions, change rendering parameters, and a new Iterative rendering mode on page 6101 for quickly testing scene changes.
- **mr Proxy Object** The new mr Proxy object on page 597 offers faster throughput and huge render-time memory savings with large scenes and high-resolution geometry. It lets you convert any object to a disk-based mental ray-format file that supports vertex-level animation as well as topology changes between frames.
- **mr A&D Render Elements** New mr A&D render elements on page 6358 support output of Arch & Design material components in HDR format to compositing applications such as Autodesk Toxik™.

- **mr Labeled Render Element** The mr Labeled render element on page 6363 enables output of one or more branches of a material tree to a custom render element.
- **mr Shader Render Element** The mr Shader render elements on page 6365 enables output of the raw contribution of any mental ray shader in the scene.
- **Geometry Caching** Geometry caching on page 6321 in mental ray stores the translated scene in a temporary file for reuse in subsequent renders. This saves time by omitting the translation step, especially with geometry-heavy scenes. Two levels of caching are available.
- **Pre-render Photon Maps and Final Gather Maps** To reduce flicker when rendering animations on a network, you can pre-generate photon maps on page 6312 and final gather solutions on page 6301.
- **New mental ray Object Properties** New mental ray object properties on page 322 allow more-flexible interaction between scene contents and indirect lighting.
- **New BSP2 Raytrace Acceleration** The implementation of a new, faster BSP on page 6280 (binary space partitioning) acceleration in mental ray 3.6 improves large-scene rendering performance and object instantiation. Unlike the traditional BSP acceleration, which is still available, the new technique automatically tunes for BSP performance and memory-consumption improvements.

Scene and Project Management

- **Enhanced Scene Explorer** Scene Explorer adds new advanced filtering on page 7397 functionality that lets the dialog list only items that meet specific criteria based on object name, type, and so on. Plus there are now more options for how groups are displayed.
- **Autodesk Mudbox Interoperability** Improved support for the OBJ file format on page 7280, including more export options, facilitates importing and exporting of model data between Mudbox and 3ds Max software products as well as other third-party 3D digital sculpting applications. Users can now take advantage of new export presets, additional geometry options, including hidden splines/lines and new optimize options, to reduce file sizes and improve performance. There is also improved texture map handling and more import information with regards to face counts per object.

- **FBX Import/Export** Improved FBX on page 7222 memory management, data translation fidelity and new import options support interoperability between 3ds Max and other Autodesk products such as Maya and MotionBuilder.
- **OpenFlight Import/Export** Artists in the visual-simulation arena can take advantage of new import and export support for the OpenFlight® (FLT) on page 7223 file format.

General Animation Improvements

- **Walkthrough Assistant** The Walkthrough Assistant on page 5239 lets you easily create a predefined walkthrough animation of your scene by placing a camera on a path and setting the height, turning the camera and viewing a preview.

Hair Improvements

Enhancements to Hair on page 1119 functionality include:

- The buffer render is multithreaded.
- Hair antialiasing is improved (needs less oversampling).
- There are no longer memory consequences for lots of hair (buffer only, not tracing).
- Tiles display as they render.
- Tile memory limit is settable (in the Hair and Fur render effect).
- Transparency depth is settable (in the render effect).
- Skylights are supported (toggle available in the render effect and controlled with a blend slider).
- Hair display is multithreaded (faster interaction on multi-core machines).
- Fast optimization on very thin hairs.

Character-Animation Improvements

New features improve the ease of animating and skinning a Biped.

- **Alternative Rotation Center** You can rotate a biped about a pivot point that is not its center of mass, which is useful if you want the biped to fall over, swing from a tree, and so on. See Track Selection Rollout on page 4348.
- **Mirror In Place** The new Mirror In Place option lets you mirror biped animation without changing the biped's orientation. See Keyframing Tools Rollout on page 4380.
- **Triangle Neck** The new Triangle Neck option, similar to Triangle Pelvis, attaches the clavicles to the top of the spine instead of to the neck. For some character meshes, this can improve mesh deformation when you use Physique to apply the mesh.
- **ForeFeet** The new ForeFeet ("Four Feet") option lets you treat hands as feet. When you choose ForeFeet, you can set planted keys for fingers, as if the fingers were toes.
  You apply both the Triangle Neck and the ForeFeet options in Figure Mode. These options are on the Structure rollout on page 4424.

Modeling Improvements

- Adjusting a soft selection is now more interactive with the new Edit Soft Selection Mode on page 2013 custom user interface action for changing the Falloff, Pinch, and Bubble values without leaving the viewport.

3ds Max Documentation Set

The documentation set for 3ds Max® comprises online material only. Most documents are available from the Start menu > Programs > Autodesk > [program folder] > Help folder, as well as from the Help menu within 3ds Max or the Additional Help dialog.

- **Autodesk 3ds Max 2009 and Autodesk 3ds Max Design 2009 Installation Guide:** Contains complete installation, configuration, and troubleshooting instructions, including system requirements. Also includes information on uninstalling and maintaining the software, as well as descriptions of the full install documentation set. View the *Installation Guide* in the Installation Wizard.

NOTE The complete installation documentation set is found on your install DVD in the *\Docs* folder at the root of the DVD. If you are deploying 3ds Max over a network, refer to the *Autodesk 3ds Max 2009 and Autodesk 3ds Max Design 2009 Network Administrator's Guide* instead of the *Installation Guide*.

- **Autodesk 3ds Max 2009 Readme (3ds_Max_Readme.rtf):** Contains the latest information about 3ds Max. You can access this file by clicking View Readme in the Autodesk 3ds Max 2009 Installation Wizard.
- **Autodesk 3ds Max 2009 Help:** This document covers fundamental concepts and strategies for using the product, as well as details about the features of 3ds Max.
  Access the reference by choosing Help > Autodesk 3ds Max Help.
- **Autodesk 3ds Max 2009 Tutorials:** Contains tutorial information and detailed procedures to walk you through increasingly complex operations. This is the best source for learning 3ds Max.
  Access the tutorials by choosing Help > Tutorials.

  NOTE All the sample files required to do the tutorials are found on DVD 1 of 2: Software in a folder called *\tutorials* at the root of the DVD. These files are not installed automatically.

- **MAXScript Reference:** Describes the MAXScript scripting language on page 11. Check out the "Learning MAXScript" chapter there if you're new to MAXScript.
  Access the *MAXScript Reference* by choosing Help > MAXScript Help.
- **Additional mental ray® Help Files:** Documentation from mental images® is available from Help menu > Additional Help. There, you'll find the *mental ray Reference*, comprising the *mental ray Manual*, *mental ray Shader Reference*, and *LumeTools Collection*. You'll also find PDF files documentating various mental ray shader libraries.

  NOTE The *Autodesk 3ds Max 2009 Help* documents most mental ray components available in the 3ds Max user interface. This includes documentation for lights for mental ray and specific shadow types, controls for adding mental ray shaders to lights and cameras, mental ray materials, custom shaders for 3ds Max, and the mental ray renderer controls.

- **Additional Backburner Files:** Procedures for using Backburner from inside 3ds Max are documented in this *Autodesk 3ds Max Help File*. For further information on using and configuring Backburner, refer to the additional

Backburner documentation available from Additional Help as well as from the Start menu location described in the introduction of this topic. The two documents related to Autodesk Backburner are called:

- *Backburner Installation Guide*
- *Backburner User's Guide*

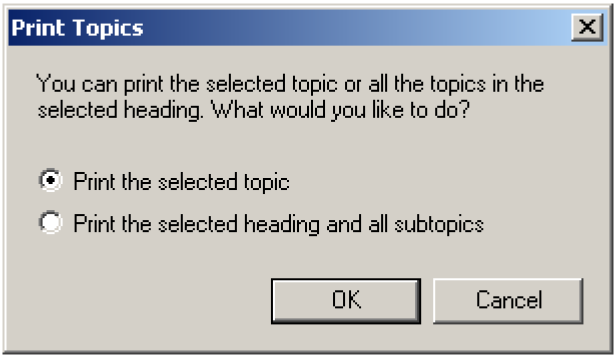
NOTE For information on installing Autodesk Backburner, see the *Autodesk 3ds Max 2009 and Autodesk 3ds Max Design 2009 Installation Guide*.

- **FBX Plug-in Help:** Access the FBX Plug-in Help from the Additional Help menu. You can also click the ? (Help) button on the FBX Importer/Exporter dialog.
  The FBX plug-in changes often, with the result that Autodesk updates it more frequently than it does this program. Be sure to check regularly for updated versions by clicking the Web Updates button on the dialog.
- **Flight Studio 2009 Reference:** Find reference and tutorial information for the Flight Studio 2009 Plug-in in the *Flight Studio 2009 Reference* available in the Additional Help menu. The Flight Studio 2009 Plug-in is selected for install by default when you install Autodesk 3ds Max or Autodesk 3ds Max Design. You can also review the Flight Studio Readme (*Flight Studio 2.0.0.1 documentation.doc*) available in the *\\Program Files\Autodesk\3ds Max 2009\help directory*.

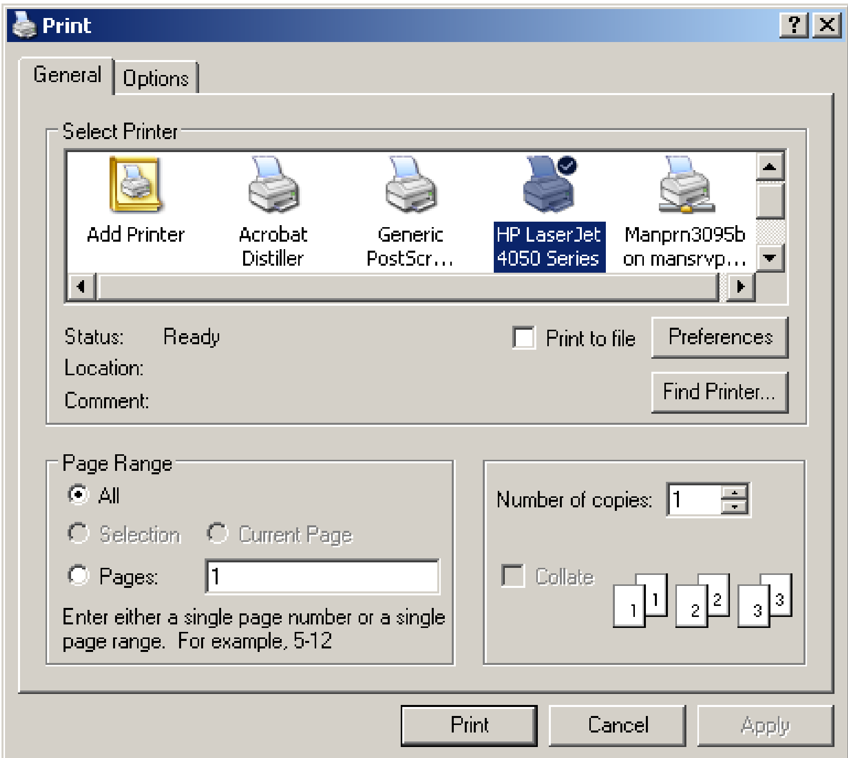
How to Print from the Online Help

If your computer is connected to a printer, you can print single help topics or entire chapters.

To print a topic or chapter, highlight the topic or chapter title and click the Print button at the top of the help display. A dialog opens.



Choose to print only the selected topic, or to print all topics in that chapter. After you make your selection, another dialog appears where you can choose your printer and other options.



The tabs available at the top of the dialog depend on the selected printer. Choose options for the print job, and click OK to begin printing.

How to Contact Us

We are also interested in hearing your views about 3ds Max. We'd like to hear ways you think we can improve our program, features you're interested in, as well as your views on the documentation set.

Please send us email about the documentation set at: *me.documentation@autodesk.com*

About MAXScript

MAXScript is the built-in scripting language for 3ds Max. It provides users with the ability to:

- Script all aspects of 3ds Max use, such as modeling, animation, materials, rendering, and so on.
- Control 3ds Max interactively through a command-line shell window.
- Package scripts within custom utility panel rollouts or modeless windows, giving them a standard 3ds Max user interface.
- Build custom import/export tools using the built-in file I/O.
- Write procedural controllers that can access the entire state of the scene. Build batch-processing tools, such as batch-rendering scripts.
- Set up live interfaces to external system using OLE Automation.

The MAXScript language is specifically designed to complement 3ds Max. It is object-oriented, and has several special features and constructs that mirror high-level concepts in the 3ds Max user interface. These include coordinate-system contexts, an animation mode with automatic keyframing, and access to scene objects using hierarchical path names that match the 3ds Max object hierarchy.

The syntax is simple enough for non-programmers to use, with minimal punctuation and formatting rules.

Visual MAXScript

Visual MAXScript is a powerful addition to MAXScript, making the MAXScript feature easier to learn and use. With Visual MAXScript, you can quickly create UI elements and layouts for scripting.

For detailed information about Visual MAXScript, open the MAXScript Reference, available from Help menu > MAXScript Reference.

See also:

- MAXScript Interface on page 7496

Procedures

To access MAXScript, do one of the following:

1. On the menu bar, choose MAXScript. The MAXScript menu appears.
2. Choose Utilities panel > MAXScript.

   From here, you can either write new scripts, edit or run existing scripts, open the MAXScript Listener, or use the Macro Recorder.

   To access the MAXScript Listener, you can also right-click in the Mini Listener and choose Open Listener Window from the right-click menu.

   For detailed information about the MAXScript utility, open the MAXScript Reference, available from Help menu > MAXScript Reference.

Find the Information You Need

2

There are various ways to find information about how to use this program, and multiple resources are available.

This program is a powerful application with tools that help you work with a high level of efficiency and productivity. You install this software with the Installation wizard that starts automatically when you insert the product C D.

This application is often intuitive, but when you do need to look something up, you can save time and avoid frustration if you use the Help system to find information. The Help system is organized in a structured design that makes information easy to locate.

Search For and Receive Information

On the InfoCenter toolbar, you can use InfoCenter to search a variety of information sources with one query. You can also easily access product updates and announcements.

Overview of Searching for and Receiving Information

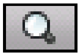
On the InfoCenter toolbar, you can use InfoCenter to search for information through key words (or by entering a phrase), display the Communication Center panel for product updates and announcements, or display the Favorites panel to access saved topics.

Left: Search field

Next: Search button

Next: Communication Center button

Right: Favorites button

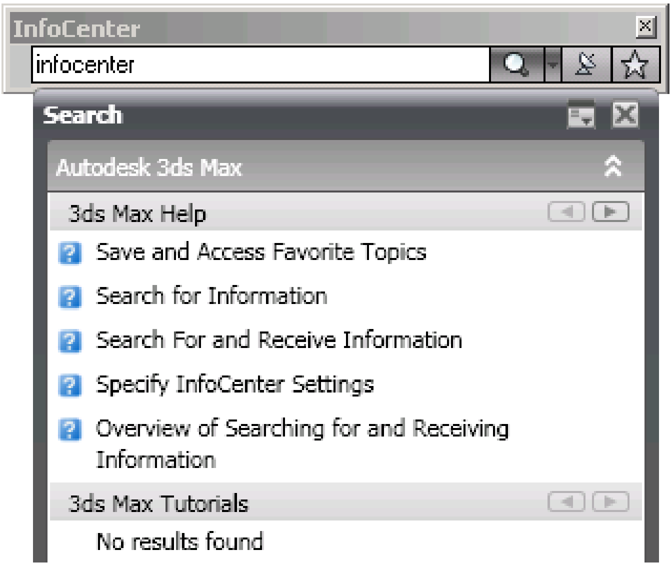
When you enter key words or a phrase and then press ENTER or click the InfoCenter button , you search multiple Help resources in addition to any files that have been specified in the InfoCenter Settings dialog box. The results are displayed as links on a panel. You can click any of these links to display the Help topic, article, or document.

NOTE It is recommended that you use key words to search for information, as key words often produce better results. In case of a misspelled word, spelling suggestions are displayed.

Get Information through InfoCenter

When you click the Communication Center button , the Communication Center panel is displayed. It displays links to information about product updates and product announcements, and may include links to Subscription Center, CAD Manager specified files, and RSS feeds.

When you click the Favorites button, a panel displays saved links to topics or web locations.

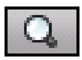


 Use the Next and Previous buttons to browse search results. Click and drag categories or groups to rearrange them.

See also:

- Using the 3ds Max Help on page 33
- What's New in Autodesk 3ds Max 2009 on page 1

To rearrange the topics displayed on a panel

1 Display a panel by doing one of the following:
   - In the InfoCenter toolbar, enter a keyword or phrase. Then press ENTER or click the InfoCenter button .
   - In the InfoCenter toolbar, click the Communication Center button .
   - In the InfoCenter toolbar, click the Favorites button .

2 Click and drag a category or group header to the desired position.

3 Release the mouse button.

NOTE Categories can be rearranged only within a group and cannot be moved into other groups.

To browse search results displayed on a panel

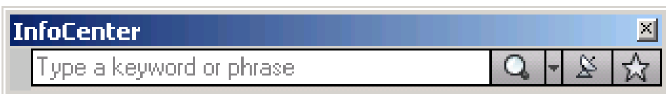
1 On the InfoCenter Search Results, Communication Center, or Favorites panel, on the right side of the category header, do one of the following:

- Click the Next button  .
- Click the Previous button  .

Search for Information

You can use InfoCenter to search multiple sources (for example, Help, the New Features Workshop, web locations, and specified files) at one time, or search a single file or location.

When you enter key words or a phrase in the InfoCenter box, and then press ENTER or click the InfoCenter button, you search the contents of multiple Help resources as well as any additional documents and web locations that have been specified in the InfoCenter Settings dialog box or through the CAD Manager Control Utility. The results are displayed as links on the InfoCenter Search Results panel. You can click any of these links to display the topic, article, or document.



NOTE It is recommended that you use key words to search for information, as key words often produce better results. In case of a misspelled word, spelling suggestions are displayed.

Get Information through InfoCenter

Click the Play arrow to start the animation.

You can also search a single location by clicking the arrow next to the InfoCenter button, and selecting a location or file from the list.

In addition, you can add a location (file or document) to search by clicking the arrow next to the InfoCenter button, and selecting Add Search Location from the list.

When you use InfoCenter to search for information, you can use the following special symbols (wild cards) in your query to refine or expand it. These symbols can be used alone or can be combined.

Symbol	Description
*	Replaces one or more characters when used at the beginning, middle, or end of a word. For example, "*lish", "p*lish", and "pub*" will all find "publish". Also, "anno*" will find "annotative", "annotation", "annoupdate", "annoreset", and so on.
?	Replaces a single character. For example, "cop?" will find "copy", but not "copybase".
~	Expands the tense of the word at the beginning or end of a word. For example, "plotting~" will find "plots", "plotted", and so on. Also, "~plot" will find "preplot", "replot", and so on.

When searching for multiple words in topics, use double quotation marks (" ") to enclose words that must appear next to each other in the specified sequence. For example, enter "**specifying units of measurement**" to find only topics with all those words in that order. You can also use the previously mentioned symbols (wild cards) in a text string that is enclosed in double quotation marks.

See also:

- Specify InfoCenter Settings on page 22
- Using the 3ds Max Help on page 33

To search multiple sources for information

1 In the InfoCenter box, enter a keyword or phrase.

2 Press ENTER, click the InfoCenter button, or click the arrow next to the InfoCenter button and select All Search Locations.

The search results are displayed on the Search Results panel.

To search a single location for information

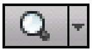
1 In the InfoCenter box, enter a keyword or phrase.

2 Click the arrow next to the InfoCenter button  .

3 Select a location from the list to search.

4 The search results from that location are displayed on the Search Results panel.

To add a location to search

1 In the InfoCenter box, click the arrow next to the InfoCenter button  .

2 Click Add Search Location.

3 In the Add Search Location dialog box, specify a document or a file location to search.

4 Click Add.

Receive Product Updates and Announcements

Communication Center provides up-to-date product information, software updates, product support announcements, and other product-related announcements.

Overview of Communication Center

To open Communication Center, click the Communication Center button  on the InfoCenter toolbar.

Access Communication Center

Communication Center provides the following kinds of announcements:

- **Product Support Information.** Get breaking news from the Product Support team at Autodesk, including when Live Update maintenance patches are released.
- **Subscription Announcements.** Receive subscription announcements and subscription program news, as well as links to e-Learning Lessons, if you are an Autodesk subscription member (available in countries/regions where Autodesk subscriptions are offered). For more information about Autodesk Subscription, see Access Subscription Center on page 28.
- **Articles and Tips.** Be notified when new articles and tips are available on Autodesk websites.
- **CAD Manager Channel.** Receive information (RSS feeds) published by your CAD manager.
- **RSS Feeds.** Receive information from RSS feeds to which you subscribe. (An RSS feed is information published by a website to which you subscribe. RSS feeds generally notify you when new content is posted.) Several default RSS feeds are automatically subscribed to when you install the program.
- **Live Update Maintenance Patches.** Receive automatic notifications whenever new maintenance patches are released from Autodesk.
- **Featured Technologies and Content.** Learn more about third-party developer applications and content.

You can customize the items that display on the Communication Center panel. For more information, see Specify InfoCenter Settings on page 22.

Communication Center Online Privacy

Communication Center is an interactive feature that must be connected to the Internet in order to deliver content and information. Each time Communication Center is connected, it sends your information to Autodesk so that you receive the correct information. All information is sent anonymously to Autodesk to maintain your privacy.

The following information is sent to Autodesk:

- Product name (in which you are using Communication Center)
- Product release number

- Product language
- Country/region (specified in the Communication Center settings)
- Your subscription contract number (if you're a subscription customer)

Autodesk compiles statistics using the information sent from Communication Center to monitor how it is being used and how it can be improved. Autodesk maintains information provided by or collected from you in accordance with the company's published privacy policy, which is available on *http://www.autodesk.com/privacy*.

See also:

- Specify InfoCenter Settings on page 22
- Access Subscription Center on page 28

Receive New Information Notifications

Whenever new information is available, Communication Center notifies you by displaying a balloon message below the Communication Center button on the InfoCenter box.

Click the link in the balloon message to open the article or announcement. If you prefer not to be notified by Communication Center, you can turn off Balloon Notification in the InfoCenter Settings dialog box. You can also customize the transparency of balloon messages as well as the length of time they are displayed.

See also:

- Specify InfoCenter Settings on page 22

Save and Access Favorite Topics

In InfoCenter, you can save links as favorites and easily access them later.

Any link that displays on the InfoCenter Search Results panel or Communication Center panel can be marked as a favorite. Links that are marked as favorites are displayed on the Favorites panel. You display the

Favorites panel by clicking the Favorites button on the InfoCenter box on the InfoCenter toolbar.

Save and Access Favorite Topics through InfoCenter

A link that is marked as a favorite displays a star icon when the link is displayed on the Search Results panel or the Communication Center panel. You can click the star icon to remove the link's "favorite" status, which removes it from the Favorites panel.

To display the InfoCenter Favorites panel

- In the InfoCenter box, click the Favorites button to display the Favorites panel.

NOTE The links displayed on the Favorites panel are organized into the same groups or categories from which they were added.

To save a link in InfoCenter as a favorite

1. Display a panel by doing one of the following:
   - In the InfoCenter box, enter a keyword or phrase. Then press ENTER or click the InfoCenter button.
   - In the InfoCenter box, click the Communication Center button.
2. Click the Star icon that appears next to the link that you want to save as a favorite.

To remove a favorite link from the InfoCenter Favorites panel

1. In the InfoCenter box, click the Favorites button to display the Favorites panel.
2. Click the Star icon that is displayed next to the link that you want to remove from the Favorites panel.

Specify InfoCenter Settings

You can specify InfoCenter Search and Communication Center settings in the InfoCenter Settings dialog box.

 In the InfoCenter Settings dialog box, you can specify the following settings:

- **General.** Your current locations, how often to check for new online content, animated transition effects for panels.
- **Search Locations.** Locations (documents, web locations and files) to search for information, as well as the name that displays for each location and the number of results to display for each. Also, you can add or remove search locations. The Web Locations checkbox provides access to important information on the Autodesk website, including the Knowledge Base and discussion groups. When you add document locations, you can specify files on your local drive or files on a network.

  NOTE User-specified CHM files must be located on your local drive. InfoCenter can search CHM files located on network drives.

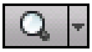
- **Communication Center.** Maximum age of the articles displayed on the Communication Center panel and the location and name of the CAD Manager Channel.
- **Autodesk Channels.** Channels to display in the Communication Center panel as well as the number of articles to display for each channel.
- **Balloon Notification.** Notifications for new product information, software updates, product support announcements, and Did You Know messages. Also, you can customize the transparency and the display time of the balloon.

  NOTE Did You Know balloons displayed below the Communication Center button on the InfoCenter box provide knowledge base information and general instructional messages such as tips. You can click on the text or the Expand icon  to expand the balloon to view the detailed information.

- **RSS Feeds.** RSS feed subscriptions. You can add or remove RSS feeds. (An RSS feed is information published by a website to which you subscribe.

RSS feeds generally notify you when new content is posted.) Several default RSS feeds are automatically subscribed to when you install the program.

To specify locations to search for information

1 In the InfoCenter box, click the arrow next to the InfoCenter button  .

2 Click Search Settings.

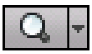
3 In the InfoCenter Settings dialog box, Search Locations panel, in the right pane, select or clear the search locations you want to include or exclude when you search for information.

NOTE You can click directly on the search location name to change the name in the right pane of the Settings dialog box.

4 Click OK.

NOTE The Search all available languages option allows you to specify whether to search default language or all available languages. This option is deselected by default.

To add a new location to search for information

1 In the InfoCenter box, click the arrow next to the InfoCenter button  .

2 Click Search Settings.

3 In the InfoCenter Settings dialog box, do one of the following:

- On the Search Locations panel, in the right pane, click Add.
- On the Search Locations panel, in the right pane, right-click anywhere in the pane. Click Add.

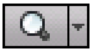
4 In the Add Search Location dialog box, specify a file location to search.

NOTE You can specify a file on your local drive or on a network.

5 Click Add.

6 Click OK.

To remove a location to search for information

1 In the InfoCenter box, click the arrow next to the InfoCenter button .

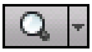
2 Click Search Settings.

3 In the InfoCenter Settings dialog box, do one of the following:
   - Select a location to remove, and then click Remove.
   - Right-click a search location. Click Remove.

4 In the Remove Search Location dialog box, click Yes to remove the selected location.

5 Click OK.

To specify the channels to display in the Communication Center panel

1 In the InfoCenter box, click the arrow next to the InfoCenter button .

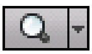
2 Click Search Settings.

3 In the InfoCenter Settings dialog box, in the left pane, click Autodesk Channels.

4 In the right pane, select or clear the channels you want to display in the Communication Center panel.

5 Click OK.

To specify InfoCenter balloon notification settings

1 In the InfoCenter box, click the arrow next to the InfoCenter button .

2 Click Search Settings.

3 In the InfoCenter Settings dialog box, in the left pane, click Balloon Notification.

4 In the right pane, select or clear the options to turn balloon notification on or off.

NOTE You can turn off the balloon notification for Did You Know messages in the InfoCenter Settings dialog box.

5 Enter the number of seconds to set the length of time for balloon notifications to display.

NOTE The default value for the balloon display time is 5 seconds.

6 Enter the transparency value of the balloon or set the value using the slider.

7 Click OK.

To turn on the display of Did You Know hidden messages

1 Click **Tools** menu ➤ **Options** .

2 In the Options dialog box, System tab, under General Options, click the Hidden Message Settings button.

3 In the Hidden Message Settings dialog box, select the InfoCenter Did You Know Balloons option to turn on the display of all hidden messages.

4 Click OK.

To add an RSS feed to Communication Center

1 In the InfoCenter box, click the arrow next to the InfoCenter button  .

2 Click Search Settings.

3 In the InfoCenter Settings dialog box, in the left pane, click RSS Feeds.

4 In the right pane, do one of the following:

■ Click Add.

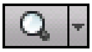
■ Right-click anywhere in the right pane. Click Add.

5 In the Add RSS Feed dialog box, enter the location of the RSS feed you want to add.

6 Click Add.

7 Click OK.

To remove an RSS feed from Communication Center

1 In the InfoCenter box, click the arrow next to the InfoCenter button .

2 Click Search Settings.

3 In the InfoCenter Settings dialog box, in the left pane, click RSS Feeds.

4 In the right pane, do one of the following:
   - Click Remove.
   - Right-click an RSS feed. Click Remove.

5 In the Remove RSS Subscription dialog box, click Yes to remove the selected RSS feed.

6 Click OK.

Learn the Product

Training programs and products from Autodesk help you learn the key technical features and improve your productivity. For the latest information about Autodesk training, visit *http://www.autodesk.com/training* or contact your local Autodesk office.

Autodesk Authorized Training Centers

The Autodesk® Authorized Training Center (ATC®) network delivers Autodesk-authorized, instructor-led training to design professionals who use Autodesk software. Autodesk Authorized Training Centers use experienced and knowledgeable instructors. More than 1,200 ATC sites are available worldwide to meet your needs for discipline-specific, locally based training.

To find a training center near you, contact your local Autodesk office or visit *http://www.autodesk.com/atc*.

Autodesk Official Training Courseware

Autodesk Official Training Courseware (AOTC) is technical training material developed by Autodesk. Designed for traditional 1/2-day to 5-day, instructor-led classroom training and used by Authorized Training Centers and other Autodesk partners, AOTC is well-suited for self-paced, stand-alone learning.

The manuals cover key concepts and software functionality with hands-on, step-by-step, real-world exercises. You can purchase AOTC from your local reseller or distributor, or you can order it online from the Autodesk Store at *http://www.autodesk.com/aotc*.

e-Learning

Autodesk e-Learning for Autodesk Subscription customers features interactive lessons organized into product catalogs. Each lesson is 20-40 minutes in length and features hands-on exercises, with an option to use a simulation of the product or the actual application. You can also use an online evaluation tool that identifies gaps in skills, determines what lessons will be most helpful, and gauges learning progress.

If you are a member of Autodesk subscription, you can access e-Learning and other subscription services from within your Autodesk product. For more information about how to access e-Learning in the product, see Access Subscription Center on page 28. For more information about Autodesk subscription resources, visit *http://www.autodesk.com/subscriptioncenter*.

Autodesk Developer Network

The Autodesk Developer (ADN) program for ADN members provides support for full-time, professional developers who want to build software based on Autodesk products. As an ADN member, you will receive the business, software, support, and training you need to be successful. If you are a developer, visit *http://www.autodesk.com/adn*.

Autodesk Consulting

Autodesk Consulting provides services that help set up processes and provide critical training that will help increase productivity so you can capitalize on the power of your products. For more information on general consulting, systems integration, or custom training services, visit *http://www.autodesk.com/consulting*.

Partner Products and Services

Autodesk works together with thousands of software partners around the world. These partners provide products and services that enhance Autodesk products for design professionals. Visit the Partner Products & Services page at *http://www.autodesk.com/partnerproducts* for a list of resources available for your Autodesk product and your industry.

Access Subscription Center

The Subscription Center is available to subscription members from within the product. If you are a subscription member, you can access subscription services by clicking the Communication Center button on the InfoCenter toolbar, and then clicking a Subscription Center link. To learn more about Autodesk subscription membership, visit *http://www.autodesk.com/subscriptioncenter*.

Overview of Subscription Center

With Autodesk Subscription, you get the latest releases of Autodesk software, incremental product enhancements, personalized web support from Autodesk technical experts, and self paced e-Learning. Subscription services are available to subscription members only.

By clicking the Communication Center button  on the InfoCenter toolbar, members have access to the following options (under Subscription Center):

- **Subscription status.** Checks your subscription status.
- **Create support request.** Provides direct one-to-one communication with Autodesk support technicians. You receive fast, complete answers to your installation, configuration, and troubleshooting questions.
- **View support requests.** Tracks and manage your questions and responses through Autodesk's state-of-the-art support system.
- **Edit Subscription Center profile.** Sets up and maintains your subscription account.
- **View e-Learning catalog.** Features interactive lessons organized into product catalogs.
- **e-Learning Lessons.** (For subscription members only.) Each lesson is 15–30 minutes and features hands-on exercises, with an option to use a simulation instead of the software application. You can use an online evaluation tool that identifies gaps in skills, determines what lessons will be most helpful, and gauges learning progress.

Subscription Resources and Privacy

Subscription resources provide interactive product features over the Internet. Each time you access subscription resources (such as e-Learning or Create Support Request) from Communication Center in an Autodesk product, product information (such as the serial number, version, language, and the subscription contract ID) is sent to Autodesk for verification that your product is on subscription.

Autodesk compiles statistics using the information sent to subscription resources to monitor how they are being used and how they can be improved. Autodesk maintains the information provided by or collected from you in accordance with Autodesk's published privacy policy, which is available at *http://www.autodesk.com/privacy*

To access the Subscription Center in the program

1. In the InfoCenter box, click the Communication Center button.
2. On the Communication Center panel, under Subscription Center, click the subscription resource you want to access.

NOTE Subscription Center is not available to all product users. If subscription resources are not available in your product, your product is not entitled to subscription benefits.

Manage Files with Autodesk Vault

If you are a subscription customer, you have access to Autodesk Vault, a file management tool that provides a repository where documents and files are stored and managed. Autodesk Vault is also shipped with certain other Autodesk softwares. Autodesk Vault gives you more power to manage files and track changes. Versioned copies of master files are maintained, allowing you to easily revert to earlier versions of files. You can check files out for editing and later check them back in. The master copy is never directly edited. During the check-in process, you can add comments regarding the edits you've made to inform other designers of your changes. You are able to quickly understand a project’s developmental flow and history.

Autodesk Vault consists of three required components: the Vault Plug-in, the Autodesk Data Management Server, and the Vault Client. The Vault Plug-in is shipped with 3ds Max. The Autodesk Data Management Server and the Vault Client are available on the Autodesk Subscription site.

For information about using the Vault, refer to the Vault Help system.

The Autodesk Data Management Server

The Autodesk Data Management Server stores the master copies of all your documents and designs. By storing all your data in a common, centralized location, you can easily share and manage information with your design team.

The Vault Client

The Vault Client software includes Autodesk Vault Explorer. The Vault Explorer is a stand-alone application that provides tools you use to access data stored on the Autodesk Data Management Server. From within , you can log in, log out, and open and attach files directly from the Vault. You can also access the Vault using the Vault Explorer. With the Vault Explorer you can

- Manage the Vault
- Manage Vault user accounts
- Set working folders
- Create folders in the Vault
- Add, check in, and check out files (including non-Inventor and non-DWG files; exception is AutoCAD Electrical files)
- Move files in the Vault
- Rename files in the Vault
- View the history of design changes
- Create file associations within the Vault by attaching two or more files
- Check out the latest version of a file
- Package files using Pack and Go

Join the Customer Involvement Program

You are invited to participate in helping guide the direction of Autodesk design software.

If you participate in the Customer Involvement Program, specific information about how you use is forwarded to Autodesk. This information includes what features you use the most, problems that you encounter, and other information helpful to the future direction of the product.

Here is a list of the information that is automatically sent to Autodesk:

- Name and version of the Autodesk product
- commands used, and amount of time spent in
- Error conditions encountered, fatal, and non-fatal
- File formats imported or exported with
- Operating system name and version
- System configuration information such as processor, amount of memory, and graphics card
- IP address, used to identify your country or region

What the Customer Involvement Program Cannot Do

The Customer Involvement Program is committed to protecting your privacy. It *cannot* do any of the following:

- Collect any drawing or design data
- Collect any identity information such as name, address, or phone number
- Send you email or contact you in any other way

For additional information, click the links in the Customer Involvement Program dialog box.

Why You Should Consider Participating

The Customer Involvement Program involves you directly in telling Autodesk

- The commands and features that Autodesk should focus on
- The commands and features that are hardly ever used
- The most common problem areas
- The hardware typically used with

NOTE You can start or stop your participation in this program at any time. Access to the controls is available from the Help menu. In network installations, your system administrator can choose whether to make the CIP program available or not.

To turn the CIP on or off

1 Click **Help** menu ➤ **Customer Involvement Program.**

2 In the Customer Involvement Program dialog box, click a level of participation, and then click OK.

3

Using the 3ds Max Help

The 3ds Max Help (this document) gives you information about every aspect of the software. Each topic contains an overview discussion, typically preceded by a path annotation showing how to access the feature in the program, and followed by a "Procedures" section with steps for using a command or feature, and an “Interface” section detailing controls and parameters for the user interface.

- **Path Annotation:** Gives one or more sequences of steps indicating how to access the feature in the user interface.
- **Topic Overview:** Tells you the name of the feature, command, user-interface control, or concept, and gives you a description.
- **Procedures:** Contains steps for tasks that illustrate the typical use of the feature.
- **Interface:** Describes the controls for this feature as they appear in the user interface, with a description of their behaviors and settings.

Important Notes

New features are indicated in this help by the "New" icon shown at the beginning of this paragraph. This makes it easy to see what's new in the software as you use the reference. You can also identify topics containing information on new features in the program using the Index. Double-click the "new feature" entry to display a list of topics describing new program features. Double-click the entry "changed feature" to see which existing features have changed.

Keypresses, both individual and combination, are indicated in this document using a special text style. For example:

- To open the Select From Scene dialog, press H.
- To undo the most recent action, press Ctrl+Z.

When you click a “book” icon on the Contents tab, the Topic pane on the right side immediately displays the introductory or first topic in that section. This differs from previous behavior, in which the first topic was a separate entry in the table of contents and needed to be accessed separately from the "book" container.

The screen shots in this document show the default user interface colors. Other color sets for the user interface are available. You can choose them, or create your own, using the Colors tab on page 7712 in the Customize User Interface dialog on page 7697.

See also:

- Finding Information Fast on page 34
- Using the HTML Help Viewer on page 36
- Searching for Help Topics on page 38
- Help Menu on page 7496

Finding Information Fast

Use the Navigation pane in the Help Window to get to information quickly. It contains tabs that let you use Contents, Index, or Search techniques to get to topics you need.

Contents Tab

The Contents tab displays the main sections of this online system as book icons. When you click a book, it expands to show the list of topics contained within it, like chapters in hardcopy books.

To go to a topic from the Contents tab:

1. Click the Contents tab to display the Table of Contents view.
2. Click the book icon representing the area for which you want information.

   The Topic pane on the right displays the introductory topic for that section. In most cases, the topic contains an overview of the section contents plus a list of relevant links. In some cases, the topic consists exclusively of links to other topics in the section.

   The page icons for the book expand below representing all the topics for the book's feature area.
3. Click a link in the introductory topic to go to that topic.

Alternatively, you can view a list of the section's topics and sub-sections by clicking the + icon to the left of the book icon in the table of contents or double-clicking the book icon or section title. This expands the section and shows its contents.

4 Continue navigating through the table of contents until you find the topic you want and then click its icon.

Index Panel

The index is an alphabetical listing of keywords found in the help. A single keyword might be linked to more than one topic. Scroll through the list, or to jump to an index entry that matches what you're looking for, type the first few letters of the subject in the editable field above the list.

To go to a topic from the Index panel:

1. Click the Index tab to display the Help index.
2. In the form at the top of the Index, type the subject you want, or scroll through the alphabetical list to find the term for which you need information.
3. Click the term, then click Display to see the topic for that term, or double-click the term to see its topic.

   The topic displays in the right pane and may show links to related topics.

Search Tab

The Search tab summons a full-text search engine that operates on a database of every word in the help system, created when the HTML Help system was compiled. You can use tools on the Search tab to find the help topics on page 38 containing any word or phrase.

Favorites Tab

Use tools on the Favorites tab to create and store a set of topics you use often; you can name them as you choose.

Using the HTML Help Viewer

This online information system is a compiled HTML help (CHM) file; you view it using Microsoft's HTML Help Viewer, powered by Internet Explorer. The HTML Help Viewer is a three-pane window:

- The Navigation pane on page 34 is on the left side of the window. It contains five navigational tabs, for Contents on page 34, Index on page 34, Search on page 38, and Favorites on page 43.
- The Topic pane is on the right side of the window. It displays the selected help topic, or the default help topic. It's the window you're reading from right now.
- The toolbar on page 44 is the third pane, located below the help window title bar.
  Here are some tips on how to find more information when using the HTML Help Viewer:
- To link to another topic or a list of other topics, click the colored, underlined words in the Topic pane.
- If you use a particular help topic often, you can add it to your favorites list on page 43.
- Right-click the Contents or Favorites tab or the Topic pane for shortcut menu commands.

Comments

Each topic in the online version of this document ends with a **Comments** link. When you click Comments, the Help Viewer opens an electronic form you can use to send us comments or requests about that topic. We'll use that information when we revise the documentation set for a future release.

TIP Frequently check the downloads section on the 3ds Max support site for updated releases of our online references. Access it from Help menu > 3ds Max on the Web > Online Support.

See also:

- Finding Information Fast on page 34
- Searching for Help Topics on page 38

- Most of information about using the HTML Help Viewer has been supplied directly by Microsoft. It has been made freely available for inclusion in HTML help projects such as this one. This information has been edited and reformatted to match that of the other online information systems shipping with the software.

Procedures

To find a help topic:

1. In the Navigation pane, click one of the following tabs:
    - To browse through a table of contents, click the Contents tab. The table of contents is an expandable list of important topics.
    - To see a list of index entries, click the Index tab, and then type a word or scroll through the list. Topics are often indexed under more than one entry.
    - To locate every occurrence of a word or phrase that may be contained in a help file, click the Search tab, and then type the word. For details on searching, see Searching for Help Topics on page 38.
2. Double-click the contents entry, index entry, or search results entry to display the corresponding topic.

To copy a help topic:

1. In the Topic pane, right-click the topic you want to copy, and then click Select All.
2. Right-click again, and then click Copy. This copies the topic to the Clipboard.
3. Open the document you want to copy the topic to.
4. Position your cursor where you want the information to appear.
5. On the Edit menu, click Paste.

To copy only part of a topic:

- Select the text you want to copy, right-click, and then click Copy.

To print the current help topic:

- Right-click a topic, and then click Print.
  If you print from the Contents tab (by right-clicking an entry, and then clicking Print) you will see options to print only the current topic, or the current topic and all subtopics.

To hide or show the Navigation pane:

- On the toolbar, click Hide or Show to close or display the Navigation pane, which contains the Contents, Index, Search, and Favorites tabs.
  If you close the Help Viewer with the Navigation pane hidden, it will appear that way when you open the Help Viewer again.

To see where the current topic fits in the information hierarchy (contents):

- Press Alt+C.
  The Contents pane displays, with the current topic highlighted.

Searching for Help Topics

A basic search consists of the word or phrase you want to find. You can use Boolean, wildcard, and nested expressions. You can also limit the search to previous results, match similar words, or search topic titles only to further define your search.

The basic rules for formulating queries are as follows:

- Searches are not case-sensitive, so you can type your search in uppercase or lowercase characters.
- You may search for any combination of letters (a through z) and numbers (0 through 9).
- Punctuation marks such as the period, colon, semicolon, comma, and hyphen are ignored during a search.
- Group the elements of your search using double quotes on page 39 or parentheses on page 41 to set apart each element. You cannot search for quotation marks.

NOTE If you are searching for a file name with an extension, you should group the entire string in double quotes, ("filename.ext"). Otherwise, the period will break the file name into two separate terms. The default operation between terms is AND, so you will create the logical equivalent to "filename AND ext."

Searching for Words or Phrases: Using Wildcards

You can search for words or phrases and use wildcard expressions. Wildcard expressions allow you to search for one or more characters using a question mark or asterisk. The table below describes the results of these different kinds of searches.

Search for	Example	Results
A single word	select	Topics that contain the word "select." (You will also find its grammatical variations, such as "selector" and "selection.")
A phrase	"new operator" or new operator	Topics that contain the literal phrase "new operator" and all its grammatical variations.Without the quotation marks, the query is equivalent to specifying "new AND operator," which will find topics containing both of the individual words, instead of the phrase.
Wildcard expressions	esc* or 80?86	Topics that contain the terms "ESC," "escape," "escalation," and so on. The asterisk cannot be the only character in the term.Topics that contain the terms "80186," "80286,"

Search for	Example	Results
		"80386," and so on. The question mark cannot be the only character in the term.

Turn on Match Similar Words to include minor grammatical variations for the phrase you search.

Defining Search Terms: Using Boolean Expressions

The AND, OR, NOT, and NEAR operators enable you to precisely define your search by creating a relationship between search terms. The following table shows how you can use each of these operators. If no operator is specified, AND is used. For example, the query "spacing border printing" is equivalent to "spacing AND border AND printing."

Search for	Example	Results
Both terms in the same topic.	dib AND palette	Topics containing both the words "dib" and "palette."
Either term in a topic.	raster OR vector	Topics containing either the word "raster" or the word "vector" or both.
The first term without the second term.	ole NOT dde	Topics containing the word "OLE," but not the word "DDE."
Both terms in the same topic, close together.	user NEAR kernel	Topics containing the word "user" within eight words of the word "kernel."

NOTE The \|, &, and ! characters don't work as Boolean operators (you must use OR, AND, and NOT).

Using Nested Expressions When Searching

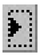
Nested expressions allow you to create complex searches for information. For example, "control AND ((active OR dde) NEAR window)" finds topics containing the word "control" along with the words "active" and "window" close together, or containing "control" along with the words "dde" and "window" close together.

The basic rules for searching help topics using nested expressions are as follows:

- You can use parentheses to nest expressions within a query. The expressions in parentheses are evaluated before the rest of the query.
- If a query does not contain a nested expression, it is evaluated from left to right. For example: "Control NOT active OR dde" finds topics containing the word "control" without the word "active," or topics containing the word "dde." On the other hand, "control NOT (active OR dde)" finds topics containing the word "control" without either of the words "active" or "dde."
- You cannot nest expressions more than five levels deep.

Procedures

To go to a topic from the Search tab:

1. Click the Search tab, and then type the word or phrase you want to find.
2.  To add Boolean operators to your search (not necessary if you're searching for a single term), click the Boolean button to the right of the text field, and then one of the operator names.
3. Click List Topics, choose the topic you want, and then click Display.
4. To sort the topic list alphabetically, click the Title column heading.

You can precisely define a search by using wildcard expressions, nested expressions, and Boolean operators.

You can request similar word matches, search only the topic titles, or search the results of a previous search.

You can set the Help Viewer to highlight all instances of search terms that are found in topic files. Click the Options button, and then click Search Highlight On.

To highlight words in searched topics:

When searching for words in help topics, you can have each occurrence of the word or phrase highlighted in the topics that are found.

- To highlight all instances of a search word or phrase, click Options on the toolbar, and then click Search Highlight On.
  To turn off this option, click Options on the toolbar, and then click Search Highlight Off. Another way to turn off highlighting without changing the Search Highlight ... setting is to go to the Contents tab, and then click the highlighted topic entry.

  If you are viewing a long topic, only the first 500 instances of a search word or phrase will be highlighted.

To search for words in the titles of HTML files:

1. Click the Search tab, type the word or phrase you want to find, and then turn on Search Titles Only.
2. Click List Topics, choose the topic you want, and then click Display.

   If you use this option, all HTML topic files will be searched, including any that are not listed in the table of contents.

To find words similar to your search term:

This feature enables you to include minor grammatical variations for the phrase you search. For example, a search on the word "add" will find "add," "adds," and "added."

1. Click the Search tab, type the word or phrase you want to find, and then turn on Match Similar Words.
2. Click List Topics, choose the topic you want, and then click Display.

   This feature only locates variations of the word with common suffixes. For example, a search on the word "add" will find "added," but it will not find "additive."

To search only the last group of topics you searched:

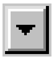
This feature enables you to narrow a search that results in too many topics found. You can search through your results list from previous search by using this option.

1. On the Search tab, turn on Search Previous Results.
2. Click List Topics, choose the topic you want, and then click Display.

If you want to search through all of the files in a help system, this check box must be off.

If you previously used this feature, the Search tab opens with this check box turned on.

To repeat an earlier search:

-  Click the down arrow on the text-entry field and choose a previously used search string, and then click List Topics.

Favorites Tab

Use tools on the Favorites tab to create a set of topics you use often; you can name them as you choose.

Procedures

To create a list of favorite help topics:

1. Locate the help topic you want to make a favorite topic.
2. Click the Favorites tab, and then click Add.

To return to a favorite topic:

1. Click the Favorites tab.
2. Choose the topic, and then click Display.

To rename a topic in the Favorites list:

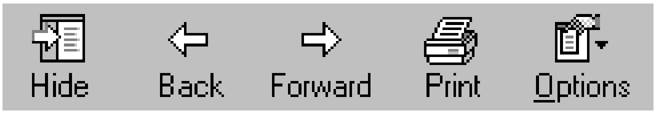
- Choose the topic, and then enter a new name in the Current topic box.

To remove a favorite topic:

- Choose the topic, and then click Remove.

HTML Help Viewer Toolbar

The Help Viewer toolbar contains the following features.



Hide/Show Click this toggle to hide the Navigation pane when it is displaying, or show it when it's hidden.

Back/Forward Click to move to the previously viewed topic, or forward to the following previously viewed topic.

Print Prints the current topic (if the Topic pane is active). If the table of contents is active on the Navigation pane, you can choose to print the current topic, or the topic and its subtopics. This is a way of printing a collection of topics.

Options Displays the options menu:

Hide/Show Tabs Same as Hide/Show buttons, described above.

Back/Forward Same as Back/Forward buttons, described above.

Home Displays the main topic of this online system.

Stop Halts display of a topic.

Refresh Redraws the Help Viewer display.

Internet Options Displays a dialog to change Internet Explorer (IE) settings. Changes you make here *do not affect* the online help or tutorials, but *do affect* your IE browser settings. We do not recommend you use this option.

Print Same as the Print button, described above.

Search Highlight On/Off Toggles highlighting of each instance of a word or phrase found with a search.

HTML Help Viewer Right-Click Menus

There are several commands on the shortcut menu that you can use to display information.

Command	Description
Right-click in the table of contents, and then click Open All.	Opens all books or folders in the table of contents. This command only works if the Contents tab is displayed.
Right-click in the table of contents, and then click Close All.	Closes all books or folders. This command only works if the Contents tab is displayed.
Right-click in the Topic pane, or an entry in the table of contents, and then click Print.	Prints the topic.
Right-click an entry in the Favorites tab.	Choose to display, add, remove, or rename a topic.

Keyboard Shortcuts in the Help Viewer

The following keyboard shortcuts can be used for navigation in the HTML Help Viewer, or the Contents on page 47, Index on page 47, Search on page 47, orFavorites on page 48 tabs on the Navigation pane.

Help Viewer

To	Press
Close the Help Viewer.	Alt+F4
Switch between the Help Viewer and other open windows.	Alt+Tab
Display the Options menu.	Alt+O
Hide or show the Navigation pane.	Alt+O, and then press T
Print a topic.	Alt+O, and then press P, or right-click in the Topic pane and choose Print.
Move back to the previous topic.	Alt+Left Arrow, or Alt+O, and then press B. Or, if the Topic pane is active, press Backspace.
Move forward to the next topic (provided you have viewed it just previously).	Alt+Right Arrow, or Alt+O, and then press F
Turn on or off search highlighting.	Alt+O, and then press O
Return to the home page (help authors can specify a home page for a help system).	Alt+O, and then press H
Switch between the Navigation and Topic panes.	F6
Scroll through a topic.	Up Arrow and Down Arrow, or Page Up and Page Down
Scroll through all the links in a topic.	Tab

Contents Tab

To	Press
Display the Contents tab. Tip: Use this shortcut to see where a topic fits in the information hierarchy.	Alt+C
Open and close a book or folder.	Click plus sign or minus sign next to book icon, or Left Arrow and Right Arrow
Choose a topic.	Down Arrow and Up Arrow
Display the selected topic.	Enter

Index Tab

To	Press
Display the Index tab.	Alt+N
Type a keyword to search for.	Alt+W, and then type the word
Choose a keyword in the list.	Up Arrow and Down Arrow
Display the associated topic.	Alt+D

Search Tab

To	Press
Display the Search tab.	Alt+S
Type a keyword to search for.	Alt+W, and then type the word
Start a search.	Alt+L

To	Press
Choose a topic in the results list.	Alt+T, and then Up Arrow and Down Arrow
Display the selected topic.	Alt+D
Search for a keyword in the result list of a prior search.	Alt+U and press Enter
Search for words similar to the keyword. For example, to find words like "running" and "runs" for the keyword "run."	Alt+M and press Enter
Search through topic titles only.	Alt+R and press Enter

Favorites Tab

To	Press
Display the Favorites tab.	Alt+I
Add the currently displayed topic to the Favorites list.	Alt+A
Choose a topic in the Favorites list.	Alt+P, and then Up Arrow and Down Arrow
Display the selected topic.	Alt+D
Remove the selected topic from the list.	Alt+R

Notes

- There are also shortcut menu commands on page 45 that can be accessed through the keyboard.
- The Match Similar Words check box, on the Search tab, will be turned on if you used it for your last search.

Getting Started with 3ds Max

4

You use 3ds Max to quickly create professional-quality 3D models, photorealistic still images, and film-quality animation on your PC.

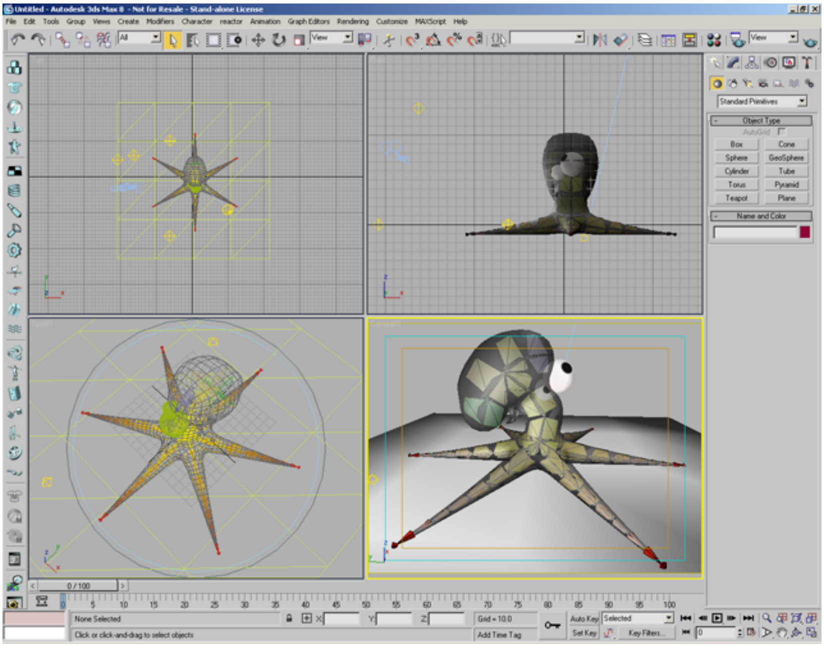
Image by Michael McCarthy

Before using this reference material, we highly recommend you get to know 3ds Max firsthand by following the included tutorials. You can access the tutorials using the Help menu > Tutorials command.

This section presents these brief topics designed to help you quickly start using 3ds Max.

Project Workflow

Once you've installed 3ds Max (see the *Installation Guide* included with your software package), you open it from the Start menu, or use any other Windows method. The figure below shows the application window with a scene file loaded.



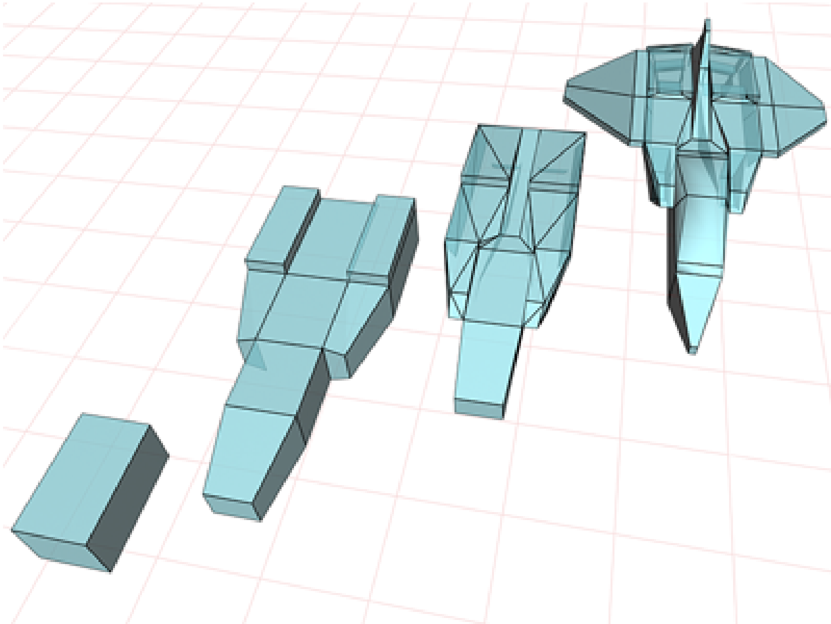
Main program window

NOTE If you open 3ds Max from a Command Prompt window or batch file, you can add command-line switches. See Starting 3ds Max from the Command Line on page 7468.

NOTE 3ds Max is a single-document application, meaning you can work on only one scene at a time. However, you can open more than one copy of 3ds Max and open a different scene in each copy. Opening additional copies of 3ds Max requires a lot of RAM. For the best performance, you should plan to open one copy and work on one scene at a time.

Opening multiple copies of 3ds Max is not supported in Windows ME.

Modeling Objects

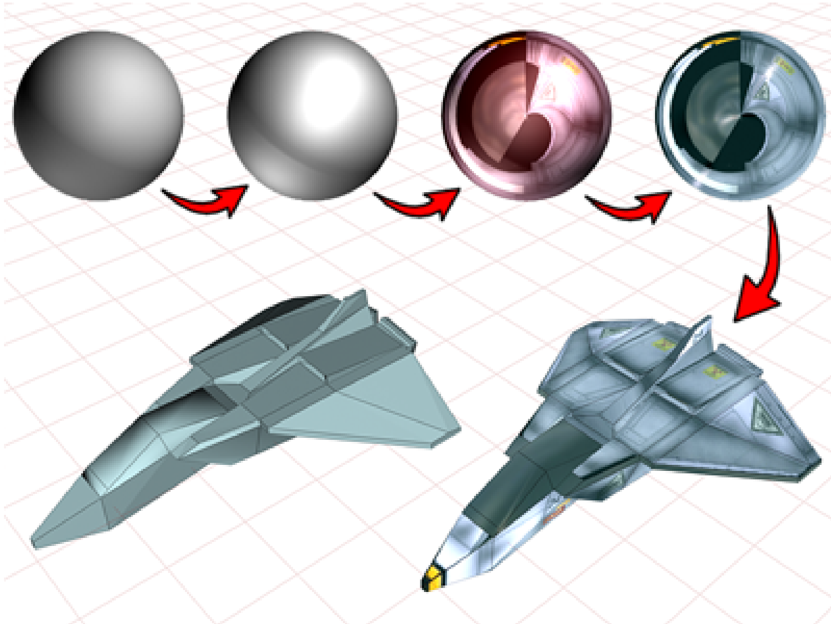


You model and animate objects in the viewports, whose layout is configurable. You can start with a variety of 3D geometric primitives. You can also use 2D shapes as the basis for lofted or extruded objects. You can convert objects to a variety of editable surface types, which you can then model further by pulling vertices and using other tools.

Another modeling tool is to apply modifiers to objects. Modifiers can change object geometry. Bend and Twist are examples of modifiers.

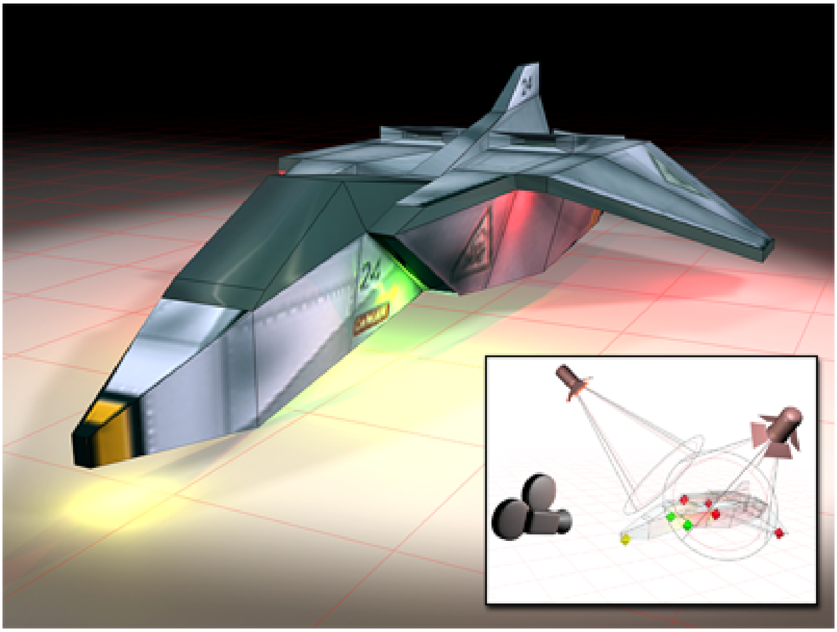
Modeling, editing, and animation tools are available in the command panels and toolbar. See Modeling Objects on page 58. Also, you can learn a good deal about modeling from the tutorials available from Help menu > Tutorials.

Material Design



You design materials using the Material Editor, which appears in its own window. You use the Material Editor to create realistic materials by defining hierarchies of surface characteristics. The surface characteristics can represent static materials, or be animated. See Material Editor on page 5284. Tutorials especially helpful for learning about materials include "Overview of Creating a Scene: Still Life" and "Using Materials."

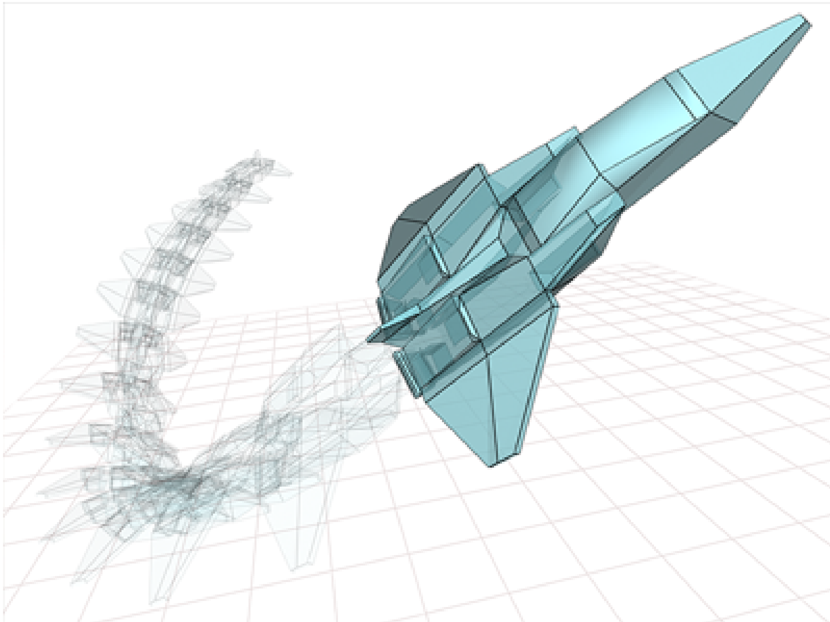
Lights and Cameras



You create lights with various properties to illuminate your scene. The lights can cast shadows, project images, and create volumetric effects for atmospheric lighting. Physically-based lights let you use real-world lighting data in your scenes and Radiosity on page 6168 provides incredibly accurate light simulation in renderings. See Lights on page 4970. You can learn more about lighting by following the Introduction to Lighting tutorial.

The cameras you create have real-world controls for lens length, field of view, and motion control such as truck, dolly, and pan. See Cameras on page 5194.

Animation

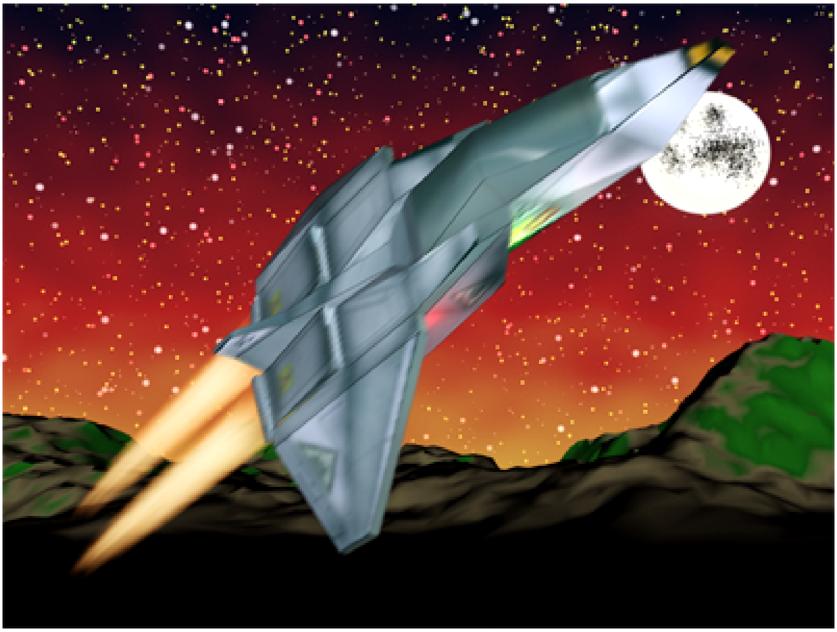


You can begin animating your scene at any time by turning on the Auto Key button. Turn the button off to return to modeling. You can also perform animated modeling effects by animating the parameters of objects in your scene. You can learn more about animating in the Animating Your Scene topic on page 65 and from most of the tutorials.

When the Auto Key button is on, 3ds Max automatically records the movement, rotation, and scale changes you make, not as changes to a static scene, but as *keys* on certain *frames* that represent time. You can also animate many parameters to make lights and cameras change over time, and preview your animation directly in the 3ds Max viewports.

You use Track View on page 3503 to control animation. Track View is a floating window where you edit animation keys, set up animation controllers, or edit motion curves for your animated effects. The Lip Sync tutorial covers Track View usage.

Rendering



Rendering adds color and shading to your scene. The renderers available with 3ds Max include features such as selective ray tracing, analytical antialiasing, motion blur, volumetric lighting, and environmental effects. See Rendering Your Scene on page 66. The tutorials can help you learn about rendering.

When you use the default scanline renderer, a radiosity solution on page 6168 can provide accurate light simulation in renderings, including the ambient lighting that results from reflected light. When you use the mental ray renderer, a comparable effect is provided by global illumination on page 6261.

If your workstation is part of a network, network rendering can distribute rendering jobs over multiple workstations. See Network Rendering on page 6433.

With Video Post on page 6773, you can also composite the scene with animations stored on disk.

A Typical Project Workflow

These topics explain the basic procedures for creating scenes:

Setting Up Your Scene on page 57

Setting Up Your Scene

You start with a new unnamed scene when you open the program. You can also start a new scene at any time by choosing New or Reset from the File menu.

Choosing a Unit Display

You choose a system of unit display on the Units Setup dialog on page 7809. Choose from Metric, Standard US, and Generic methods, or design a custom measuring system. You can switch between different systems of unit display at any time.

NOTE For best results, use consistent units when you are going to:

- Merge scenes and objects on page 7058.
- Use XRef objects on page 6936 or XRef scenes on page 6959.

Setting the System Unit

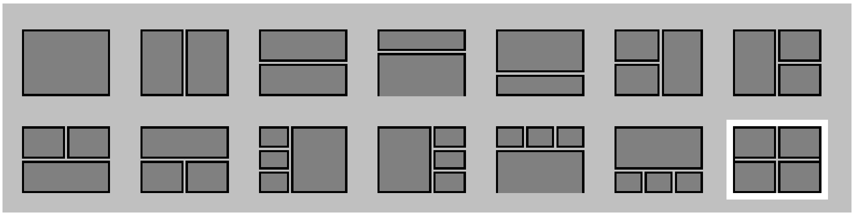
The System Unit setting, in the Units Setup dialog on page 7809, determines how 3ds Max relates to distance information you input to your scene. The setting also determines the range for round-off error. Consider changing the system unit value only when you model very large or very small scenes.

Setting Grid Spacing

Set spacing for the visible grid in the Grid And Snap Settings dialog > Home Grid panel on page 2675. You can change grid spacing at any time.

See Precision and Drawing Aids on page 2585 for information about the system unit, unit display, and grid spacing.

Setting the Viewport Display



Viewport layout options

The default four viewports in 3ds Max represent an efficient and popular screen layout. Set options in the Viewport Configuration dialog on page 7817 to change viewport layout and display properties.

See Viewing and Navigating 3D Space on page 87 for more information.

Saving Scenes

Save your scene frequently to protect yourself from mistakes and loss of work. See Backing Up and Archiving Scenes on page 85.

Modeling Objects

1. Modify panel

2. Create panel

3. Object categories

You model objects in your scene by creating standard objects, such as 3D geometry and 2D shapes, and then applying modifiers to those objects. The program includes a wide range of standard objects and modifiers.

Creating Objects

You create objects by clicking an object category and type on the Create panel and then clicking or dragging in a viewport to define the object's creation parameters. The program organizes the Create panel into these basic categories: Geometry, Shapes, Lights, Cameras, Helpers, Space Warps, and Systems. Each category contains multiple subcategories from which you can choose.

You can also create objects from the Create menu by choosing an object category and type and then clicking or dragging in a viewport to define the object's creation parameters. The program organizes the Create menu into these basic categories: Standard Primitives, Extended Primitives, AEC Objects, Compound, Particles, Patch Grids, NURBS, Dynamics, Shapes, Lights, Cameras, Helpers, Space Warps, and Systems.

See Basics of Creating and Modifying Objects on page 377.

Selecting and Positioning Objects

You select objects by clicking or dragging a region around them. You can also select objects by name or other properties such as color or object category.

After selecting objects, you position them in your scene using the transform tools Move, Rotate, and Scale. Use alignment tools to precisely position objects.

See Selecting Objects on page 191, Moving, Rotating, and Scaling Objects on page 929, and Precision and Drawing Aids on page 2585.

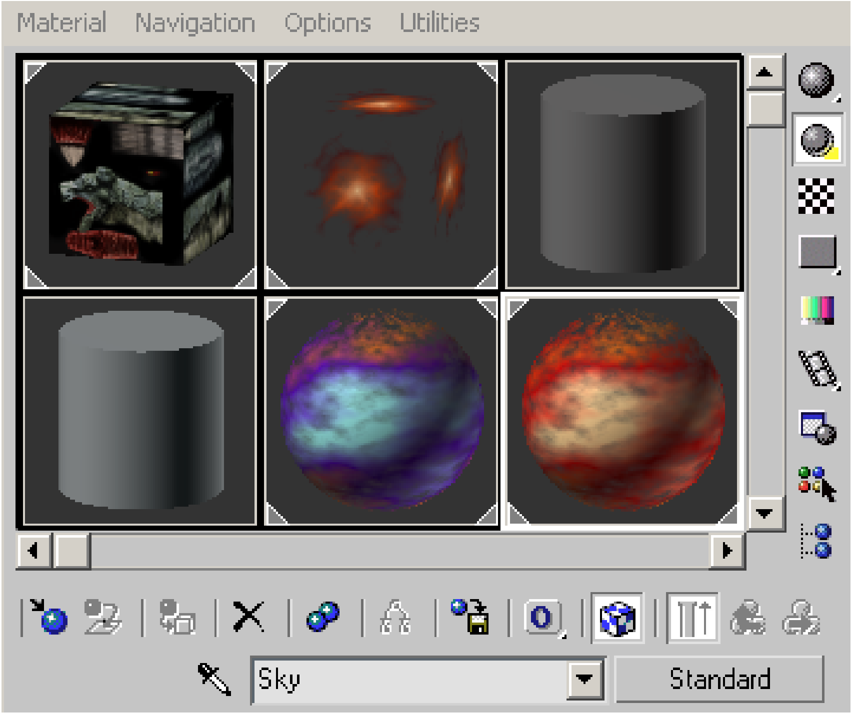
Modifying Objects

You sculpt and edit objects into their final form by applying modifiers from the Modify panel. The modifiers you apply to an object are stored in a stack. You can go back at any time and change the effect of the modifier, or remove it from the object.

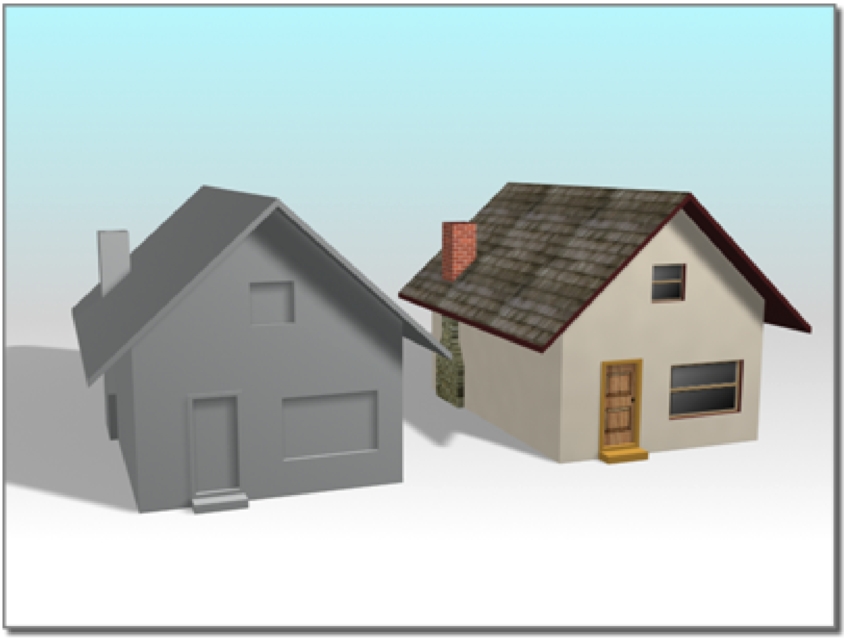
See Basics of Creating and Modifying Objects on page 377.

Using Materials

You use the Material Editor to design materials and maps to control the appearance of object surfaces. Maps can also be used to control the appearance of environmental effects such as lighting, fog, and the background.



A variety of materials in the Material Editor's sample slots



House on left uses the default standard material.

House on right uses a compound material.

Basic Material Properties

You set basic material properties to control such surface characteristics as default color, shininess, and level of opacity. You can create realistic, single-color materials using just the basic properties.

Using Maps

You extend the realism of materials by applying maps to control surface properties such as texture, bumpiness, opacity, and reflection. Most of the basic properties can be enhanced with a map. Any image file, such as one you might create in a paint program, can be used as a map, or you can choose procedural maps that create patterns based on parameters you set.

The program also includes a raytrace material and map for creating accurate reflections and refraction.

Viewing Materials in the Scene

You can view the effect of materials on objects in a shaded viewport, but the display is just an approximation of the final effect. Render your scene to view materials accurately.

See Designing Materials on page 5260.

Placing Lights and Cameras

You place lights and cameras to complete your scene in much the same way lights and cameras are placed on a movie set prior to filming.

Lights and cameras placed to compose a scene

The resulting scene

Default Lighting

Default lighting evenly illuminates the entire scene. Such lighting is useful while modeling, but it is not especially artistic or realistic.

Placing Lights

You create and place lights from the Lights category of the Create panel or menu when you are ready to get more specific about the lighting in your scene.

The program includes the following standard light types: omni, spot, and directional lights. You can set a light to any color and even animate the color to simulate dimming or color-shifting lights. All of these lights can cast shadows, project maps, and use volumetric effects.

See Guidelines for Lighting on page 4986.

Photometric Lights

Photometric lights on page 5005 provide you with the ability to work more accurately and intuitively using real-world lighting units (lumens and candelas). Photometric lights also support industry-standard photometric file formats (IES on page 5034, CIBSE on page 7935, LTLI on page 8029) so that you can model the characteristics of real-world manufactured luminaires, or even drag ready-to-use luminaires from the Web. Used in conjunction with the 3ds Max radiosity solution on page 6168, photometric lights let you evaluate more accurately (both physically and quantitatively) the lighting performance of your scene.

Photometric lights are available from the Create panel > Lights drop-down list.

Daylight System

The Daylight system on page 5139 combines sunlight on page 8140 and skylight on page 8129 to create a unified system that follows the geographically correct angle and movement of the sun over the earth at a given location. You can choose location, date, time, and compass orientation. You can also animate the date and time. This system is suitable for shadow studies of proposed and existing structures.

Viewing Lighting Effects in the Scene

When you place lights in a scene, the default lighting turns off and the scene is illuminated only by the lights you create. The illumination you see in a viewport is just an approximation of the true lighting. Render your scene to view lighting accurately.

TIP If the Daylight system appears to wash out the scene, try using the Logarithmic exposure control on page 6740.

Placing Cameras

You create and place cameras from the Cameras category of the Create panel. Cameras define viewpoints for rendering, and you can animate cameras to produce cinematic effects such as dollies and truck shots.

You can also create a camera automatically from a Perspective viewport by using the Create Camera from View command on page 168 found on the Views menu. Just adjust your Perspective viewport until you like it, and then choose

Views > Create Camera From View. 3ds Max creates a camera and replaces the Perspective viewport with a Camera viewport showing the same perspective.

See Common Camera Parameters on page 5210.

Animating Your Scene

You can animate almost anything in your scene. Click the Auto Key button to enable automatic animation creation, drag the time slider, and make changes in your scene to create animated effects.

Controlling Time

The program starts each new scene with 100 frames for animation. Frames are a way of measuring time, and you move through time by dragging the time slider on page 7528. You can also open the Time Configuration dialog on page 7565 to set the number of frames used by your scene and the speed at which the frames are displayed.

Animating Transforms and Parameters

While the Auto Key button is on, the program creates an animation key on page 8020 whenever you transform an object or change a parameter. To animate a parameter over a range of frames, specify the values at the first and last frames of the range. The program calculates the values for all of the frames in between.

See Animation Concepts and Methods on page 3078.

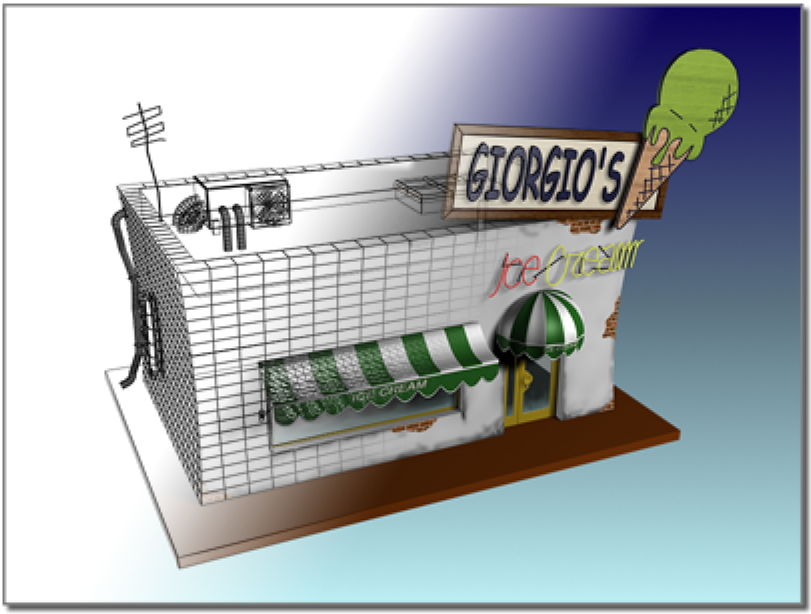
Editing Animation

You edit your animation by opening the Track View window or by changing options on the Motion panel. Track View is like a spreadsheet that displays animation keys along a time line. You edit the animation by changing the keys.

Track View has two modes. You can display the animation as a series of function curves that graphically show how a value changes over time in the Curve Editor mode. Alternatively, you can display your animation as a sequence of keys or ranges on a grid in the Dope Sheet mode.

See Track View on page 3503.

Rendering Your Scene



Rendering "fills in" geometry with color, shadow, lighting effects, and so on.

Use the rendering features to define an environment and to produce the final output from your scene.

Defining Environments and Backgrounds

Rarely do you want to render your scene against the default background color. Open the Environment And Effects dialog > Environment panel on page 6689 to define a background for your scene, or to set up effects such as fog.

Setting Rendering Options

To set the size and quality of your final output, you can choose from many options on the Render Setup dialog on page 6067. You have full control over professional grade film and video properties as well as effects such as reflection, antialiasing, shadow properties, and motion blur.

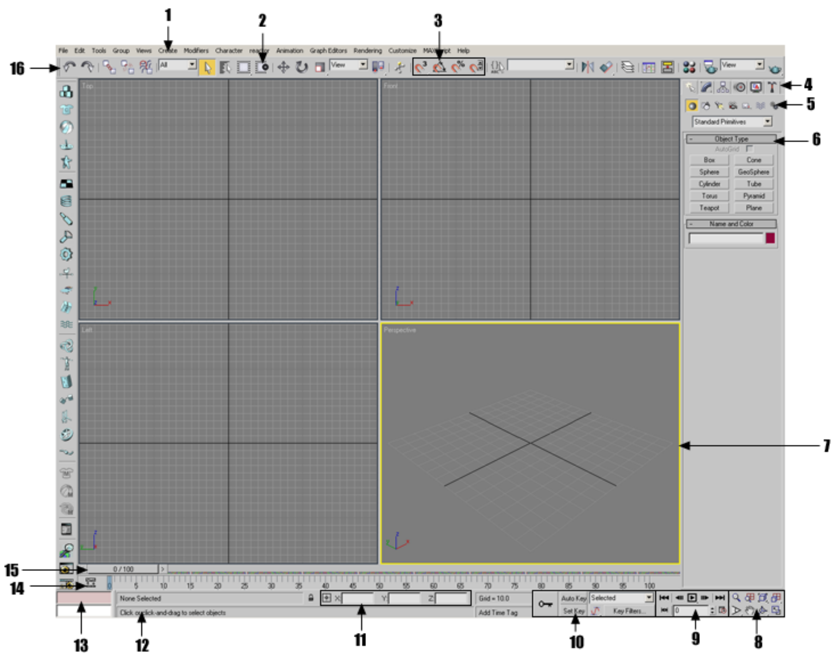
Rendering Images and Animation

You render a single image by setting the renderer to render one frame of your animation. You specify what type of image file to produce and where the program stores the file.

Rendering an animation is the same as rendering a single image except that you set the renderer to render a sequence of frames. You can choose to render an animation to multiple single frame files or to popular animation formats such as AVI or MOV.

See Render Setup Dialog on page 6067.

The 3ds Max Window



1. Menu bar

2. Window/Crossing selection toggle

3. Snap tools

4. Command panels

5. Object categories

6. Rollout

7. Active viewport

8. Viewport navigation controls

9. Animation playback controls

10. Animation keying controls

11. Absolute/Relative coordinate toggle and coordinate display

12. Prompt line and status bar

13. MAXScript mini-listener

14. Track bar

15. Time slider

16. Main toolbar

Most of the main window is occupied by the viewports, where you view and work with your scene. The remaining areas of the window hold controls and show status information.

One of the most important aspects of using 3ds Max is its versatility. Many program functions are available from multiple user-interface elements. For example, you can open Track View for animation control from the Main toolbar as well as the Graph Editors menu, but the easiest way to get to a specific object's track in Track View is to right-click the object, and then choose Track View Selected from the quad menu.

You can customize the user interface in a variety of ways: by adding keyboard shortcuts, moving toolbars and command panels around, creating new toolbars and tool buttons, and even recording scripts into toolbar buttons.

MAXScript lets you create and use custom commands in the built-in scripting language. For more information, access the MAXScript Reference from the Help menu.

Menu Bar

A standard Windows menu bar with typical File on page 7473, Edit on page 7474, and Help on page 7496 menus. Special menus include:

- Tools on page 7475 contains duplicates of many of the Main toolbar commands.
- Tools on page 7475 contains many important program functions, including precision functions.
- Group on page 7477 contains commands for managing combined objects.

- Views on page 7477 contains commands for setting up and controlling the viewports.
- Create on page 7478 contains commands for creating objects.
- Modifiers on page 7485 contains commands for modifying objects.
- Animation on page 7490 contains commands for animating and constraining objects, plus commands such as Bone Tools for setting up animated characters.
- Graph Editors on page 7493 provides graphical access to editing objects and animation: Track View lets you open and manage animation tracks in Track View on page 3503 windows, and Schematic View on page 7411 gives you an alternate way to work with the objects in your scene and navigate to them.
- Rendering on page 7493 contains commands for rendering, Video Post, radiosity, on page 6168and the environment.
- Rendering on page 7493 contains commands for rendering, using radiosity on page 6168, and changing the environment.
- Customize on page 7494 gives you access to controls that let you customize the user interface.
- MAXScript on page 7496 has commands for working with MAXScript, the built-in scripting language.

For more information about the 3ds Max menus, see Menu Bar on page 7471.

Time Controls

The Auto Key button on page 3083 turns on animation mode. The other controls navigate through time and play back an animation.

Command Panel

This collection of six panels provides handy access to most of the modeling and animation commands.

You can "tear off" the command panel and place it anywhere you like.

By default, the command panel is docked at the right of your screen. You can access a menu that lets you float on page 7957 or dismiss the command panel by right-clicking just above it. If it is not displayed, or you want to change its location and docking or floating status, right-click in a blank area of any toolbar, and choose from the shortcut menu.

- Create on page 7631 holds all object creation tools.
- Modify on page 7633 holds modifiers and editing tools.
- Hierarchy on page 7661 holds linking and inverse kinematics parameters.
- Motion on page 7663 holds animation controllers and trajectories.
- Display on page 7665 holds object display controls.
- Utilities on page 7671 holds miscellaneous utilities.

Status Bar and Prompt Line

These two lines display prompts and information about your scene and the active command. They also contain system toggles controlling selections, precision, and display properties. See Status Bar Controls on page 7524.

Viewports

You can display from one to four viewports. These can show multiple views of the same geometry, as well as the Track View, Schematic View, and other informational displays. See Viewports on page 7572.

Viewport Navigation Buttons

The button cluster at the lower-right corner of the main window contains controls for zooming, panning, and navigating within the viewports. See Viewport Controls on page 7572.

Special Controls

3ds Max uses some special user interface controls, which are described in this topic.

- Right-click menus on page 71
- Flyouts on page 71
- Rollouts on page 72
- Scrolling panels and toolbars on page 72

- Spinners on page 73
- Numerical Expression Evaluator on page 73
- Entering numbers on page 74
- Controls and color on page 74
- Undoing actions on page 75

Right-Click Menus

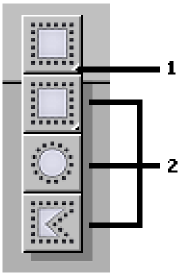
The program uses several different types of right-click menus.

For object editing and ActiveShade control on page 6111, you use the quad menu on page 7516. Commands on the quad menu vary depending on the kind of object you are editing and the mode you are in.

Right-clicking a viewport label displays the viewport right-click menu on page 7576, which lets you change viewport display settings, choose which view appears in the viewport, and so on.

Also, the command panel and the Material Editor have right-click menus that let you manage rollouts and navigate the panel quickly. And most other windows, including Schematic View and Track View, have right-click menus that provide fast access to commonly used functions.

Flyouts



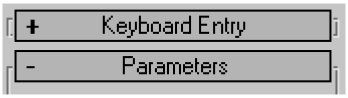
1. Flyout arrow

2. Flyout buttons

A flyout on page 7985 is similar to a menu, except that its items are buttons. A flyout button is indicated by a small arrow in the lower-right corner. To display the flyout, click and hold the button for a moment, then choose a button by dragging the cursor to it and then releasing the mouse button.

NOTE You can define customized text annotations for flyouts by editing the *maxstart.cui* file. See Customize Menu on page 7494.

Rollouts



Rollouts are areas in the command panels and dialogs that you can expand (roll out) or collapse (roll in) to manage screen space. In the illustration above, the Keyboard entry rollout is collapsed, as indicated by the + sign, and the Parameters rollout is expanded, as indicated by the – sign.

To open and close a rollout:

- Click the rollout title bar to toggle between expanded and collapsed.

To move a rollout:

- You can move a rollout in the expanded or collapsed state. To move the rollout, drag the rollout title bar to another location on the command panel or dialog. As you drag, a semi-transparent image of the rollout title bar follows the mouse cursor. When the mouse is positioned over or near a qualifying position for the rollout, a blue, horizontal line appears at the position where the rollout will drop when you release the mouse button.

Scrolling Panels and Toolbars

Sometimes a command panel or dialog is not large enough to display all of its rollouts. In this case, a pan ("hand") cursor appears over the inactive parts of the panel. You can scroll command panels and dialogs vertically, and you can scroll a toolbar along its major axis.

To scroll a panel:

1. Place the pointer over an empty area of a panel to display the pan cursor.
2. When the pointer icon changes to a hand, drag the panel up or down.

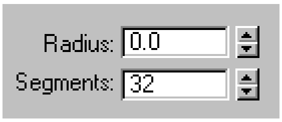
   A thin scroll bar also appears on the right side of the scrolling panel. You can use the pointer to drag the scroll bar as well.

To scroll a toolbar:

You can scroll a toolbar only when some tool buttons are not visible. This typically occurs when the program window is smaller than full screen.

1. First, follow either of the procedures below:
   - Place the pointer over an empty area of a toolbar to display the pan cursor.
   - Place the pointer over any part of a toolbar, then press and hold the middle mouse button.
2. When the pointer icon changes to a hand, drag the toolbar horizontally.

Spinners



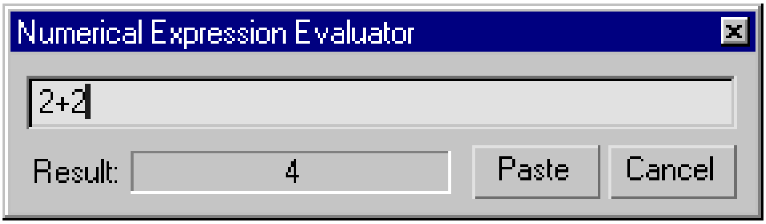
A spinner is a mouse-based control for numeric fields. You can click or drag the spinner arrows to change the value in the field.

To change a value using a spinner, do any of the following:

1. Click the spinner's up arrow to increment the value; click the down arrow to decrement the value. Click and hold for continuous change.
2. Drag upward to increase the value, or drag downward to decrease it.
3. Press Ctrl while you drag to increase the rate at which the value changes.
4. Press Alt while you drag to decrease the rate at which the value changes.
5. Right-click a spinner to reset the field to its minimum value.

Numerical Expression Evaluator

While a numeric field is active, you can display a calculator called the Numerical Expression Evaluator. To display the calculator, press Ctrl+N.



The expression you enter is evaluated, and its result is displayed in the Result field. Click Paste to replace the field value with the result of the calculation. Click Cancel to exit the Expression Evaluator.

The expressions you can enter are described in Expression Techniques on page 361. You can't use variables in the Expression Evaluator, but you can enter the constants pi (circular ratio), e (natural logarithm base), and TPS (ticks per second). These constants are case-sensitive: the Expression Evaluator does not recognize PI, E, or tps.

You can also enter a vector expression or an Expression Controller function call, but the result of the expression or function *must* be a scalar value. Otherwise, the Expression Evaluator won't evaluate it.

Entering Numbers

You can change a numeric value by a relative offset by highlighting the contents of a numeric field (not in the Numerical Expression Evaluator) and typing R or r followed by the offset amount.

For example, a Radius field shows 70 and you highlight it:

- If you enter **R30**, 30 is added to the radius and the value changes to 100.
- If you enter **R-30**, 30 is subtracted from the radius and the value changes to 40.

Controls and Color

The user interface uses color cues to remind you what state the program is in.

NOTE You can customize most of these colors by using the Colors panel on page 7712 of the Customize User Interface dialog on page 7697.

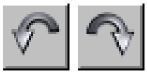
- **Red for animation:** The Auto Key button, the time slider background, and the border of the active viewport turn red when you are in Animate mode.

- **Yellow for modal function buttons:** When you turn on a button that puts you in a generic creation or editing mode, the button turns yellow.
- **Yellow for special action modes:** When you turn on a button that alters the normal behavior of other functions, the button is highlighted in yellow. Common examples of this behavior include sub-object selection and locking your current selection set.

You can exit a functional mode by clicking another modal button. Other exit methods supported by some buttons include right-clicking in a viewport, or clicking the modal button a second time.

Undoing Actions

You can easily undo changes you make to your scene and your viewports. There are separate Undo buffers for both the scene objects and each viewport.

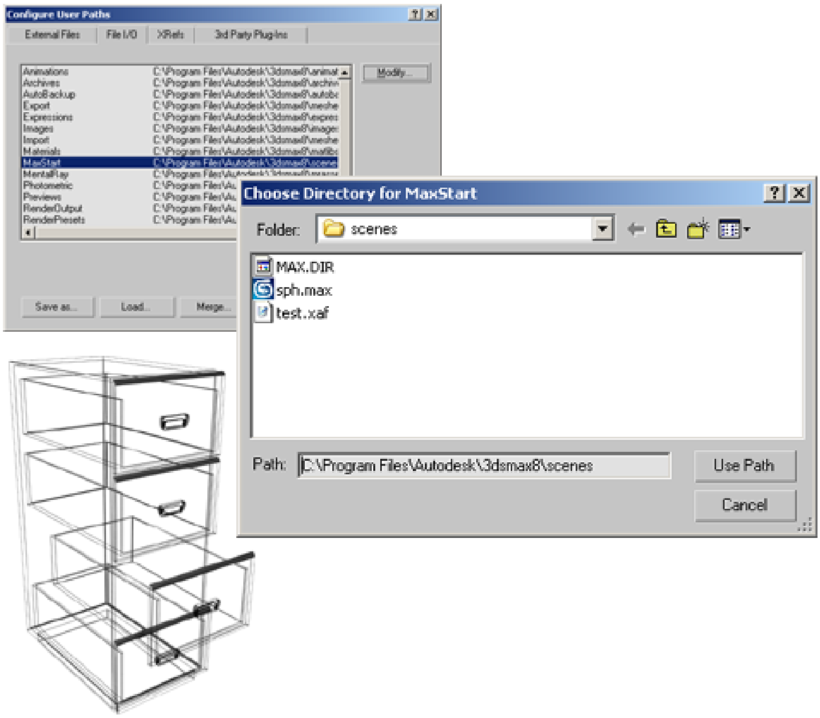


Use the toolbar Undo and Redo buttons on page 262 or the Edit menu > Undo and Redo commands to reverse the effects of most scene operations. You can also use Ctrl+Z for Undo and Ctrl+Y for Redo. Most things you do in the program can be undone.

Use the Views menu > Undo and Redo commands on page 146 to reverse the effects of most viewport operations, such as zooming and panning. You can also use Shift+Z for Undo View Change and Shift+Y for Redo View Change.

You can also undo actions by using the Hold and Fetch commands on the Edit menu. Choose Edit menu > Hold to save a copy of your scene in a temporary file. Then choose Edit menu > Fetch to discard your current scene and revert to the held scene at any time.

Managing Files



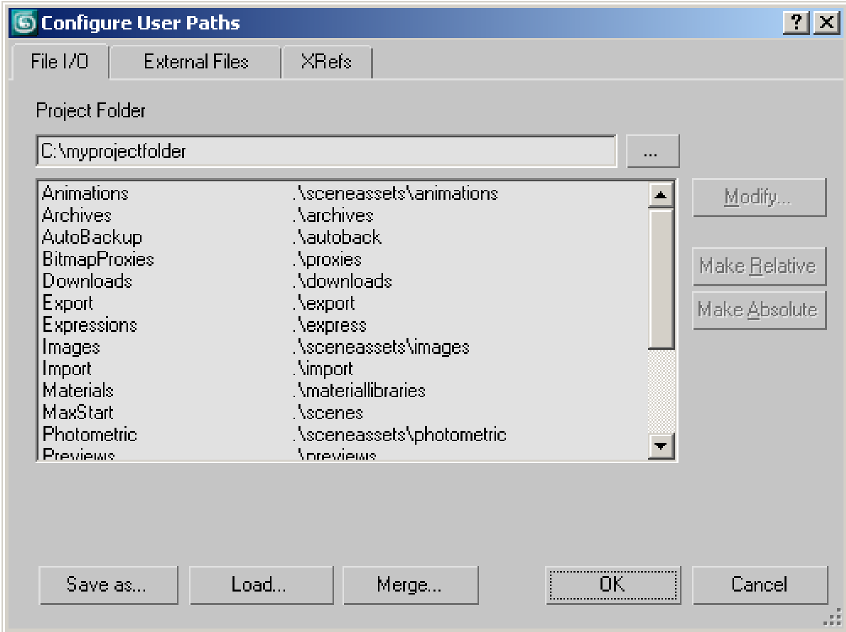
File-management dialogs

3ds Max supports many types of files for working with plug-ins, image maps, models from other programs, rendering images and animations, and of course saving and opening your scene files.

File dialogs (such as Open, Save, Save As) uniformly remember the previous path you used, and default to that location.

Configuring File Paths

The locations that 3ds Max searches to locate all file types are specified on the Customize menu > Configure Paths dialogs on page 7728.



You can choose to open and save files in any path location. The Configure Paths dialog contains four panels for the general categories of support files.

Setting General File Paths

The File I/O panel on page 7733 contains paths for most of the standard support files. You can specify one path for each of file types 3ds Max uses.

Setting Plug-In File Paths

Many features of 3ds Max are implemented as plug-ins. This means you can change and extend 3ds Max functionality by adding new plug-ins from Autodesk Media and Entertainment or from third-party developers.

You tell 3ds Max where to find additional plug-in files by adding path entries on the 3rd Party Plug-Ins panel on page 7740. If you place all of your plug-ins in a single directory, plug-in file management can become messy. That's why the program supports multiple entries on the 3rd Party Plug-Ins panel.

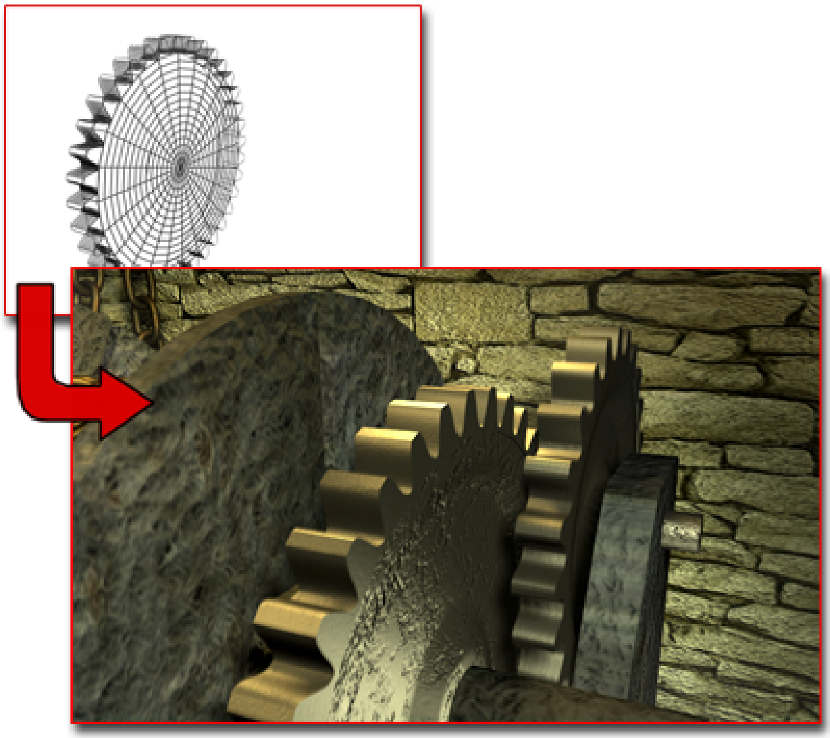
Setting Bitmap, FX, and Download File Paths

The External Files panel on page 7735 contains multiple path entries that the program searches for image files on page 7324, downloaded files (via i-drop on page 7160), and FX files on page 7990. Image files are used for many purposes, such as material and map definition, light projections, and environment effects.

Setting XRefs File Paths

The XRefs panel on page 7738 contains multiple path entries that the program uses to search for externally referenced files. These are used for sharing files in a workgroup situation.

Importing, Merging, Replacing, and Externally Referencing Scenes



Gear model imported to become part of another scene

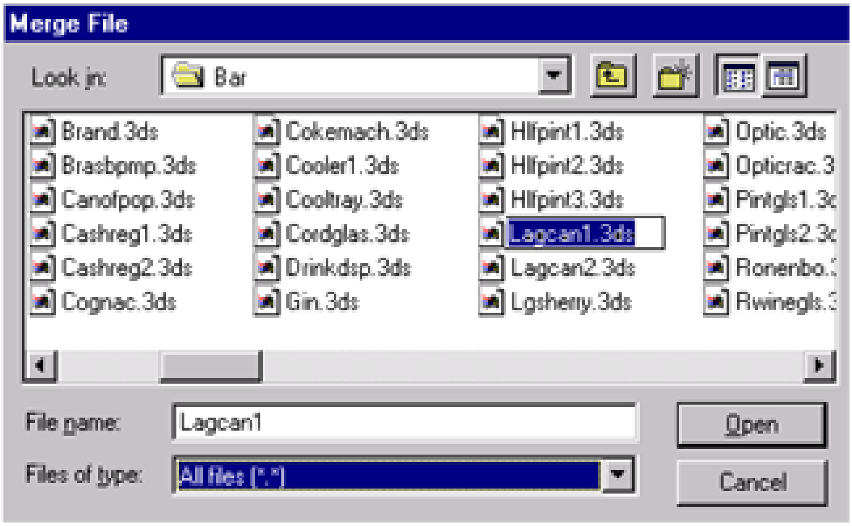
You can realize great productivity gains when you reuse work by combining geometry from scenes or other programs. 3ds Max supports this technique with the Import, Merge, and Replace commands. You can also share scenes and objects with others working on the same project using XRef functionality.

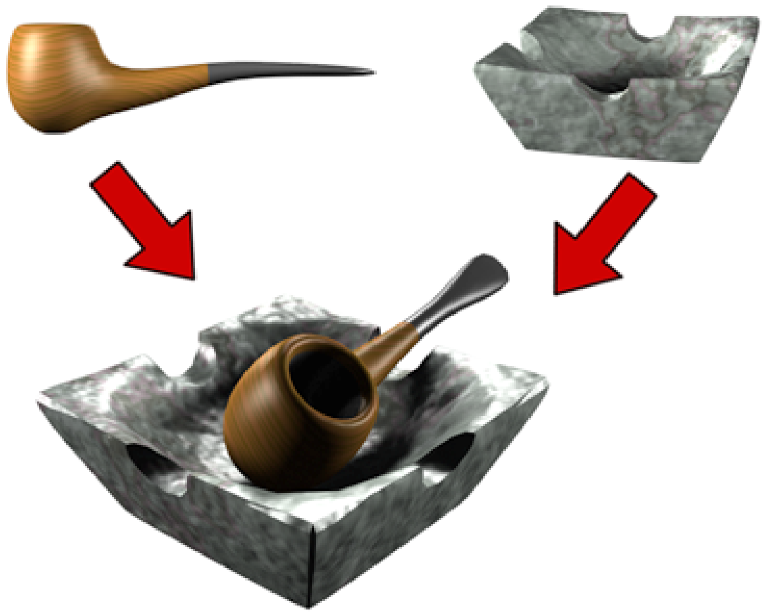
Importing Geometry from Other Programs

Use File menu > Import on page 7096 to bring objects from other programs into a scene. The types of files that you can import are listed in the Files Of Type list in the Select File To Import dialog.

Depending on the file type you choose, you might have options available for that import plug-in.

Merging Scenes Together





Pipe and ashtray models merged into one scene

Use Merge on page 7058 to combine multiple scenes into a single large scene. When you merge a file, you can select which objects to merge. If objects being merged have the same name as objects in your scene, you have the option to rename or skip over the merged objects.

Merging Animation into Scenes

Use Merge Animation on page 7063 to merge the animation from one scene into another with the same (or similar) geometry.

Replacing Scene Objects

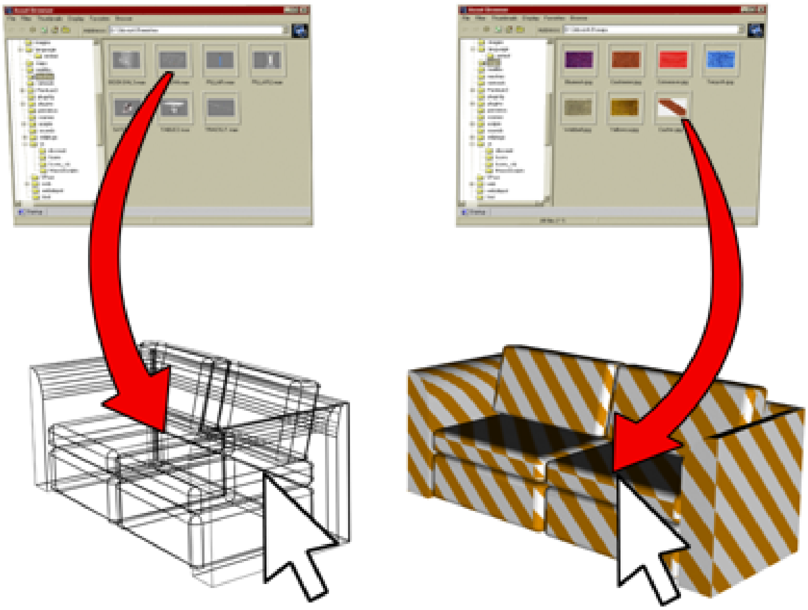
Use Replace on page 7070 to replace objects in your scene with objects in another scene that have duplicate names. Replace is useful when you want to set up and animate your scene with simplified objects, and then replace the simple objects with detailed objects before rendering.

The Replace dialog looks and functions the same as Merge, except that it lists only objects that have the same name as objects in your current scene.

Using External References

Use XRef Objects on page 6936 and XRef Scenes on page 6959 to use objects and scene setups in your scene that are actually referenced from external MAX files. These functions allow sharing files with others in your workgroup, with options for updating and protecting external files.

Using the Asset Browser



Left: Dragging geometry into the scene

Right: Dragging a bitmap onto the geometry

The Asset Browser provides access from your desktop to design content on the World Wide Web. From within 3ds Max you can browse the Internet for texture samples and product models. This includes bitmap textures (BMP, JPG, GIF, TIF, and TGA) and geometry files (MAX, 3DS, and so on).

You can drag these samples and models into your scene for immediate visualization and presentation. You can snap geometry into predefined locations, or drag and drop them interactively in your scene.

You can also use the Asset Browser to browse thumbnail displays of bitmap textures and geometry files on your hard disk or shared network drives. Then you can either view them or drag and drop them into your scene or into valid map buttons or slots.

NOTE The thumbnail display of a geometry file is a bitmap representation of a view of the geometry. Since the thumbnail display is not a vector-based representation, you can't rotate it or perform zooms on it.

You can drag and drop most graphic images that are embedded in a Web page into your scene. The exception is images or regions of a Web page that are tagged as hyperlinks or other HTML controls (such as when a bitmap is tagged as a button).

IMPORTANT Downloaded content might be subject to use restrictions or the license of the site owner. You are responsible for obtaining all content license rights.

For complete details, see Asset Browser on page 7132.

Startup Files and Defaults

When you start 3ds Max, several auxiliary files load, setting things like program defaults and UI layout. You can even create a scene, named *maxstart.max*, that automatically loads when you start or reset the program. In some cases, the program updates files when you change settings and when you quit the program.

NOTE 3ds Max comes with several different market-specific defaults on page 7694. These set different program defaults on startup, based on the type of files you expect to work on most often. You can load the preset defaults that come with 3ds Max, or you can create your own.

In general, you don't need to work directly with the auxiliary files, but it's good to know about them. Among the auxiliary files the program uses are:

- **3dsmax.ini on page 83** : This file gets updated when you start and exit 3ds Max, as well as when you change most Preferences settings. It contains values relating to program defaults, including the graphics driver, directories used to access external files such as sounds and images, preset render sizes, dialog positions, snap settings, and other preferences and default settings. If you edit this file, be sure to make a copy first, so you can return to the original if anything goes wrong.

NOTE Many program defaults are set in *currentdefaults.ini*, found within the *\defaults* directory. For more information on this file, see Market-Specific Defaults on page 7694.

- **maxstart.max**: At startup and when you reset the program, 3ds Max looks for this file in the *MaxStart* folder specified in Configure User Paths > File I/O panel on page 7733, and if found, loads it. This allows you to specify the default state of the workspace whenever you start or reset the program. For example, if you always use a ground plane, you can make it the default setup by creating one, and then saving it as *maxstart.max*.
  If you save a different file over *maxstart.max*, you can return to program defaults by deleting the *maxstart.max* file, and then resetting the program.
- *maxstart.cui*: This is the default custom user interface file. You can load and save CUI files, and set the program to use a different default CUI file. See Customize Menu on page 7494.
- *plugin.ini*: This file contains directory paths for plug-ins. Most other paths are kept in the program INI file, but *plugin.ini* is maintained as a separate file because third-party plug-ins often add entries to the list at installation.

  NOTE It is possible to use multiple plug-in configuration files by nesting additional paths in your *plugin.ini* file. This can be very useful for allowing an entire network of users to share one *plugin.ini* file, making the system easier to maintain for the network administrator. For more information, see Network Plug-In Configuration on page 7742.

- *startup.ms*: A MAXScript file that automatically executes at startup time. For more information, see Startup Script on page 8136.
- *splash.bmp*: To substitute a custom splash screen (startup screen) for the default image, copy any Windows Bitmap (*.bmp*) file into the program root directory and rename it *splash.bmp*. The program will thereafter use this image at startup.

The Initialization File

The file 3ds Max uses to store settings between sessions is named *3dsmax.ini*. By default, you can find it in the location indicated by the MaxData setting on the Configure System Paths dialog on page 7732.

You can make changes to 3ds Max startup conditions by editing the *3dsmax.ini* file in a text editor such as Notepad. If you do edit the file, be sure to maintain the structure and syntax of the original file.

TIP If you encounter unusual and unexplained user-interface problems using 3ds Max, try deleting the *3dsmax.ini* file and restarting. 3ds Max writes a new INI file to replace the deleted one. Often this fixes problems related to the state of the user interface.

NOTE Startup scene conditions are defined by the *maxstart.max* file. To save a particular startup condition, such as a Plane object representing the ground, create a scene file with the condition present and then save it as *maxstart.max*. 3ds Max automatically opens this file when you start 3ds Max.

The *3dsmax.ini* file includes the following categories of settings:

[Directories] Defines the default paths for various file operations.

[Performance] Controls that speed up viewport performance.

[PlugInKeys] Turns on or off the keyboard shortcuts for plug-ins.

[Renderer] Controls for rendering alpha and filter backgrounds.

[RenderPresets] Defines the paths for Rendering Preset files.

[BitmapDirs] Defines the default map paths for bitmaps used by materials.

[Modstack] Controls modifier stack button sets and icon display.

[WindowState] Settings for software display, OpenGL, or Direct3D drivers.

[CustomMenus] Defines path for the .**mnu** file.

[CustomColors] Defines the path for the *.clr* file.

[KeyboardFile] Defines the path for the *.kbd* file.

[Material Editor] Material Editor settings.

[ObjectSnapSettings] Settings associated with snaps.

[CommandPanel] Sets number of columns, and controls rollout display in multiple columns.

Backing Up and Archiving Scenes

You should regularly back up and archive your work. One convenient method is to save incremental copies of your scenes. This method creates a history of your work process.

Saving Incremental Files

If you turn on the Increment On Save option on the Files panel on page 7750 of the Preferences dialog, the current scene is renamed by appending a two-digit number to the end of the file and incrementing the number each time you save. For example, if you open a file named *myfile.max* and then save it, the saved file is named *myfile01.max*. Each time you save the file its name is incremented, producing the files *myfile02.max*, *myfile03.max*, and so on.

You can also use Save As on page 6928 to increment the file name manually with a two-digit number by clicking the increment button (+) on the Save As dialog.

Using Auto Backup

You can automatically save backup files at regular intervals by setting the Auto Backup options on page 7752 on the Preferences dialog (see File Preferences on page 7750). The backup files are named *AutoBackupN.max*, where N is a number from 1 to 99, and stored, by default, in the *\autoback* folder. You can load a backup file like any other scene file.

Archiving a Scene

3ds Max scenes can make use of many different files. When you want to exchange scenes with other users or store scenes for archival purposes, you often need to save more than just the scene file.

Use the File menu > Archive command on page 7121 to pass the scene file and any bitmap files used in the scene to an archiving program compatible with PKZIP® software.

Crash Recovery System

If 3ds Max encounters an unexpected crash, it attempts to recover and save the file currently in memory. This is fairly reliable, but it does not always work: the recovered scene could be damaged during the crash.

The recovered file is stored in the configured Auto Backup path. It is saved as "*<filename>_recover.max*" in this path. It is also placed in the INI file as the most recently used file in the File menu. This makes it easy to return to the file, if you choose to do so.

The crash recovery system identifies when something in an object's modifier stack is corrupt. In these cases, the corrupt object is replaced with a red dummy object to maintain the object's position and any linked object hierarchy.

NOTE We recommend that you not rely on this file-recovery mechanism as an alternative to good data backup practices:

- Save your work frequently.
- Take advantage of automatic incremental file naming: Go to Customize menu > Preferences > Files panel on page 7750 > File Handling group, and turn on Increment On Save.
- Use File menu > Save As, and click the Increment button (+) to save incremental copies of work in progress.
- If you are forgetful about saving, use the Auto Backup feature. Go to Customize menu > Preferences > Files tab > Auto Backup group, and turn on Enable.

Viewing and Navigating 3D Space

5

Everything in 3ds Max is located in a three-dimensional world. You have a variety of options for viewing this enormous stage-like space, from the tiniest details to the full extent of your scene.

Using the view options discussed in this section you move from one view to another, as your work and imagination require. You can fill your screen with a single, large viewport, or set multiple viewports to track various aspects of your scene. For exact positioning, flat drawing views are available, as are 3D perspective on page 8087 and axonometric views on page 7918.

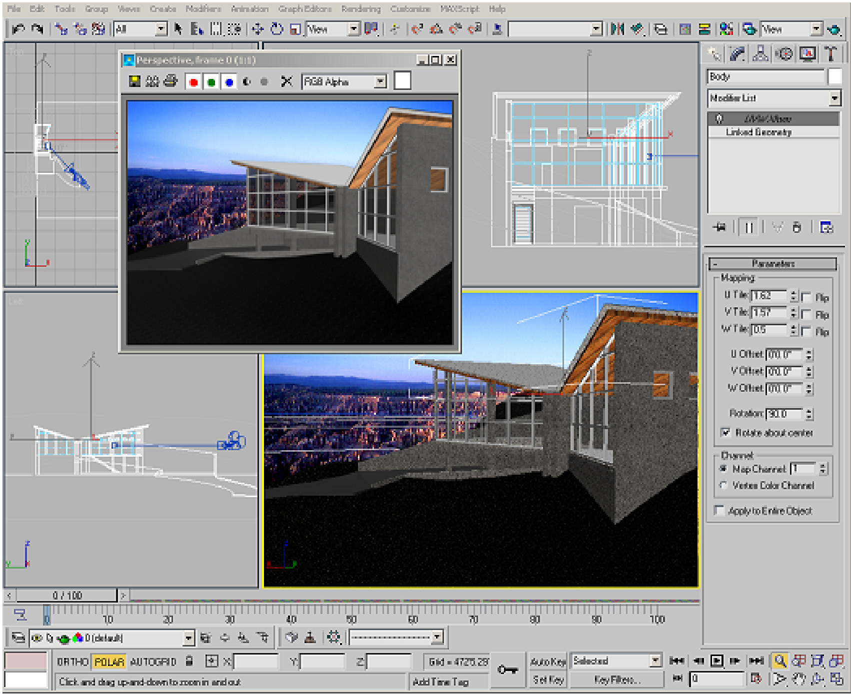
You navigate 3D space by adjusting the position, rotation and magnification of your views. You have full control over how objects are rendered and displayed on the screen.

You can also use the Grab Viewport command on page 145 to create snapshots of your work as you go.

This section presents these brief topics designed to help you quickly start learning how to organize viewports and navigate through 3D space:

For details about viewport commands, see Viewport Controls on page 7572.

General Viewport Concepts



The 3ds Max main window, with a docked toolbar and viewport layout displaying multiple views.

Viewports are openings into the three-dimensional space of your scene, like windows looking into an enclosed garden or atrium. But viewports are more than passive observation points. While creating a scene, you can use them as dynamic and flexible tools to understand the 3D relationships among objects.

At times you might want to look at your scene through a large, undivided viewport, giving you a "picture-window" view of the world you're creating. Often you use multiple viewports, each set to a different orientation.

If you want to move an object horizontally in the world space, you might do this in a top viewport, looking directly down on the object as you move it. At the same time, you could be watching a shaded perspective viewport to see when the object you're moving slides behind another. Using the two windows together, you can get exactly the position and alignment you want.

You also have pan and zoom features available in either view, as well as grid alignment. With a few mouse clicks or keystrokes, you can reach any level of detail you need for the next step in your work.

Another way to use viewports is to place a camera in your scene and set a viewport to look through its lens. When you move the camera, the viewport tracks the change. You can do the same thing with spotlights.

In addition to geometry, viewports can display other views such as Track View and Schematic View, which display the structure of the scene and the animation. Viewports can be extended to display other tools such as the MAXScript Listener and the Asset Browser. For interactive rendering, the viewport can display the ActiveShade window.

Active Viewport

One viewport, marked with a highlighted border, is always active. The active viewport is where commands and other actions take effect. Only one viewport can be in the active state at a time. If other viewports are visible, they are set for observation only; unless disabled, they simultaneously track actions taken in the active viewport.

Saving the Active Viewport

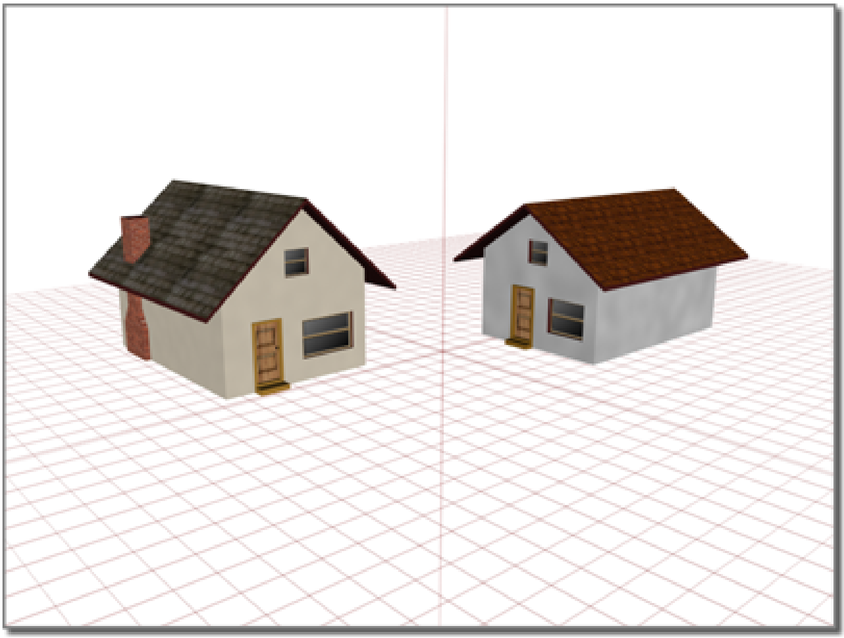
You can save the view in any active viewport and later restore it with the Views menu's Save Active View on page 147 and Restore Active View on page 147 commands. One view can be saved for each of the following view types: Top, Bottom, Left, Right, Front, Back, Orthographic, Perspective.

For example, while in the Front view, you choose Save Active Front View, and then zoom and pan that view. You then activate the Top viewport, choose Save Active Top View, and then click Zoom Extents. You return to the Front view, and choose Restore Active Front View to return to its original zoom and pan. At any time, you can activate the Top viewport, and then choose Restore Active Top View to restore its saved view.

Home Grid: Views Based on the World Coordinate Axes

The grid you see in each viewport represents one of three planes that intersect at right angles to one another at a common point called the origin. Intersection occurs along three lines (the world coordinate axes: X, Y, and Z) familiar from geometry as the basis of the Cartesian coordinate system.

Home Grid

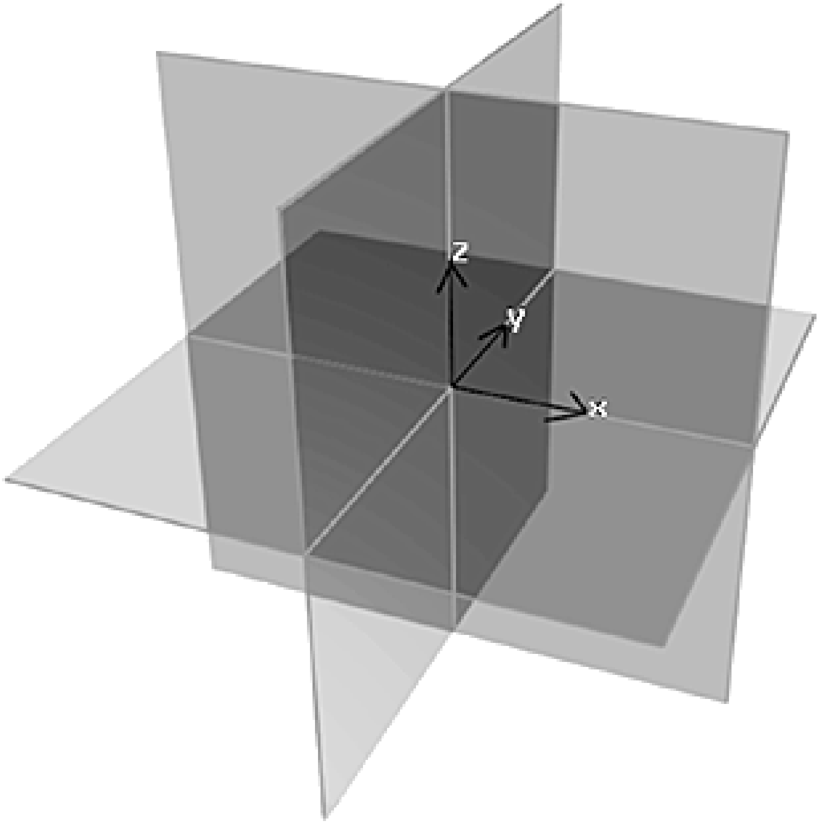


Using the home grid to position houses

The three planes based on the world coordinate axes are called the home grid; this is the basic reference system of the 3D world.

To simplify the positioning of objects, only one plane of the home grid is visible in each viewport. The figure shows all three planes as they would appear if you could see them in a single perspective viewport.

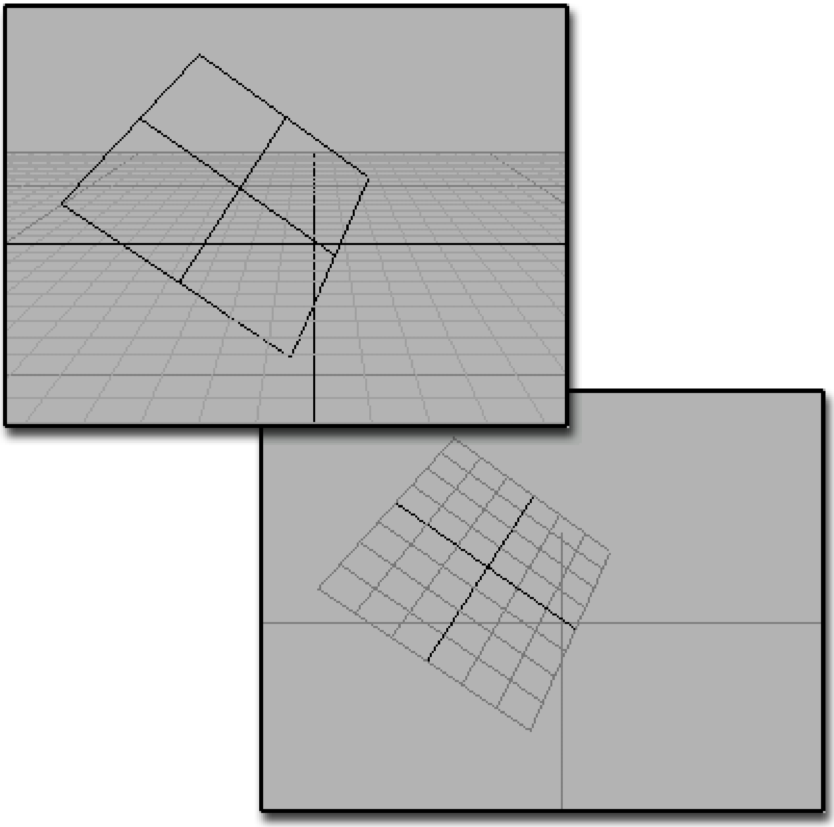
Axes, Planes, and Views



Home grid axes and planes

Two axes define each plane of the home grid. In the default Perspective viewport, you are looking across the XY plane (ground plane), with the X axis running left-to-right, and the Y axis running front-to-back. The third axis, Z, runs vertically through this plane at the origin.

Home Grid and Grid Objects

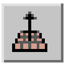


Above: Inactive grid object in a scene

Below: Activated grid object

The home grid is aligned with the world coordinate axes. You can turn it on and off for any viewport, but you can't change its orientation.

For flexibility, the home grid is supplemented by grid objects: independent grids you can place anywhere, at any angle, aligned with any object or surface. They function as "construction planes" you can use once and discard or save for reuse. See Precision and Drawing Aids on page 2585.



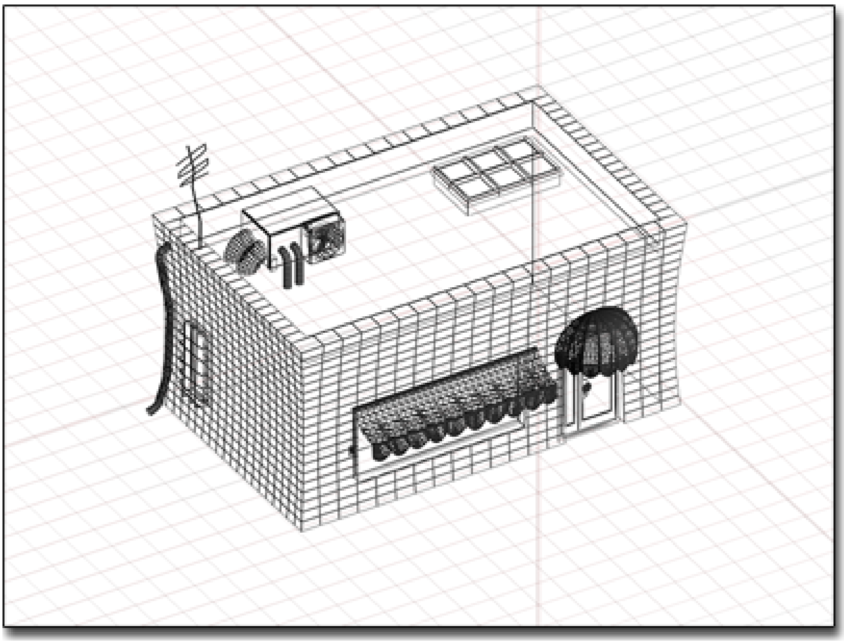
AutoGrid

The AutoGrid feature lets you create and activate temporary grid objects on the fly. This lets you create geometry off the face of any object by first creating the temporary grid, then the object. You also have the option to make the temporary grids permanent. See AutoGrid on page 2597.

Understanding Views

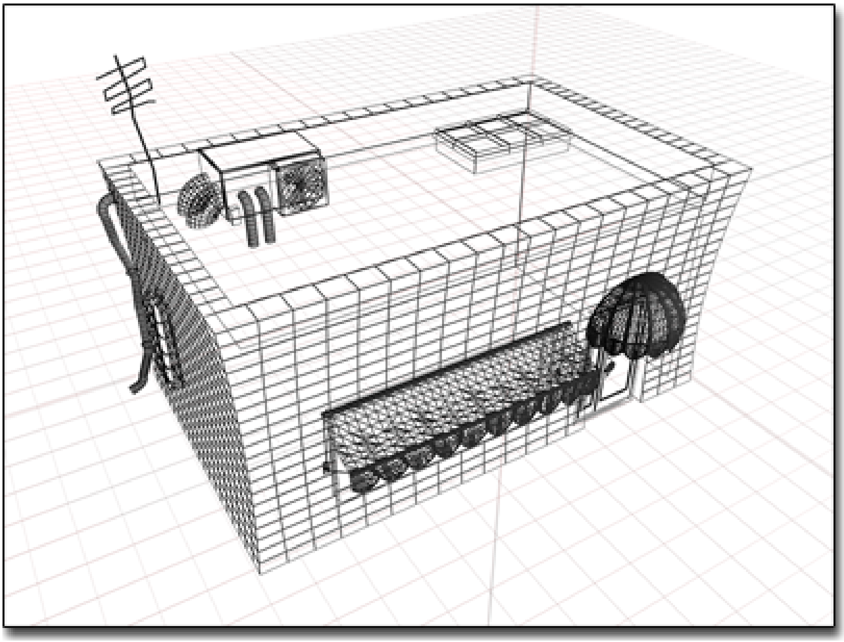
Each viewport can be set to display either of two types of views:

- Axonometric views on page 7918 show the scene without perspective. All lines in the model are parallel to one another. The Top, Front, Left, and Orthographic viewports are axonometric views.



Axonometric view of a scene

- Perspective views on page 8087 show the scene with lines that converge at the horizon. The Perspective and Camera viewports are examples of perspective views.



Perspective view of the same model

Perspective views most closely resemble human vision, where objects appear to recede into the distance, creating a sense of depth and space. Axonometric views provide an undistorted view of the scene for accurate scaling and placement. A common workflow is to use axonometric views to create the scene, then use a perspective view to render the final output.

Axonometric Views

There are two types of axonometric views you can use in viewports: head-on and rotated.

An orthographic view on page 8073 is often a head-on view of the scene, such as the view shown in the Top, Front, and Left viewports. You can set a viewport to a specific orthographic view using the viewport right-click menu on page 7576, keyboard shortcuts on page 7857, or the ViewCube on page 107. For example, to set an active viewport to Left view, press L.

You can also rotate an orthographic view to see the scene from an angle while retaining parallel projection. However, when viewing the scene from an angle, it's often more helpful to use a perspective view.

Perspective Views

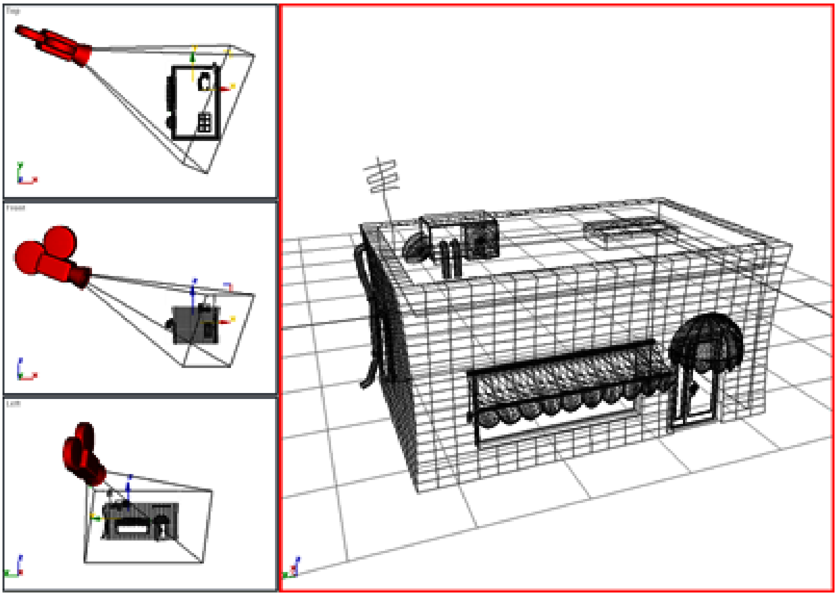
A perspective viewport, labeled Perspective, is one of the startup viewports in 3ds Max. You can change any active viewport to this "eye-like" point of view by pressing P.

Camera View

Once you create a camera object in your scene, you can change the active viewport to a camera view by pressing C and then selecting from a list of cameras in your scene. You can also create a camera view directly from a perspective viewport, using the Create Camera from View on page 168 command.

A camera viewport tracks the view through the lens of the selected camera. As you move the camera (or target) in another viewport, you see the scene move accordingly. This is the advantage of the Camera view over the Perspective view, which can't be animated over time.

If you turn on Orthographic Projection on a camera's Parameters rollout, that camera produces an axonometric view on page 7918. See Cameras on page 5194.



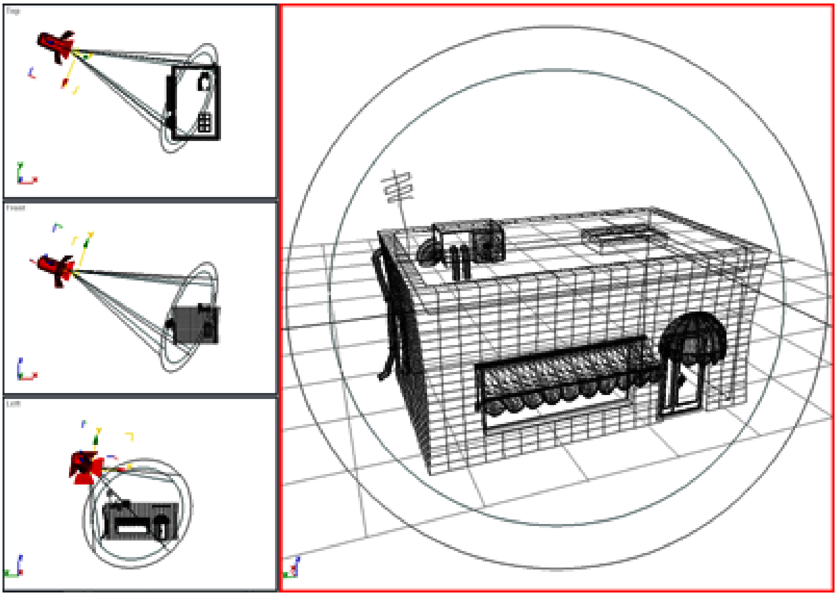
The viewport on the right is seen through a camera in the scene.

Two and Three-Point Perspective and the Camera Correction Modifier

By default, camera views use three-point perspective, in which vertical lines appear to converge with height (in traditional photography this is known as *keystoning*). The Camera Correction modifier on page 5247 applies two-point perspective to a camera view. In two-point perspective, vertical lines remain vertical. A similar effect can be attained by putting a Skew modifier on a camera.

Light View

Light view works much like a targeted camera view. You first create a spotlight or directional light and then set the active viewport to that spotlight. The easiest way is to press the keyboard shortcut $. See Lights on page 4970.

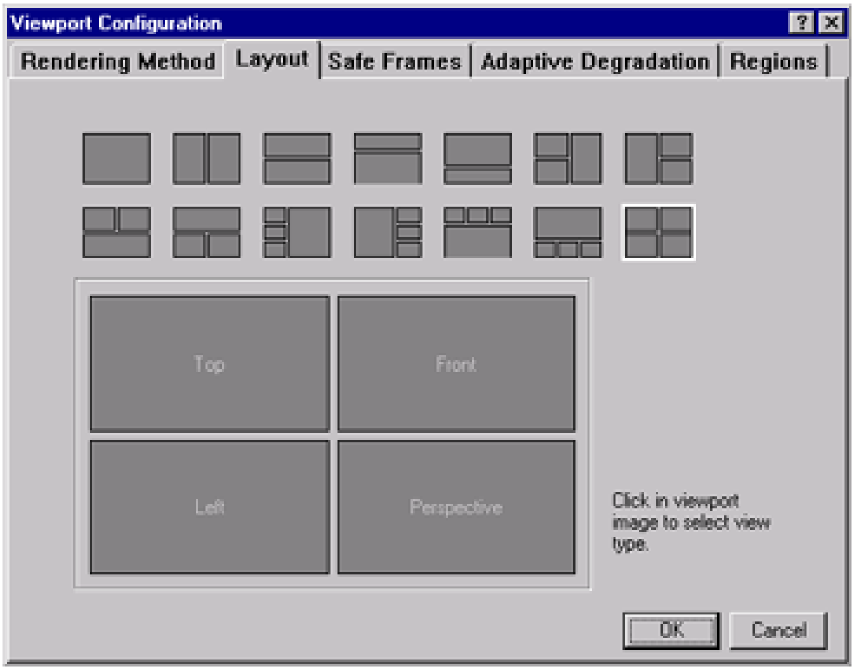


The viewport on the right looks through the lens of a spotlight in the scene.

Setting Viewport Layout

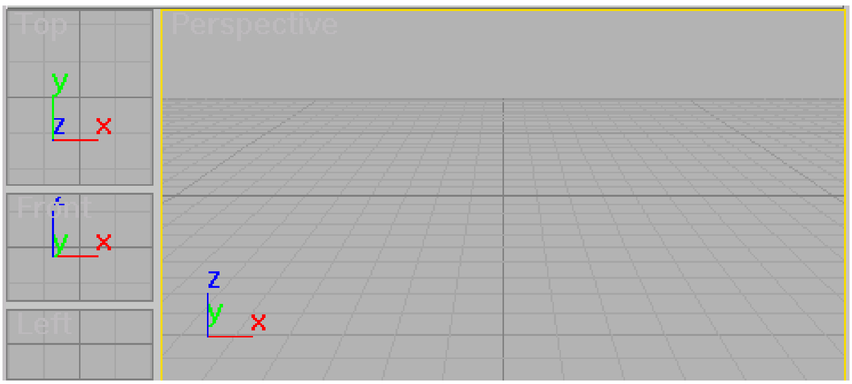
3ds Max defaults to a two-over-two arrangement of viewports. Thirteen other layouts are possible, but the maximum number of viewports on the screen remains four.

Using the Layout panel on page 7824 of the Viewport Configuration dialog, you can pick from the different layouts and customize the viewports in each. Your viewport configuration is saved with your work.



Resizing the Viewport

After choosing a layout you can resize the viewports so they have different proportions by moving the splitter bars that separate the viewports. This is only available when multiple viewports are displayed.



Resized viewport

Changing the View Type

As you work, you can quickly change the view in any viewport. For example, you can switch from front view to back view. You can use either of two methods: menu or keyboard shortcut.

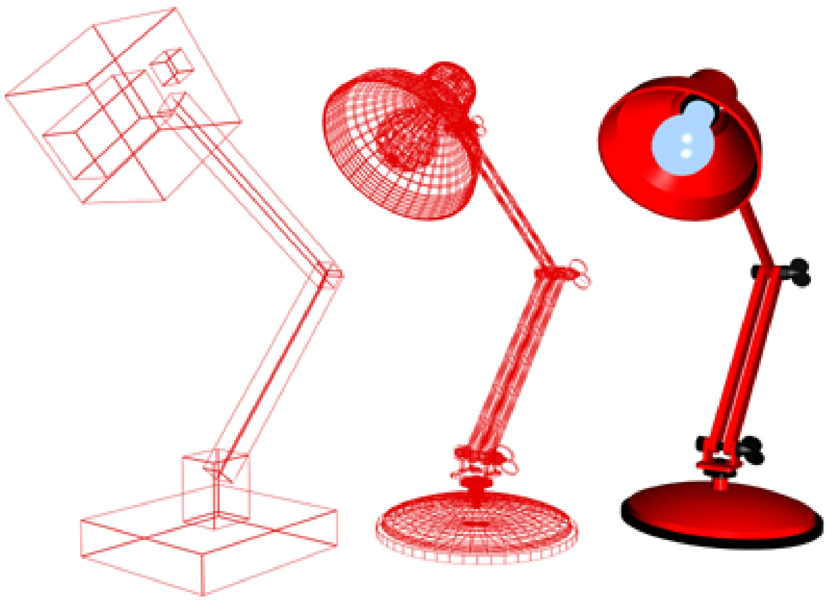
- Right-click the label of the viewport you want to change and click Views. Then, click the view type that you want.
- Click the viewport you want to change, and then press one of the keyboard shortcuts in the following table.

Key	View type
T	Top view
B	Bottom view
F	Front view
L	Left view
C	Camera view. If your scene has only one camera, or you select a camera before using this keyboard shortcut, that camera supplies the view. If your scene has more than one camera, and none are selected, a list of cameras appears.
P	Perspective view. Retains viewing angle of previous view.
U	Orthographic view. Retains viewing angle of previous view. Allows use of Zoom Region on page 7599.
none	Right view. Use viewport right-click menu.
none	Shape view. Use viewport right-click menu. Automatically aligns view to the extents of a selected shape and its local XY axes.

See also:

- Viewport Layout on page 7824
- Camera Viewport Controls on page 7604
- Spotlight Parameters on page 5090
- Precision and Drawing Aids on page 2585
- Track View on page 3503

Controlling Viewport Rendering



Box display, wireframe display, and smooth shading

You can choose from multiple options to display your scene. You can display objects as simple boxes, or render them with smooth shading and texture mapping. If you want, you can choose a different display method for each viewport.

TIP If you want to display an individual object as wireframe, you can use a Standard or Raytrace material and set its shader to Wire on page 5399. If you want an individual object to display as a box, you can select the object and choose Display as Box on page 184 on the Display properties rollout on the display panel.

Using Viewport Rendering Controls

Viewport rendering options are found on the Rendering Method panel on page 7818 of the Viewport Configuration dialog. Using this panel you choose a rendering level and any options associated with that level. You can then choose whether to apply those settings to the active viewport or all viewports, or to all but the active viewport.

The rendering level you choose is determined by your need for realistic display, accuracy, and speed. For example, Bounding Box display level is much faster than Smooth Shading with Highlights. The more realistic the rendering level, the slower the display speed.

After choosing a rendering level, you can set rendering options. Different options are available for different rendering levels.

You can also use ActiveShade on page 6110 in a viewport. This feature helps you quickly preview changes you make to lighting and materials.

Viewport rendering has no effect on final renderings produced by clicking Render Scene.

Rendering Methods and Display Speed

The rendering methods not only affect the quality of your view display, they can also have a profound effect on display performance. Using higher quality rendering levels and realistic options slows display performance.

After setting a rendering method, you can choose additional options that adjust display performance. One of these controls, Adaptive Degradation, speeds up display performance when you use realistic rendering levels.

See Rendering Method on page 7818.

Controlling Display Performance

3ds Max contains controls to help you adjust display performance: the balance between quality and time in displaying objects.

Depending on your needs, you might give up some display speed to work at higher levels of rendering quality, or you might choose to maximize display speed by using Wireframe or Bounding Box display. Which method you choose depends on your preferences and the requirements of your work.

Display Performance Controls

You use display performance controls to determine how objects are rendered and displayed.

Viewport Preferences

The Customize > Preferences dialog's Viewports panel contains options for fine-tuning the performance of the viewport display software. See Viewport Preferences on page 7753.

How Objects Are Displayed

To see and modify an object's display properties, right-click the object, select Properties, and go to the Display Properties group box; see Object Properties on page 305. These options affect display performance much the same way as viewport rendering options. For example, turning on Vertex Ticks for an object with a lot of vertices will slow performance.

NOTE Display Properties are only available when the By Object/By Layer toggle is set to By Object.

To see and modify how objects are displayed, you can use layers on page 7438. You can then quickly control the visibility and editability of similar objects from the quad menu.

Which Objects Are Displayed

One way to increase display speed is not to display something. You can use the Hide and Freeze functions on the Display panel, quad menu, Layer Manager on page 7441, and Scene Explorer on page 7379 to change the display state of objects in your scene. The Hide and Freeze features also affect final Rendering and Video Post output. See Hide Rollout on page 179 and Freeze Rollout on page 181.

Setting Adaptive Degradation

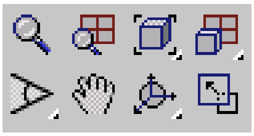
Adaptive Degradation dynamically adjusts viewport rendering levels to maintain a desired level of display speed. You have direct control over how much "degradation" occurs and when it occurs.

The active levels determine which rendering levels 3ds Max falls back to when it cannot maintain the desired display speed. You can choose as many levels as you want but you are advised to choose only one or two levels for each type of degradation.

See Adaptive Degradation on page 7900.

Using Standard View Navigation

To navigate through your scene, use the view navigation buttons located at the lower-right corner of the program window. All view types, except Camera and Light views, use a standard set of view navigation buttons.



The standard navigation controls

Button Operation

Clicking standard view navigation buttons produces one of two results:

- Executes the command and returns to your previous action.
- Activates a view navigation mode.

You can tell that you are in a navigation mode because the button remains highlighted (orange background). This mode remains active until you right-click or choose another command.

While in a navigation mode, you can activate other viewports of the same type, without exiting the mode, by clicking in any viewport. See Viewport Controls on page 7572.

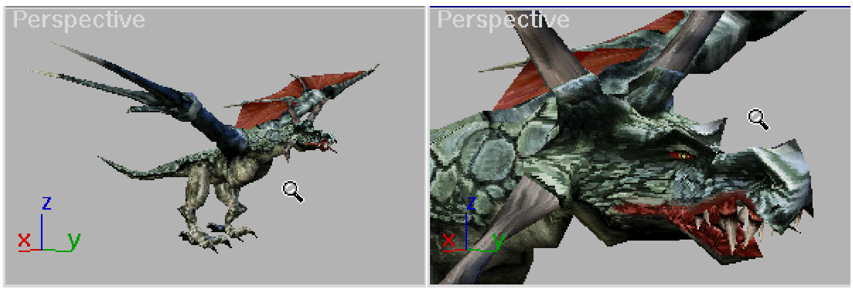
Undoing Standard View Navigation Commands

Use the Undo View Change and Redo View Change commands on page 146 on the Views menu to reset standard view navigation commands without affecting other viewports or the geometry in your scene. These commands are also found in the menu displayed when you right-click a viewport label. Or you can use the keyboard shortcuts: Shift+Z for Undo View Change and Shift+Y for Redo View Change.

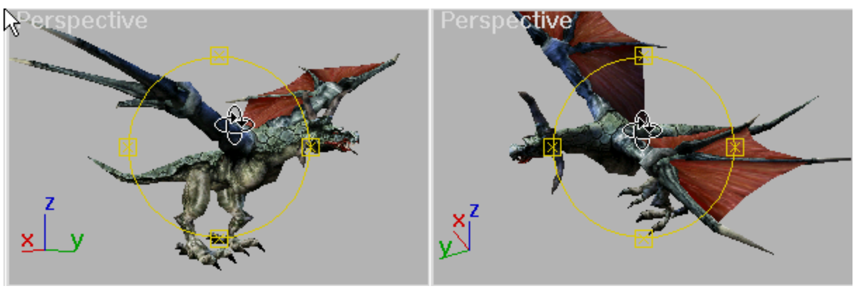
Views menu > Undo View Change and Views menu > Redo View Change are different from Undo and Redo commands on the Edit menu and main toolbar. 3ds Max maintains separate Undo/Redo buffers for scene editing and for each viewport.

The View Change Undo/Redo buffer stores your last 20 view navigation commands for each viewport. You can step back through the Undo View/Redo View buffer until you have undone all of the stored view-navigation commands.

Zooming, Panning, and Rotating Views

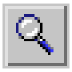


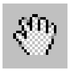
Before and after zooming a viewport

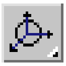


Before and after rotating a viewport

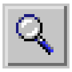
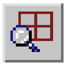
When you click one of the view navigation buttons, you can change these basic view properties:

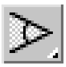
 **View magnification** Controls zooming in and out.

 **View position** Controls panning in any direction.

 **View rotation** Controls rotating in any direction

Zooming a View

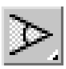
  Click Zoom on page 7592 or Zoom All on page 7593 and drag in a viewport to change the view magnification. Zoom changes only the active view, while Zoom All simultaneously changes all non-camera views.

 If a perspective view is active, you can also click Field of View (FOV) on page 7596. The effect of changing FOV is similar to changing the lens on a camera. As FOV gets larger you see more of your scene and perspective becomes distorted, similar to using a wide-angle lens. As FOV gets smaller you see less of your scene and the perspective flattens, similar to using a telephoto lens.

WARNING Be cautious using extreme Field of View settings. These can produce unexpected results.

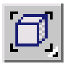
Zooming a Region

 Click Zoom Region on page 7599 to drag a rectangular region within the active viewport and magnify that region to fill the viewport. Zoom Region is available for all standard views.

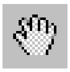
 In a perspective viewport, Zoom Region mode is available from the Field of View flyout on page 7596.

Zooming to Extents

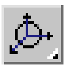
  Click the Zoom Extents or Zoom Extents All flyout buttons to change the magnification and position of your view to display the extents of objects in your scene. Your view is centered on the objects and the magnification changed so the objects fill the viewport.

-  The Zoom Extents, Zoom Extents Selected buttons on page 7588 zoom the active viewport to the extents of all visible or selected objects in the scene.
-  The Zoom Extents All, Zoom Extents All Selected buttons on page 7588 zoom all viewports to the extents of all objects or the current selection.

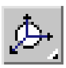
Panning a View

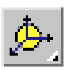
 Click Pan View on page 7600 and drag in a viewport to move your view parallel to the viewport plane. You can also pan a viewport by dragging with the middle mouse button held down while any tool is active.

Rotating a View

 Click Orbit, Orbit Selection, or Orbit Sub-Object on page 7602 to rotate your view around the view center, the selection, or the current sub-object selection respectively. When you rotate a head-on viewport, such as a Top view, it is converted to an Orthogonal view.

With Orbit, objects near the edges of the viewport can rotate out of view.

 With Orbit Selected, selected objects remain at the same position in the viewport while the view rotates around them. If no objects are selected, the function reverts to the standard Orbit function.

 With Orbit Sub-Object, selected sub-objects or objects remain at the same position in the viewport while the view rotates around them.

NOTE You can rotate a view by holding down the Alt key while you drag in a viewport using middle-button. This uses the current Orbit mode, whether or not the Orbit button is active. You can also activate Orbit by pressing Ctrl+R.

ViewCube

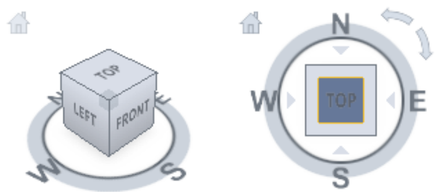
The ViewCube provides visual feedback of the current orientation of a viewport, and lets you adjust the view orientation.

Overview of the ViewCube

The ViewCube is a 3D navigation tool that also lets you switch between standard and isometric views.

Once the ViewCube is displayed, it appears by default in the upper-right corner of the active viewport, superimposed over the scene in an inactive state. It does not appear in camera, light, or shape viewports, or in other types of views such as ActiveShade or Schematic. While the ViewCube is inactive, its primary function is to show the orientation of the scene based on the North direction of the model.

When you position the cursor over the ViewCube, it becomes active. Using the left mouse button, you can switch to one of the available preset views, rotate the current view, or change to the Home view of the model. Right-clicking opens a context menu with additional options.



Control the Appearance of the ViewCube

The ViewCube is displayed in one of two states: inactive and active. When the ViewCube is inactive, it appears transparent over the viewport by default so as not to completely obscure the view of the model. When the ViewCube is active, it is opaque and might obscure the view of objects in the scene.

You can control the opacity level of the ViewCube when it is inactive as well as its size, the viewports it appears in, and the display of the compass. These settings are located on the ViewCube panel on page 7837 of the Viewport Configuration dialog.

Using the Compass

The ViewCube compass indicates the North direction for the scene. You can toggle the compass display below the ViewCube and specify its orientation with the Compass settings on page 7839.

Procedures

To display or hide the ViewCube:

1. Do either of the following:
   - Press the default keyboard shortcut: Alt+Ctrl+V.
   - Open the Viewport Configuration dialog to the ViewCube panel on page 7837 and toggle the Show The ViewCube check box.

To control the size and inactive opacity of the ViewCube:

1. Open the Viewport Configuration dialog to the ViewCube panel on page 7837.
2. In the Display Options group, click the ViewCube Size drop-down list and choose a size. The choices are Large, Normal, Small, and Tiny.
3. Also in the Display Options group, click the Inactive Opacity drop-down list and choose an opacity value. The choices range from 0% (invisible when inactive) to 100% (always fully opaque).
4. Click OK.

To display the compass for the ViewCube

1. Open the Viewport Configuration dialog to the ViewCube panel on page 7837.
2. In the Compass group, turn on Show Compass Below The ViewCube.

   The compass is displayed below the ViewCube and indicates the North direction in the scene.
3. Click OK.

ViewCube Menu

Right-click the ViewCube.

The ViewCube menu provides options to define the orientation of the ViewCube, switch between orthographic and perspective projection, define the Home and Front views for the model, and control the appearance of the ViewCube.

The following options can be found on the shortcut menu of the ViewCube:

- **Home** Restores the Home view saved with the model.
- **Orthographic** Switches the current view to orthographic projection.
- **Perspective** Switches the current view to perspective projection.
- **Set Current View as Home** Defines the Home view of the model based on the current view.
- **Set Current View as Front** Defines the Front view of the model based on the current view.
- **Reset Front** Resets the Front view of the scene to its default orientation.
- **Configure** Opens the Viewport Configuration dialog to the ViewCube panel on page 7837, where you can adjust the appearance and behavior of the ViewCube.
- **Help** Launches the online Help system and displays the topic on the ViewCube.

Procedure

To display the ViewCube menu:

- Right-click the compass, Home icon, or the main area of the ViewCube.

Change the View of the Scene with the ViewCube

You can change the view of a model by choosing a preset view or dragging the ViewCube, roll the current view 90 degrees, switch to an adjacent face view, define and restore the Front and Home views, and switch between projection modes.

Change the Current View

You can change the current view of a model by clicking predefined areas on the ViewCube or dragging the ViewCube.

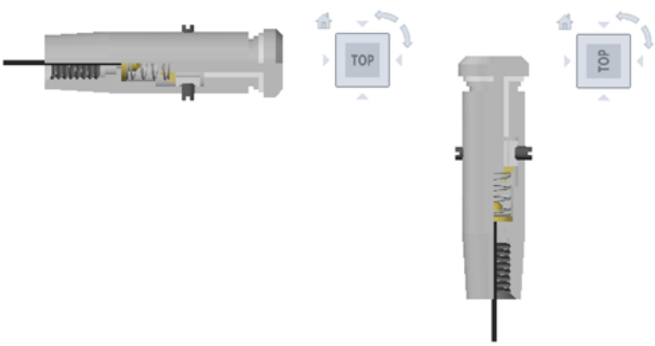
The ViewCube provides 26 defined areas you can click to change the current view of a model. The defined areas are categorized into three groups: corner, edge, and face. Of the26 defined areas, six represent standard orthogonal views of a model: top, bottom, front, back, left, and right. Orthogonal views are set by clicking one of the faces on the ViewCube.

You use the other 20 defined areas to access angled views of a model. Clicking one of the corners of the ViewCube changes the current view of the model to a three-fourths view, based on a viewpoint defined by three sides of the model. Clicking one of the edges changes the view of the model to a three-fourths view based on two sides of the model.

In addition, you can click and drag the ViewCube to change the view of the scene to a custom viewpoint other than one of the predefined viewpoints.

Roll a Face View

When you view a model from one of the face views such as Front, two additional icons, called *roll arrows*, are displayed near the ViewCube. To roll or rotate the current view 90 degrees in the positive or negative directions around the center of the view, click one of the roll arrows.



Switch to an Adjacent Face

While viewing a model from one of the face views, you can use the ViewCube to switch to one of the adjacent face views without first changing the view of the model, in order to see the adjacent view. When the ViewCube is active and a face view is current, four triangles are displayed, one on each side of the ViewCube. To rotate the current view to display the face view indicated by one of the triangles, click the triangle.

Front View

You can define the Front view of a model. The Front view is used to define the direction of the face views on the ViewCube. Along with the Front view, the Up direction of a model is also used to define the direction of the face views on the ViewCube.

Procedures

To change the current view to a preset view:

1. Activate the ViewCube.
2. Click one of the faces, edges or corners on the ViewCube.

To view an adjacent face:

1. Activate the ViewCube.

   NOTE Make sure a face view is current.

2. Click one of the triangles displayed near the edges of the ViewCube.

To roll a face view:

1. Activate the ViewCube.

   NOTE Make sure a face view is current.

2. Click one of the roll arrows displayed above and to the right of the ViewCube.

Click the left roll arrow to rotate the view 90 degrees in a counterclockwise direction or click the right roll arrow to rotate the view 90 degrees in a clockwise direction.

To change the view interactively:

- Click the ViewCube, hold down the button on your pointing device and drag to orbit the model.
  Drag in the direction that you want to orbit the model.

To define the Front view:

- Right-click on the ViewCube and click Set Current View as Front.

NOTE You first need to orient the view using a view tool so that you are looking at what you consider to be the Front of the model with its Top facing upward and then choose Set Current View as Front.

To restore the default Front view:

- Right-click the ViewCube and click Reset Front.

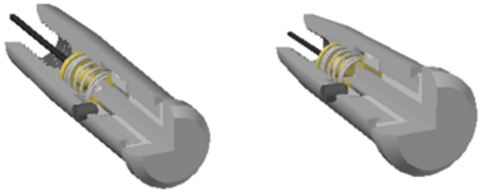
Set the View Projection Mode

View projection produces realistic visual effects of a model.

The ViewCube supports two different view projections:

- **Perspective** Perspective projected views involve foreshortening (closer objects appear larger than more distant ones) and are calculated based on the distance between a theoretical camera and target point. The shorter the distance between the camera and the target point, the more severe the perspective effect appears; greater distances produce less severe affects on the model.
- **Orthographic** Orthographic projected views display all the points of a model being projected parallel to the screen.

The following illustration shows the same model viewed from the same viewing direction, but with different view projections.



When you change the view for a scene, the view is updated using the current projection mode.

Procedure

To change the view projection mode

1 Right-click the ViewCube and click one of the following options:
   - Orthographic
   - Perspective

Define the Home View

You can define a Home view for a model so you can restore a familiar view when using the navigation tools.

The Home view is a special view of the model. The default Home position is the same as that of the Perspective viewport. You can also define a view of the scene as the Home view so you can easily return to the familiar view by clicking the Home icon or by choosing Home from the shortcut menu.

Procedure

To define the Home view

- Right-click the ViewCube and click Set Current View as Home.

To restore the Home view

Use one of the following methods to restore the Home view:

-  Click the Home icon located near the active ViewCube.

- Right-click the ViewCube and then choose Home.

SteeringWheels

SteeringWheels are tracking menus that allow you to access different 2D and 3D navigation tools from a single tool.

Overview of SteeringWheels

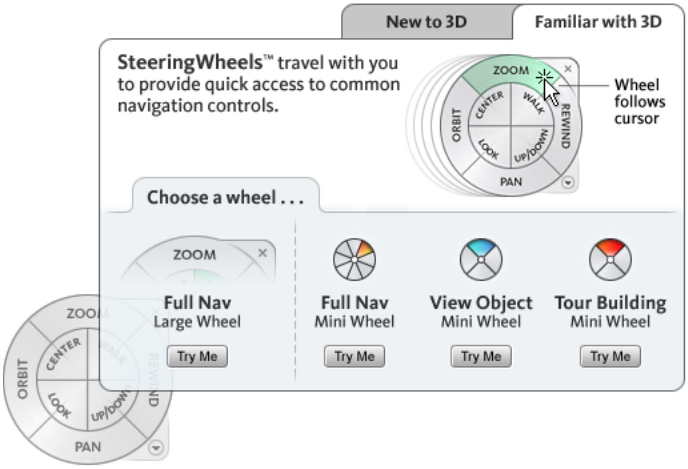
SteeringWheels are tracking menus that are divided into different sections known as wedges. Each wedge on a wheel represents a single navigation tool. You can pan, zoom, or manipulate the current view of a scene in different ways.

SteeringWheels, also known as wheels, can save you time by combining many of the common navigation tools into a single interface. Wheels are specific to the context that a scene is being viewed in.

The Full Navigation wheel

First Contact Balloon

By default, the wheel is “pinned” at startup to the lower-left corner of the Perspective viewport. It does not follow the cursor and, when you position the cursor over the wheel, the First Contact balloon for the SteeringWheels opens. The First Contact balloon serves as an introduction to the purpose of the wheels and how you can use them.



Display and Use Wheels

You can display a wheel from the Views menu or with a keyboard shortcut (Shift+W, by default). After a wheel is displayed, you can activate one of the available navigation tools either by clicking one of the wedges on the wheel or clicking and holding down the button on the pointing device. While the button is held down, dragging over the viewport causes the current view to change. Releasing the button returns you to the wheel, and right-clicking at any time closes the wheel.

Not all navigation tools on a wheel support click actions. The following navigation tools support click actions:

- **Zoom** - Adjusts the magnification of the view.
- **Center** - Centers the view based on the position of the cursor over the wheel.
- **Rewind** - Restores the previous view.
- **Forward** - Increases the magnification of the view.

Control the Appearance of Wheels

You can control the appearance of the wheels by changing the current mode, or by adjusting the size and opacity. Wheels are available in two different modes: big and mini. To use a different wheel, choose it from the Wheel menu on a full-size wheel (click the lower-right button on the wheel) or from the Views menu > SteeringWheels submenu.

In addition to changing the current mode, you can adjust the opacity and size for the wheels. The size of a wheel controls how large or small the wedges and labels appear on the wheel; the opacity level controls the visibility of the objects in the viewport behind the wheel. The settings used to control the appearance of the wheels are on the SteeringWheels panel on page 7839 of the Viewport Configuration dialog.

Control Tooltips for Wheels and Messages for Tools

Tooltips are displayed for each wedge and button on a wheel as the cursor hovers over them. The tooltips appear below the wheel and identify the action that is performed if the wedge or button is clicked. You can toggle the display of tooltips on the SteeringWheels panel on page 7839 of the Viewport Configuration dialog.

Similar to tooltips, tool messages are displayed when you use one of the navigation tools from a wheel. Tool messages are displayed over the viewport and provide instructions for using the active navigation tool. Like tooltips, you can turn tool messages on or off on the SteeringWheels panel on page 7839 of the Viewport Configuration dialog. Disabling tool messages affects only messages that are displayed when using the Full Navigation wheel.

Procedures

To close a wheel:

Use one of the following methods to close a wheel:

- Press the Esc key.
- Press Shift+W (this toggles the wheel).
- Click the Close button (the small x in the upper right-hand corner of the wheel).
- Right-click the wheel.

To change the size of the wheels:

1. Display a wheel, if necessary (press Shift+W).
2. Open the SteeringWheels panel on page 7839 of the Viewport Configuration dialog.
3. In the Display Options group, under Big Wheels or Mini Wheels, drag the Wheel Size slider left or right.

Dragging the slider to the left decreases the size of the wheel, while sliding the slider to the right increases the size of the wheel.

4 Click OK.

To change the opacity of the wheels:

1 Display a wheel, if necessary (press Shift+W).

2 Open the SteeringWheels panel on page 7839 of the Viewport Configuration dialog.

3 In the Display Options group, under Big Wheels or Mini Wheels, drag the Wheel Opacity slider left or right.

   Dragging the slider to the left increases the transparency of the wheel, while sliding the slider to the right decreases the transparency of the wheel.

4 Click OK.

To control the startup display of the wheels:

1 Open the SteeringWheels panel on page 7839 of the Viewport Configuration dialog.

2 In the Display Options group, toggle the Always Show Pinned Wheel On Start check box.

   When on, the wheel appears at the cursor position (pinned) whenever you start 3ds Max. When off, you must invoke the wheel explicitly (press Shift+W).

3 Click OK.

Wheel Menu

From the Wheel menu, you can switch among different wheels and change the behavior of some of the navigation tools on the current wheel.

Use the Wheel menu to switch among the big and mini wheels that are available, go to the Home view, change the wheels configuration, and control the behavior of the walk navigation tool. The availability of some items on the Wheel menu depends on the current wheel.

The Wheel menu has the following options:

- **Mini View Object Wheel** Displays the mini version of the View Object wheel.
- **Mini Tour Building Wheel** Displays the mini version of the Tour Building wheel.
- **Mini Full Navigation Wheel** Displays the mini version of the Full Navigation wheel.
- **Full Navigation Wheel** Displays the big version of the Full Navigation wheel.
- **Basic Wheels** Displays the big version of the View Object or Tour Building wheel.
- **Go Home** Restores the Home view saved with the scene.
- **Restore Original Center** Pans the view to the world center (0,0,0).
- **Increase Walk Speed** Doubles the walk speed used for the Walk tool.
- **Decrease Walk Speed** Halves the walk speed used for the Walk tool.
- **Help** Launches the online Help system and displays the topic about the wheels.
- **Configure** Displays the dialog box where you can adjust the preferences for the wheels.

Procedure

To display the wheel menu

- Click the down arrow in the lower-right corner of the wheel.

Navigation Wheels

You can choose from several different wheels. Each has its own drafting theme, and is designed for a different type of 3D navigation.

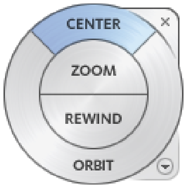
Wheels are available in two sizes: big and mini. The big wheel is larger than the cursor. A label is on each wedge in the wheel. The mini wheel is about the same size as the cursor. Labels are not displayed on the wheel wedges.

You can choose from the following wheels:

- View Object
- Tour Building
- Full Navigation

View Object Wheel

The View Object wheel is for general 3D navigation; it includes the orbit 3D navigation tool. Use the View Object wheel to examine 3D objects from the outside.



The big View Object wheel is divided into the following wedges:

- Center **on page 123** Specifies a point on a model to adjust the center of the current view or change the target point used for some of the navigation tools.
- Zoom **on page 133** Adjusts the magnification of the current view.
- Rewind **on page 129** Restores the most recent view. You can move backward or forward through previous views.
- Orbit **on page 127** Rotates the current view around a fixed pivot point.

The mini View Object wheel is divided into the following wedges:

- Zoom (Top wedge) **on page 133** Adjusts the magnification of the current view.
- Rewind (Right wedge) **on page 129** Restores the most recent view. You can move backward or forward through previous views.
- Pan (Bottom wedge) **on page 129** Repositions the current view by panning.
- Orbit (Left wedge) **on page 127** Rotates the current view around a fixed pivot point.

Procedures

To switch to the big View Objects wheel:

Use one of the following methods:

- From the Views menu, choose SteeringWheels > View Object Wheel.
- Click the wheel menu button at the lower-right corner of a big wheel and choose Basic Wheels > View Object Wheel.

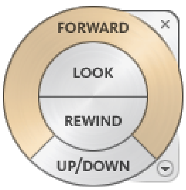
To switch to the mini View Objects wheel:

Use one of the following methods:

- Click the wheel menu button at the lower-right corner of a big wheel and choose Mini View Object Wheel.
- From the Views menu, choose SteeringWheels > Mini View Object Wheel.

Tour Building Wheels

The Tour Building wheels are designed for 3D navigation within the interior of a model.



The big Tour Building wheel is divided into the following wedges:

- **Forward on page 124** Adjusts the distance between the current point of view and the defined pivot point of the model.
- **Look on page 125** Swivels the current view.
- **Rewind on page 129** Restores the most recent view. You can move backward or forward through previous views.
- **Up/Down on page 131** Moves the view on the vertical axis of the screen.

The mini Tour Building wheel is divided into the following wedges:

- **Walk (Top wedge) on page 132** Simulates walking through a model.
- **Rewind (Right wedge) on page 129** Restores the most recent view. You can move backward or forward through previous views.
- **Up/Down (Bottom wedge) on page 131**Moves the view on the vertical axis of the screen.
- **Look (Left wedge) on page 125** Swivels the view.

Procedures

To switch to the big Tour Building wheel:

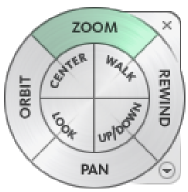
- From the Views menu, choose SteeringWheels > Tour Building Wheel.
- Click the wheel menu button at the lower-right corner of a big wheel and choose Basic Wheels > Tour Building Wheel.

To switch to the mini Tour Building wheel:

- Click the wheel menu button at the lower-right corner of a big wheel and choose Mini Tour Building Wheel.
- From the Views menu, choose SteeringWheels > Mini Tour Building Wheel.

Full Navigation Wheel

The Full Navigation wheel combines navigation tools found on the View Object and Tour Building wheels.



The big Full Navigation wheel is divided into the following wedges:

- **Zoom on page 133.** Adjusts the magnification of the current view.

- **Rewind on page 129.** Restores the most recent view. You can move backward or forward through previous views.
- **Pan on page 129.** Repositions the current view by panning.
- **Orbit on page 127.** Rotates the current view around a fixed pivot point.
- **Center on page 123.** Specifies a point on a model to adjust the center of the current view or change the target point used for some of the navigation tools.
- **Walk on page 132** Simulates walking through the scene.
- **Look on page 125** Swivels the view.
- **Up/Down on page 131** Moves the view on the vertical axis of the screen.

The mini Full Navigation wheel is divided into the following wedges:

- **Zoom (Top wedge) on page 133** Adjusts the magnification of the view.
- **Walk (Upper-right wedge) on page 132** Simulates walking through a model.
- **Rewind (Right wedge) on page 129** Restores the most recent view. You can move backward or forward through previous views.
- **Up/Down (Lower-right wedge) on page 131** Moves the view on the vertical axis of the screen.
- **Pan (Bottom wedge) on page 129** Repositions the current view by panning.
- **Look (Lower-left wedge) on page 125** Swivels the current view.
- **Orbit (Left wedge) on page 127** Rotates the current view around a fixed pivot point.
- **Center (Upper-left wedge) on page 123** Specifies a point on a model to adjust the center of the current view or change the target point used for some of the navigation tools.

Procedures

To switch to the big Full Navigation wheel:

Use one of the following methods:

- From the Views menu, choose SteeringWheels > Full Navigation Wheel.

■ Click the wheel menu button at the lower-right corner of a big wheel and choose Full Navigation Wheel.

To switch to the mini Full Navigation wheel:

Use one of the following methods:

■ Click the wheel menu button at the lower-right corner of a big wheel and choose Mini Full Navigation Wheel.

■ From the Views menu, choose SteeringWheels > Mini Full Navigation Wheel.

Navigation Tools

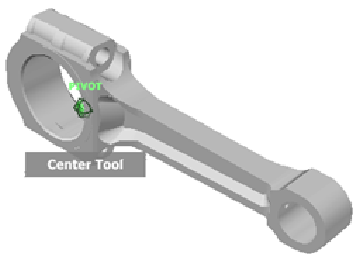
The navigation tools change the current view of the scene.

The availability of a navigation tool depends on the current wheel mode.

Center Tool

The Center tool specifies a point on an object as the center of the current view. It also changes the target point used for some of the navigation tools.

When using the Center tool, you adjust the location of the center of the current view by clicking and dragging. As you do so, the cursor changes to a sphere, indicating where the new center of the view will be established when you release the button on the pointing device. Releasing the button pans the model until the sphere is centered in the view and you are returned to the wheel.



The center point defined by the Center tool is used to constrain the Zoom tool and define the pivot point for the Orbit tool. The Zoom tool is only constrained to the center point when used from the View Object wheel unless the Ctrl key is held down when using the Zoom tool on the Full Navigation wheel.

Procedure

To specify a point on an object as the center of a view:

1. Display a wheel that has the Center tool.
2. Click the Center wedge. Hold down the button on your pointing device and drag the cursor to the object.
3. When the cursor changes to an arrow with a sphere at the tip, release the button on your pointing device.

   The viewport is panned until the sphere is centered.

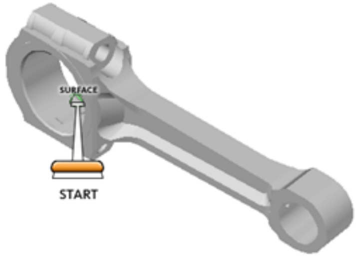
   NOTE This also sets the pivot point for the Orbit tool on page 127. Also, in the context of the View Object wheel on page 119, it sets the pivot point for the Zoom tool on page 133.

   ---
4. To exit the wheel, right-click.

Forward Tool

The Forward tool adjusts the distance between the current point of view and the defined pivot point of the model.

With the Forward tool, you can change the magnification of the model by increasing or decreasing the distance between the current point of view and the pivot point. The distance that you can move forward or backward is limited by the position of the pivot point.



To adjust the distance between the current point of view and the pivot point, you drag the cursor up or down after the pivot point is defined. As you drag the cursor, the current distance from the pivot point is displayed on a graphical element called the Drag Distance indicator. The Drag Distance indicator has two marks on it that show the start and ending distances from the current point of view. While changing the distance with the Drag Distance indicator, the current distance is shown by the bright orange indicator.

To change a view by moving towards or away from the model

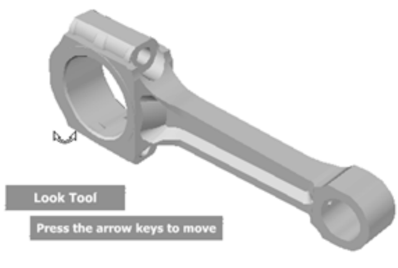
1. Display a wheel that has the Forward tool.
2. Click the Forward wedge. Hold down the button on your pointing device.
3. When the Drag Distance indicator is displayed, drag the cursor up or down to change the distance from which you view the model.
4. Release the button on your pointing device to return to the wheel.
5. To exit the wheel, right-click.

Look Tool

The Look tool rotates the view horizontally and vertically from a fixed point.

With the Look tool, you can rotate the current view vertically and horizontally. When rotating the view, the Look tool rotates your line of sight about the current eye position, like turning your head. Look is comparable to standing in a fixed location while looking up, down, left, and right.

When using the Look tool, you adjust the view of the model by dragging the cursor. As you drag the cursor, the cursor icon changes to the Look cursor and the model rotates around the location of the current view.



Walking through a Model

When using the Look tool, you can walk through a model by using the arrow keys on the keyboard. You can adjust the walk speed with the Walk tool settings on page 7841.

Invert Vertical Axis

By default, when you drag the cursor upward, the target point of the view raises; dragging the cursor downward lowers the target point of the view. If you prefer, you can reverse this behavior by turning on the Invert Vertical Axis on page 7841 option.

Procedures

To look around the scene:

1. Display a wheel that has the Look tool.
2. Click the Look wedge. Hold down the button on your pointing device.
3. When the Look cursor is displayed, drag the cursor up, down, left, and right to change the direction in which you are looking.
4. Release the button on your pointing device to return to the wheel.
5. To exit the wheel, right-click.

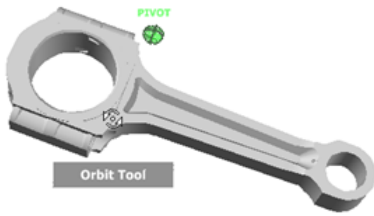
To look around the view and walk through the model

1. Display the Full Navigation wheel.
2. Click the Look wedge. Hold down the button on your pointing device.
3. When the Look cursor is displayed, drag the cursor up, down, left, and right to change the direction in which you are looking.
4. While holding down the button on your pointing device, press the arrow keys to walk in the model.
5. Release the button on your pointing device to return to the wheel.
6. To exit the wheel, right-click.

Orbit Tool

The Orbit tool rotates the current view around a model based on a fixed pivot point.

You use the Orbit tool to change the orientation of a model. The cursor changes to the Orbit cursor. As you drag the cursor, the model rotates around a pivot point while the view remains fixed.



Specify the Pivot Point

The pivot point is the base point used when rotating the model with the Orbit tool. You can specify the pivot point in the following ways:

- **Default pivot point** When you first open a model, the target point of the current view is used as the pivot point for orbiting the model.
- **Select objects** With Selection Sensitivity on page 7842 enabled, you can select objects before using the Orbit tool to calculate the pivot point. The pivot point is calculated based on the center of the extents of the selected objects.
- **Center tool**. You can specify a point on the model to use as the pivot point for orbiting with the Center tool on page 123.

Maintain Up Direction

You can control how the model orbits around the pivot point by choosing to maintain the Up direction of the model. When the Up direction is maintained, orbiting is constrained to the *XY* plane and along the *Z* axis. If you drag the cursor horizontally, the camera moves parallel to the *XY* plane. If you drag the cursor vertically, the camera moves along the *Z* axis. To control if the Up direction is maintained for the Orbit tool, use the Keep the Scene Upright on page 7841 option.

Procedures

To orbit around the center of the view:

1. Display a wheel that has the Orbit tool.
2. Click the Orbit wedge. Hold down the button on your pointing device.
3. When the cursor changes to the Orbit cursor, drag the cursor to rotate the model.

   NOTE If you need to change the part of the model that is displayed in the center of the view, use the Center tool.

   ---
4. Release the button on your pointing device to return to the wheel.
5. To exit the wheel, right-click.

To orbit around a selection set:

1. Press Esc to make sure no commands are active.
2. Make sure Selection Sensitivity on page 7842 is on.
3. Select the objects in the scene that are to define the pivot point location.
4. Display a wheel that has the Orbit tool.
5. Click the Orbit wedge. Hold down the button on your pointing device.
6. When the cursor changes to the Orbit cursor, drag the cursor to rotate the model.
7. Release the button on your pointing device to return to the wheel.
8. To exit the wheel, right-click.

To maintain the up direction for the Orbit tool:

1. Display a wheel that has the Orbit tool.
2. On the SteeringWheels panel on page 7839 of the Viewport Configuration dialog, make sure the Orbit Tool > Keep the Scene Upright on page 7841check box is on.

   Orbiting the model is constrained to the *XY* plane and *Z* axis.
3. Click OK.

Pan Tool

The Pan tool adjusts the viewpoint of the model by moving it within the screen plane.

When the pan tool is active, the cursor changes to a four-sided arrow. As you drag the cursor, the direction you drag moves the scene in the same direction. For example, dragging upward moves the scene up while dragging the cursor downward moves the scene down.

TIP If the cursor reaches the edge of the screen, you can continue panning by dragging the cursor further to force it to wrap around the screen.

Procedure

To pan the view with the wheel Pan tool:

1. Display a wheel that has the Pan tool.
2. Click the Pan wedge. Hold down the button on your pointing device and drag to reposition the model.
3. Release the button on your pointing device to return to the wheel.
4. To exit the wheel, right-click.

Rewind Tool

The Rewind tool restores the most recent view. You can also move backward and forward through a series of saved views.

As you pan, zoom, orbit, and use the other navigation tools (including the ViewCube) to change the view of a scene, each previous view is saved automatically to the navigation history. The navigation history holds the previous views along with a thumbnail image for each. A separate navigation history is maintained for each viewport; the histories are not saved with the scene.

With the Rewind tool, you can retrieve the navigation history of a model. You can restore the previous view or you can scroll through any of the views saved to the navigation history.

When you hold down the button on the pointing device over the Rewind tool on the wheel, the Rewind interface is displayed as a horizontal strip of thumbnails showing each stored view. You can scroll through the navigation history. To restore one of the previous views in the navigation history, drag the bracket horizontally in the Rewind interface. Dragging to the left goes to previous views, and if you've rewound but haven't changed the view, you can restore later stored views by "unwinding" to the right.

If you rewind to a previous view and then use a navigation tool, the software deletes the history after the current frame. However, it still remembers all saved views prior to the current frame.

Procedures

To restore the previous view:

1 Display a wheel.

2 Click the Rewind wedge.

3 Click Close to exit the wheel.

To restore a previous view with the Rewind interface:

1 Display a wheel.

2 Click the Rewind wedge. Hold down the button on your pointing device.

   The Rewind interface is displayed.

3 While holding down the button on your pointing device, drag to the left or to the right to restore a previous view.

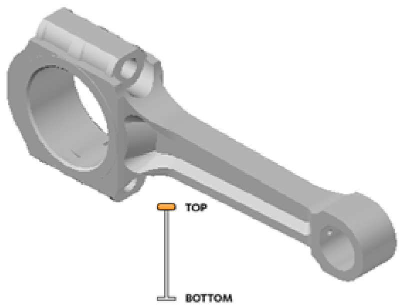
Dragging to the left restores an older previous view. Dragging to the right restores one of the more-recent previous views.The current position in the navigation history is indicated by the orange box that is dragged along the Rewind UI.

4 To exit the wheel, click Close.

Up/Down Tool

The Up/Down tool slides the current view of a model along the vertical screen axis.

Unlike the Pan tool, the Up/Down tool does not significantly displace the model being viewed because the view is sliding along the vertical axis of the screen. You can think of the Up/Down tool as similar to looking in a fixed direction while riding in a glass elevator.



To adjust the vertical elevation of the current view, drag the cursor up or down. As you drag, the current elevation and the allowed range of motion are shown on a graphical element called the Vertical Distance indicator. This indicator has two marks that show the highest (Top) and lowest (Bottom) elevation the view can have. While changing the elevation with the Vertical Distance indicator, the current elevation is shown by the bright orange indicator while the previous elevation is shown by the dim orange indicator.

Procedure

To change the elevation of a view:

1 Display a wheel that has the Up/Down tool.

2 Click the Up/Down wedge. Hold down the button on your pointing device.

3 When the Vertical Distance indicator is displayed, drag the cursor up or down to change the elevation of the view.

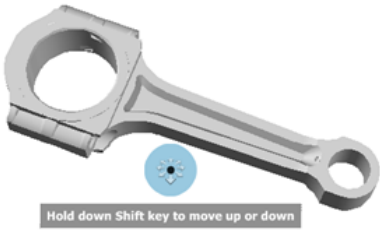
4 Release the button on your pointing device to return to the wheel.

5 To exit the wheel, click Close.

Walk Tool

The Walk tool simulates walking through a model.

With the Walk tool, you can navigate through a model as if you were walking through it. Once you start the Walk tool, the Center Circle icon is displayed near the center of the view and the cursor changes to display a series of arrows. To walk through the model, you drag the cursor in the direction that you want to move in.



Constrain the Walk Angle

When walking through a model, you can constrain the movement angle to the ground plane. If the Constrain Walk Movement Angle to Ground Plane on page 7841 option is on (the default setting), you can look around freely while the current view moves parallel to the ground plane. If the walk angle is not constrained, you "fly" in the direction you are looking.

Movement Speed

As you walk or "fly" through a model, you can control the movement speed. You control the walk speed by the distance you move the cursor from the Center Circle icon. To set the base movement speed, you use the Speed Factor on page 7841 setting or the Increase Walk Speed and Decrease Walk Speed options on the Wheel menu.

Change the Elevation of the View

As you use the Walk tool, you can adjust the elevation of the view by holding down the Shift key. You switch temporarily to the Up/Down tool on page 131. To change the current elevation of the view for the model, you drag up and down.

Procedure

To use the Walk tool to move through the model:

1. Display a wheel that has the Walk tool.
2. Click the Walk wedge. Hold down the button on your pointing device.
3. When the Center Circle icon is displayed, drag the cursor in the direction that you want to walk.
4. Release the button on your pointing device to return to the wheel.
5. Click Close to exit the wheel.

Zoom Tool

The Zoom tool adjusts the magnification of the current view of a model.

With the Zoom tool, you can change the zoom magnification of a model in the following ways:

- **Click** If you click the Zoom tool on a wheel, the current view is zoomed in by a factor of 25 percent. If you are using the Full Navigation wheel, Incremental Zoom-in on page 7841 must be enabled on the SteeringWheels panel of the Viewport Configuration dialog.
- Shift + **click** If you hold down the Shift key before you click the Zoom tool on a wheel, the current view is zoomed out by a factor of 25 percent.
- Ctrl+**click** If you hold down the Ctrl key before you click the Zoom tool on a wheel, the current view is zoomed in by a factor of 25 percent.
- **Click and drag** If you click the Zoom tool and hold down the button on your pointing device, you can adjust the magnification of the model by dragging up and down.

- Ctrl+**click and drag** When using the Full Navigation wheel, you can control the target point used by the Zoom tool. By holding down Ctrl, the Zoom tool uses the center point defined by the Center tool.

NOTE When you invoke the Zoom tool from the Full Navigation Wheel, Ctrl+click causes zooming to occur about the current pivot point rather than the current cursor position. Click and Shift+click perform incremental zooming only if Incremental Zoom-in on page 7841 is enabled on the SteeringWheels panel of the Viewport Configuration dialog.

Zoom Constraints

When changing the magnification of a model with the Zoom tool, you cannot zoom in any further than the focus point or out past the extents of the model. The direction you can zoom in and out is controlled by the center point set by the Center tool.

NOTE Unlike the Zoom tool on the View Object wheels, the Zoom tool on the Full Navigation wheels is not constrained.

Procedures

To zoom the view with a single click:

To use this function with the Full Navigation wheel, you must enable the Incremental Zoom-in option on page 7841.

1 Display a wheel that has the Zoom tool.

2 Click the Zoom wedge.

   The magnification of the model is increased and you are zoomed in closer to the model. If you hold down the Shift key while clicking the Zoom wedge, the model is zoomed out or you can hold down the Ctrl key to zoom in.

3 Click Close to exit the wheel.

To zoom a view in and out by dragging:

1 Display a wheel that has the Zoom tool.

2 Click the Zoom wedge. Hold down the button on your pointing device and drag vertically to zoom in and out.

3 Release the button on your pointing device to return to the wheel.

4 To exit the wheel, right-click.

Using Walkthrough Navigation

Walkthrough navigation lets you move through a viewport by pressing a set of shortcut keys, including the arrow keys, much as you can navigate a 3D world in many video games.

When you enter the walkthrough navigation mode, the cursor changes to a hollow circle that shows a directional arrow while you are pressing one of the directional keys (forward, back, left, or right).

This feature is available for perspective and camera viewports. It is not available for orthographic views or for spotlight viewports.

Animating a Walkthrough

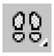
When you use walkthrough navigation in a Camera viewport, you can animate the camera walkthrough using either Auto Key on page 7549 or Set Key on page 3086. In either case, to get an animated camera you have to change the frame number manually (the easiest way is to use the Time Slider on page 7528), and in the case of Set Key, you have to change the frame number *and* click Set Keys.

TIP Select the camera before you animate it. If the camera isn't selected, its keys won't appear in the Track Bar on page 7531.

Procedures

To begin using walkthrough navigation, do one of the following:

1 Press the Up Arrow key.

2  Click the Walk Through button on page 7590 to turn it on.

This button is found on the Pan/Truck And Walkthrough flyout on page 7590.

To stop using walkthrough navigation, do one of the following:

1. Right-click.
2. Activate a different viewport.
3. Change the active viewport to a different type.
4. Turn on a different viewport navigation tool (such as Zoom or Pan).
5. Turn on Select Object or one of the transform tools.

NOTE You *do not* exit walkthrough mode when you select an object or change the viewport shading type (between shaded and wireframe, for example).

Interface

The Walk Through button is the only graphical element of the interface to walkthrough navigation. The other features are provided by mouse actions or by keyboard shortcuts. The following table shows the keyboard actions:

Command	Shortcut
Accelerate Toggle	Q
Back	S, Down Arrow
Decelerate Toggle	Z
Decrease Rotation Sensitivity	
Decrease Step Size	[
Down	C, Shift+Down Arrow
Forward	W, Up Arrow

Command	Shortcut
Increase Rotation Sensitivity	
Increase Step Size	]
Invert Vertical Rotation Toggle	
Left	A, Left Arrow
Level	Shift+Spacebar
Lock Horizontal Rotation	
Lock Vertical Rotation	Spacebar
Reset Step Size	Alt+[
Right	D, Right Arrow
Up	E, Shift+Up Arrow

If nothing appears in the Shortcut column, no default key is assigned to this command. You can set custom keystrokes using the Keyboard panel on page 7698 of the Customize User Interface dialog.

Forward, Backward, and Sideways Movement

For movement, you can use either the arrow keys, or letters at the left of the keyboard pad.

TIP When you are in a Perspective viewport, you can use Undo View Change and Redo View Change (Shift+Z, Shift+Y) to undo or redo your navigation. However, when you are in a Camera viewport, walkthrough animation transforms the camera object, so you must use Edit > Undo and Edit > Redo (Ctrl+Z and Ctrl+Y).

Holding down any of these keys causes the motion to be continuous.

Forward W or the Up Arrow. Moves the camera or the viewpoint forward.

NOTE If you are not already in walkthrough navigation mode, pressing Up Arrow enters it.

Back S or Down Arrow. Moves the camera or the viewpoint backward.

When you are in a camera viewport, Forward and Back are equivalent to dollying in or out.

Left A or Left Arrow. Moves the camera or the viewpoint to the left.

Right D or Right Arrow. Moves the camera or the viewpoint to the right.

When you are in a camera viewport, Left and Right are equivalent to trucking left or right.

Up E or Shift+Up Arrow. Moves the camera or the viewpoint up.

Down C or Shift+Down Arrow. Moves the camera or the viewpoint down.

Acceleration and Deceleration

Accelerate Toggle and Decelerate Toggle Pressing Accelerate (Q) causes motion to be quicker. Pressing Decelerate (Z) causes movement to be slower. These controls are toggles: pressing the key a second time restores the default motion rate (and pressing the alternate key turns off the first). They are especially useful when you are navigating by holding down keys.

The acceleration and deceleration toggles are independent of the step size.

Adjusting Step Size

Increase Step Size and Decrease Step Size Pressing Increase Step Size (]) increases the motion increments when you move the camera or viewpoint. Pressing Decrease Step Size ([) reduces them. You can press either of these shortcuts repeatedly, to increase the effect. Changing the step size is apparent when you navigate either by single clicks, or by holding down keys. Step size changes are useful for adjusting movement to the scale of the scene. They are saved with the MAX file.

Reset Step Size Pressing Reset Step Size (Alt+[) restores the step size to its default value.

The step size is independent of acceleration or deceleration.

Rotation (Tilting)

Tilt View Click+drag to tilt the camera or viewpoint.

When you are in a camera viewport, Tilt View is equivalent to panning the camera.

Increase Rotation Sensitivity and Decrease Rotation Sensitivity Pressing Increase Rotation Sensitivity (no default key) increases the motion increments when you use Tilt View. Pressing Decrease Rotation Sensitivity (no default key) decreases them. You can press either of these shortcuts repeatedly, to increase the effect. They are useful for adjusting movement to the scale of the scene. They are saved with the MAX file.

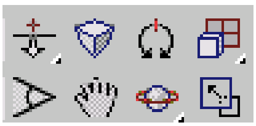
Lock Horizontal Rotation Pressing Lock Horizontal Rotation (no default key) locks the horizontal axis, so the camera or viewpoint tilts only vertically.

Lock Vertical Rotation Pressing Lock Vertical Rotation (Spacebar) locks the vertical axis, so the camera or viewpoint tilts only horizontally.

Invert Vertical Rotation Toggle Pressing Invert Vertical Rotation (no default key) inverts the tilt direction when you drag the mouse. When this toggle is off, dragging up causes scene objects to descend in the view, and dragging down causes them to rise (this is like tilting a physical camera). When this toggle is on, objects in the view move in the same direction you are dragging the mouse.

Level Pressing Level (Shift+Spacebar) removes any tilt or roll the camera or viewpoint might have, making the view both level and vertical.

Navigating Camera and Light Views



The camera navigation buttons

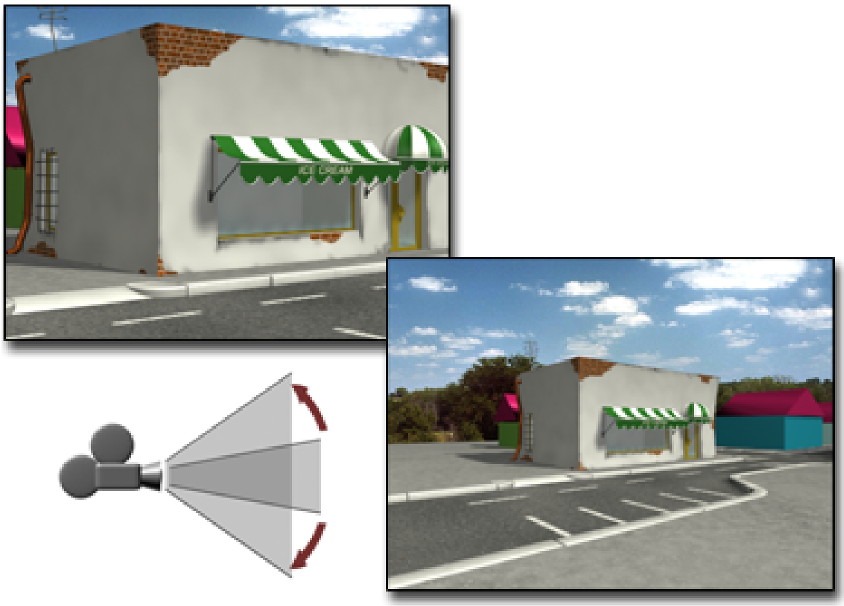
The Camera and Light view navigation buttons are the same with a few exceptions. The buttons are visible when a viewport with a Camera or Light view is active. The Camera and Light view navigation buttons do more than adjust your view. They transform and change the parameters of the associated camera or light object.

Light views treat the light (spotlight or directional light) as if it were a camera. The light falloff is treated the same as the camera field of view.

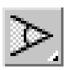
Keep in mind the following:

- Using the Camera and Light viewport navigation buttons is the same as moving or rotating the camera or Light, or changing their base parameters.
- Changes made with Camera or Light view navigation buttons can be animated the same as other object changes.

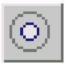
Zooming a Camera or Light View



Zooming a camera

 You zoom a camera view by clicking FOV on page 7596 and then dragging in the Camera viewport.

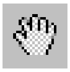
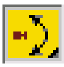
The field of view defines the width of your view as an angle with its apex at eye level and the ends at the sides of the view. The effect of changing FOV is exactly like changing the lens on a camera. As the FOV gets larger you see more of your scene and the perspective becomes distorted, similar to using a wide-angle lens. As the FOV gets smaller you see less of your scene and the perspective flattens, similar to using a telephoto lens. See Cameras on page 5194.

 Click Light Hotspot on page 7618 for a light viewport to achieve the same effect as zooming.

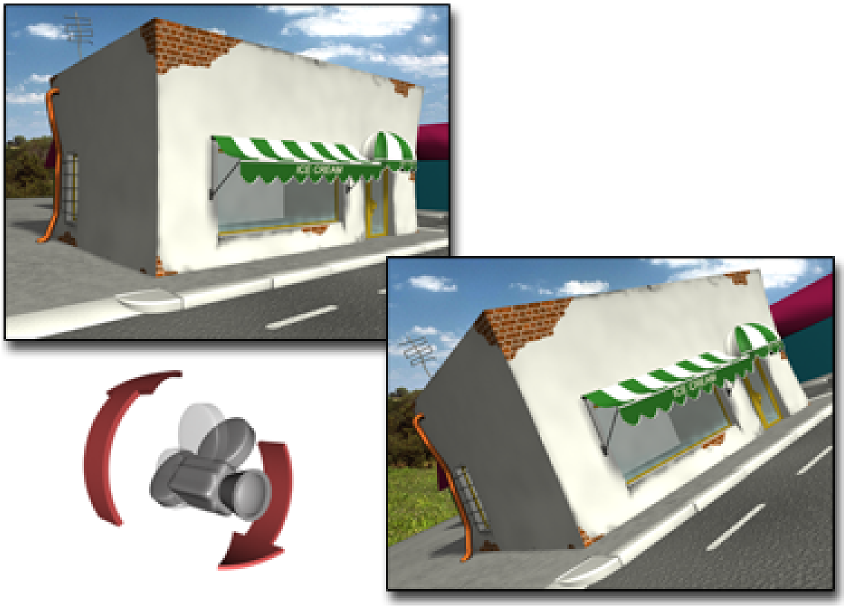
The hotspot is the inner of the two circles or rectangles visible in a light viewport. Objects inside the hotspot are illuminated with the full intensity of the light. Objects between the hotspot and falloff are illuminated with decreasing intensity as objects approach the falloff boundary. See Using Lights on page 4974.

Moving a Camera or Light View

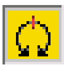
You move a camera or light view by clicking one of the following buttons and dragging in the camera or light viewport.

-  Dolly on page 7605 moves the camera or light along its line of sight.
-  Truck on page 7610 moves the camera or light and its target parallel to the view plane.
-  Pan on page 7611 moves the target in a circle around the camera or light. This button is a flyout that shares the same location with Orbit.
-  Orbit on page 7611 moves the camera or light in a circle around the target. The effect is similar to Orbit for non-camera viewports.

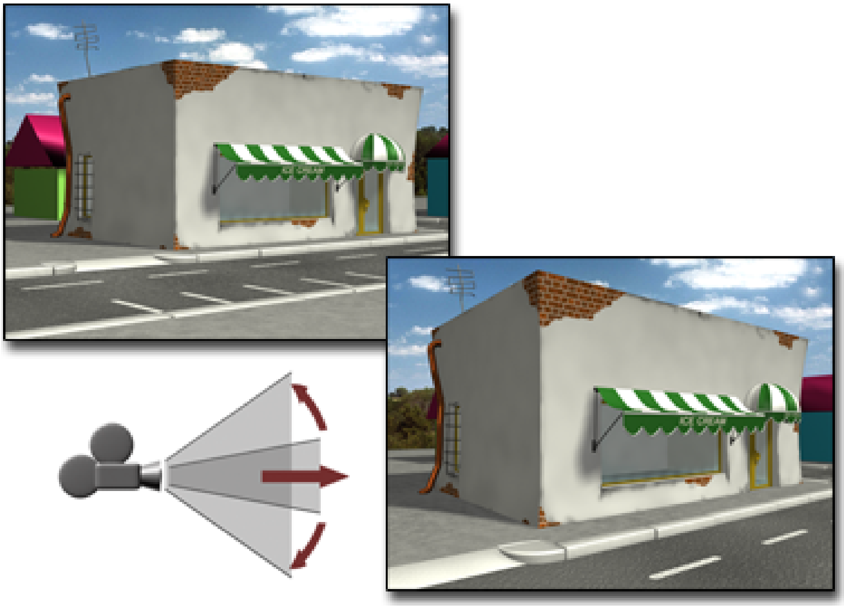
Rolling a Camera or Light View



Rolling a camera

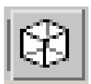
 Click Roll on page 7608, and drag in a camera or a light viewport to rotate the camera or light about its line of sight. The line of sight is defined as the line drawn from the camera or light to its target. The line of sight is also the same as the camera's or the light's local Z axis.

Changing Camera Perspective



Changing perspective

 Click Perspective on page 7607, and drag in a camera viewport to change the Field of View (FOV) and dolly the camera simultaneously. The effect is to change the amount of perspective flare while maintaining the composition of the view.

Adaptive Degradation 

Status bar > Adaptive Degradation button

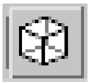
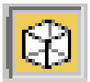
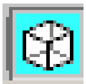
Views menu > Adaptive Degradation

Keyboard > O (the letter O)

Adaptive degradation can improve viewport performance when you transform geometry, change the view, or play back an animation. It does so by decreasing the visual fidelity of certain objects temporarily; for example, by drawing larger objects or those closer to the camera as bounding boxes instead of wireframes. Without adaptive degradation, the geometry is displayed as usual, even if that slows down viewport display and animation playback. Animation playback might drop frames if the graphics card cannot display the animation in real time.

Turn on adaptive degradation if you have large models you need to navigate around and if you are finding performance sluggish.

The Adaptive Degradation button on the status bar has three states:

-  **Off**: No degradation occurs.
-  **On**: Degradation can occur under the specified conditions. This is the default setting.
-  **Active**: Degradation is being applied currently.

You can change the display options and set other adaptive degradation parameters, on the Viewport Configuration dialog (Customize menu > Viewport Configuration > Adaptive Degradation panel on page 7828). Also, you can toggle adaptive degradation for individual objects with the Object Properties > Never Degrade setting on page 312.

Procedures

To toggle adaptive degradation, do one of the following:

- Choose Views menu > Adaptive Degradation.
- Press O (the letter O).

To change the level of adaptive degradation in the viewports:

1. Right-click the viewport label and choose Configure, or choose Customize > Viewport Configuration.
2. On the Viewport Configuration dialog, open the Adaptive Degradation panel and adjust the settings.

Grab Viewport

Tools menu > Grab Viewport

Grab Viewport creates a snapshot of the active viewport in the Rendered Frame Window on page 6073, where you can save it as an Image file on page 7324.

Procedures

To create a snapshot of a viewport:

1. Activate the viewport you want to capture.
2. Choose Tools menu > Grab Viewport.

   A dialog appears that allows you to add a label to your snapshot.
3. Enter a label for your snapshot, if desired.

   The label appears in the lower-right corner of the image as you enter it into the dialog.
4. Click Grab.

   The Rendered Frame Window opens to display a snapshot of your viewport.
5. Use the controls in the Rendered Frame Window to save your image.

Interface

Label Enter text here to add a label to your screenshot. The text you enter is displayed in the lower-right corner of your screenshot.

Grab Opens the Rendered Frame Window with a snapshot of the active viewport.

Cancel Cancels the Grab Viewport command.

View-Handling Commands

Most viewport-handling commands are found on the Views menu on page 7477 and on the Viewport Right-Click menu on page 7576.

See also:

- Viewing and Navigating 3D Space on page 87
- Quad Menu on page 7516

Undo View Change/Redo View Change

Views menu > Undo View Change or Redo View Change

Keyboard > Shift+Z (Undo) or Shift+Y (Redo)

Undo View Change cancels the last change made to the current viewport. Redo View Change cancels the last Undo in the current viewport.

These commands act like Undo and Redo on the main toolbar and Edit menu, but operate on a different list of events. They affect changes made *to* the viewport, rather than changes made to *objects in* the viewport.

Use Undo View Change and Redo View Change when you have inadvertently made a view unusable by zooming in too close, or rotating the wrong way. You can keep stepping back until a useful view appears. The keyboard shortcuts are handy for multiple commands.

You can also access Undo View Change and Redo View Change of view changes by right-clicking the viewport label and choosing Undo View or Redo View. The last change made in that viewport will be indicated (for example, "Undo View Zoom"). Each viewport has its own independent undo/redo stack.

Camera and Spotlight viewports use object-based Undo and Redo, because the viewport change is actually a change to the camera or spotlight object. In these viewports, use Edit > Undo (Ctrl+Z) or Edit > Redo (Ctrl+Y).

Interface

Undo View Change Cancels viewport changes. The name of the change you're undoing is displayed in the View menu beside the command.

Undo is useful when you are working with a background image in the viewport. You can zoom into the geometry to adjust it, then use Undo Viewport Zoom to restore the original alignment of the geometry with the background.

Redo View Change Cancels the previous Undo View Change. The name of the change you're redoing appears in the View menu beside the command.

Save Active View

Views menu > Save Active View (the name of the active viewport is part of the command)

Save Active View stores the active view to an internal buffer. If you have framed a shot in any view other than a camera, use Save Active View to preserve the viewport's appearance. The saved active view is saved with the scene file. Once saved, you can retrieve it using Restore Active View on page 147.

The viewport that will be restored is displayed in the menu item (for example, "Save Active Perspective View"). You can save and restore up to eight different views (Top, Bottom, Left, Right, Front, Back, Orthographic, Perspective).

Viewport changes that are saved include viewport type, zoom and rotations, and field-of-view (FOV).

The options available on the viewport right-click menu on page 7576, such as Show Safe Frame and Viewport Clipping, are not saved. If these settings are important to the view, make a note of what they are so you can reset them after restoring the view.

Procedures

To save an active view:

1. Activate the viewport with the view you want to save.
2. Choose Views menu > Save Active View. The view is now saved and can be recalled using Restore Active View.

Restore Active View

Views menu > Restore Active View (the name of the active viewport is part of the command.)

Restore Active View displays the view previously stored with Save Active View on page 147.

The viewport to be restored is displayed in the menu item (for example, "Restore Active Perspective View").

The active view is restored if the same viewport and layout are active.

If an active view won't restore with this command, check the following:

- Be sure the viewport is active.
- Make sure the layout is the same as before. Use Viewport Configuration (right-click any viewport label and choose Configure) and choose Layout.
- If the layout and active viewport are the same, be sure Viewport Clipping on the Viewport Right-Click Menu on page 7576 is set the same as it was when the viewport was saved.

Procedures

To restore a saved view:

1. Activate the viewport where you saved the view.
2. Choose Views menu > Restore Active View. This option is available only in a viewport with a saved view.
3. The viewport returns to the saved view.

   If you're not sure whether a viewport has a saved view, check the Views menu. Restore Active View is unavailable unless a view is saved in the active viewport.

Viewport Background Dialog

Views menu > Viewport Background > Viewport Background dialog

Keyboard > Alt+B

The Viewport Background dialog controls display of an image or animation as the background for one or all viewports.You can use this for modeling, for example, by placing front, top or side view sketches in the corresponding viewports. Or use Viewport background to match 3D elements with digitized camera footage, or for rotoscoping on page 8109.

You select the image or animation to display in the active viewport, set the frame synchronization between the animated image file and the current scene, and turn the assigned image on and off. These changes do not affect the rendered scene.

To place an image in the background of the rendered scene, use the Environment And Effects dialog > Environment panel on page 6687, accessed from the Rendering menu.

NOTE When safe frames are displayed in a viewport, and the Aspect Ratio options are set to either Match Viewport or Match Rendering Output, the assigned viewport background image is confined to the Live area of the safe frames and will correctly match the rendered background bitmap.

TIP If you are using a viewport driver with hardware acceleration (OpenGL or Direct3D), the viewport background might not appear. If this happens, choose Customize > Preferences. In the Viewports preferences on page 7753, click Configure Driver. Then in the Configure OpenGL dialog on page 7796 or the Configure Direct3D dialog on page 7802, go to the Background Texture Size group and turn on Match Bitmap Size As Closely As Possible (*do not change* the numeric setting). Click OK in both dialogs to accept your change.

See also:

- Select Background Image Dialog on page 156
- Update Background Image on page 161
- Reset Background Transform on page 162

Procedures

To assign an image to one or all viewports:

1. Activate the viewport that is to display the background image.
2. Choose Views menu > Viewport Background or press Alt+B.

   This opens the Viewport Background dialog.
3. In the Background Source group, click the Files button.

   This opens the Select Background Image dialog.
4. Use the dialog to open the image or animation to use.

5 To display the image in all viewports, choose All Views in the Apply Source And Display To group.

6 Click OK.

The image is displayed in a single viewport or all viewports.

To update the image or map in the viewport:

Because of the time it takes to render the image or map in the viewport, the map is not automatically updated when you alter the bitmap or assign a new bitmap.

- Choose Views menu > Update Background Image.
  The revised image or map is displayed in the viewport.

To display the environment map in a viewport:

1 In the Environment dialog, assign an environment map. (See the procedure "To choose an environment map." on page 6689)

2 In the Environment dialog > Background group, be sure Use Map is turned on (the default).

3 Activate the viewport where you want the map displayed.

4 Choose Views menu > Background Image.

5 In the Viewport Background dialog > Background Source group, turn on Use Environment Background.

6 Click OK.

The map is displayed in the viewport.

To display an animated background:

1 Assign an animation file (AVI, MOV, or IFL file) as the viewport background.

2 Turn on Animate Background.

3 Choose Customize > Preferences. On the Viewports panel, turn on Update Background While Playing.

Now the background plays when you click Play, or when you drag the time slider.

TIP If you follow these steps and the background still doesn't appear to animate, open the Time Configuration dialog on page 7565 and in the Playback group, turn off Real Time.

To use the environment map with animation controls:

This procedure is useful if you've assigned an animated environment map and want access to the animation controls on the Viewport Background dialog.

1. In the Viewport Background dialog > Background Source group, turn off Use Environment Background.
2. In the same group, click File.
3. Choose the same map you're using as the environment map.
4. Set parameters in the Animation Synchronization group.
5. Click OK.

   The environment map appears in the viewport. The image is renderable.

To match the viewport background with the rendered background:

1. Activate the viewport to render.
2. Right-click the viewport label and choose Show Safe Frame.

   This turns on Safe Frames on page 7825 in the viewport.

NOTE You can also use Views menu > Configure > Safe Frame tab. In the Application group, turn on Show Safe Frames In Active View.

---

3. In the Material Editor, create a material that contains the bitmap for your rendered background.
4. At the bitmap level of the Material Editor, on the Coordinates rollout, choose Environ.

   The Mapping control is automatically set to Screen. This is the only mapping type that works for this purpose.
5. On the main menu, choose Rendering > Environment.

6 Drag the map from the Material Editor > Maps rollout to the Environment Map button on the Environment dialog. Click OK on the Instance (Copy) Map dialog.

7 In the Viewport Background dialog > Background Source group, click Files to assign the same bitmap.

8 In the Aspect Ratio group, turn on either Match Viewport or Match Rendering Output. Click OK.

9 Render the viewport.

   The background displayed in the rendered scene should exactly match the background displayed in the Live area of the safe frames.

   NOTE When you use the Match Bitmap option, the bitmap reverts to its original aspect ratio and does not match the rendered scene, unless you're rendering to the same aspect ratio.

To remove a background image:

1 Activate the viewport in which the background image is visible.

2 On the Views menu, choose Viewport Background.

   Notice the name and path of the background file is displayed in the Current field in the Background Source group

3 In the Background Source group, click Devices.

4 On the Select Image Input Device dialog, choose No I/O Handlers from the drop-down list, then click OK.

5 On the Bitmap Manager Error dialog, click OK.

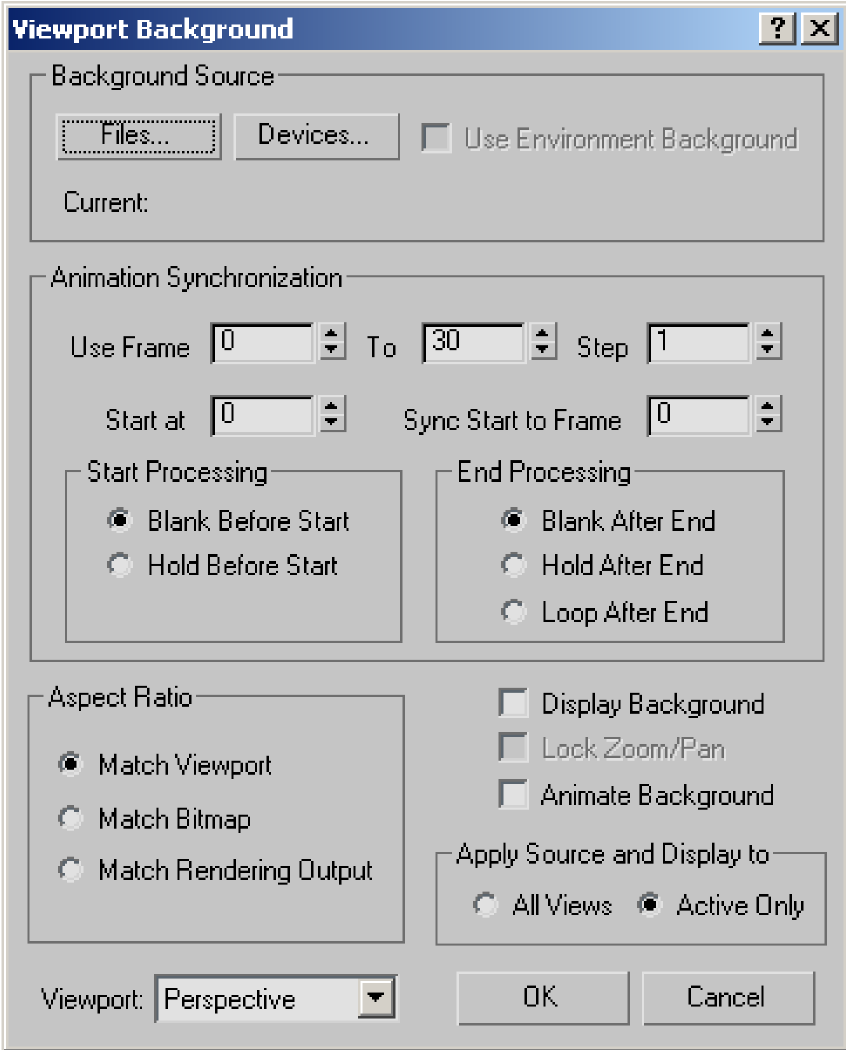
   The current field no longer displays the background file name. Instead No I/O Handler is listed in the Current field.

6 Click OK to close the Viewport Background dialog.

   Next time you open up the Viewport Background dialog, no file name will be displayed in the Current field.

   TIP This technique will work only on systems that don't have any other Image Input Devices installed.

Interface



Background Source group

Options let you select the background image, either from a bitmap image file on page 7926, a video file, or from a device such as a video recorder.

Files Displays the Select Background Image dialog on page 156, which lets you select a file or sequence of files for your background.

Devices Displays the Select Image Input Device dialog. This lets you use a background from a digital device. (No device is supported by the default 3ds Max installation.)

Use Environment Background Lets you display in the viewports the map you've assigned as your environment background. If no environment map has been assigned in the Environment dialog, or Use Map in that dialog is off, then the Use Environment Background check box is not available.

Animation Synchronization group

Controls how sequences of images (for example, from IFL on page 7339, AVI on page 7326, or MOV on page 7348 files) are synchronized to the viewport for rotoscoping on page 8109.

Use Frame The first field sets the first frame of the incoming sequence that you want to use, and the second field sets the last one.

Step Sets the interval between the frames you want to use. For example, if this spinner is set to 7, 3ds Max uses every seventh frame.

Start At Specifies the frame number at which you want the first input frame to appear. What happens in the viewport before the start frame depends on the option you choose for "Start Processing," below.

Sync Start To Frame Determines which frame from your incoming sequence is displayed at the Start At frame. For example, you could have a 30-frame IFL sequence that starts in your scene at frame 10, but you could use the 5th frame from the IFL on frame 10 by setting Sync Start to 5.

Start Processing group

Determines what happens in the viewport background before the start frame.

Blank Before Start Makes the viewport background blank before the start frame.

Hold Before Start Specifies that the viewport background will contain the start frame.

End Processing group

Determines what happens in the viewport background after the last input frame.

Blank After End Makes the viewport background blank after the last input frame.

Hold After End Specifies that the viewport background will contain the last input frame until the last frame in the animation.

Loop After End Specifies that the viewport background will loop from the end frame back to the start frame, ad infinitum.

Aspect Ratio group

Controls the proportions of the viewport background by matching it to the bitmap, rendering output, or to the viewport itself.

Match Viewport Changes the aspect ratio on page 7914 of the image to match the aspect ratio of the viewport.

Match Bitmap Locks the aspect ratio of the image to the native aspect ratio of the bitmap.

Match Rendering Output Changes the aspect ratio of the image to match the aspect ratio of the currently chosen rendering output device.

NOTE When the Match Bitmap or Match Rendering Output option is chosen, 3ds Max centers the image and clears the edges of the viewport to the background color.

Display Background

Turns on display of the background image or animation in the viewport.

Lock Zoom/Pan

Locks the background to the geometry during zoom and pan operations in orthographic or user viewports. When you Zoom or Pan the viewport, the background zooms and pans along with it. When Lock Zoom/Pan is turned off, the background stays where it is, and the geometry moves independently of it. Use Match Bitmap or Match Rendering Output to enable Lock Zoom/Pan. This control is disabled if you choose Match Viewport.

Keyboard shortcut: Ctrl+Alt+B

WARNING If you zoom in too far, you can exceed the limit of virtual memory, and crash 3ds Max. When you perform a zoom that requires more than 16 megabytes of virtual memory, an alert asks if you want to display the background during the zoom. Choose No to perform the zoom and turn off the background. Choose Yes to zoom with the background image. Your machine might run out of memory as a result.

Animate Background

Turns on animation of the background. Shows the appropriate frame of the background video in the scene.

Apply Source And Display To group

All Views Assigns the background image to all viewports.

Active Only Assigns the background image to only the active viewport.

Viewport

The name of the currently active viewport appears in a list to the left of the OK and Cancel buttons. This reminds you which viewport you're working with and lets you change the active viewport by selecting its name from the list.

NOTE When you use different images for different viewports, the settings for each viewport are stored separately. Each time you display the Viewport Background dialog, the settings of the currently active viewport are displayed. If you switch the viewport using the list, the settings remain the same. This is useful for copying settings from one viewport to another.

Select Background Image Dialog

Views menu > Viewport Background > Background Source group > Files > Select Background Image dialog

UFO model rendered against a background

The Select Background Image dialog allows you to choose a file or sequence of files for a viewport background.

You can also convert a set of sequentially numbered files to an Image File List (IFL) on page 7339. This is the same process used by the IFL Manager Utility on page 7344.

Procedures

To select a background image for a viewport:

1. Activate the viewport where you want the image.
2. Choose Views menu > Viewport Background.
3. Under Background Source in the dialog that displays, click Files.
4. In the Look In field, navigate to the directory containing the file you want to use for the background.

   NOTE The Select Background Image File dialog uses the last location where a bitmap was chosen, rather than the default bitmap path defined on the Configure User Paths dialog on page 7729.

5 Highlight the file name in the file list window.

6 Click Open to select the image and close the dialog.

7 Click OK to close the Viewport Background dialog and display the background image.

To select a set of still images as a viewport background:

1 Activate the viewport where you want the image.

2 Choose Views menu > Viewport Background.

3 Under Background Source, click Files.

4 In the Look In field, navigate to the directory containing the sequence of files.

The files must be sequentially numbered (for example, *image01.bmp, image02.bmp, image03.bmp*).

TIP If necessary, change Files Of Type to match the file extension of the sequence, or choose All Formats.

---

5 Turn on Sequence, and choose the name of the first sequential file (for example, image01.bmp).

TIP Click the Setup button to display the Image File List Control dialog on page 7342.

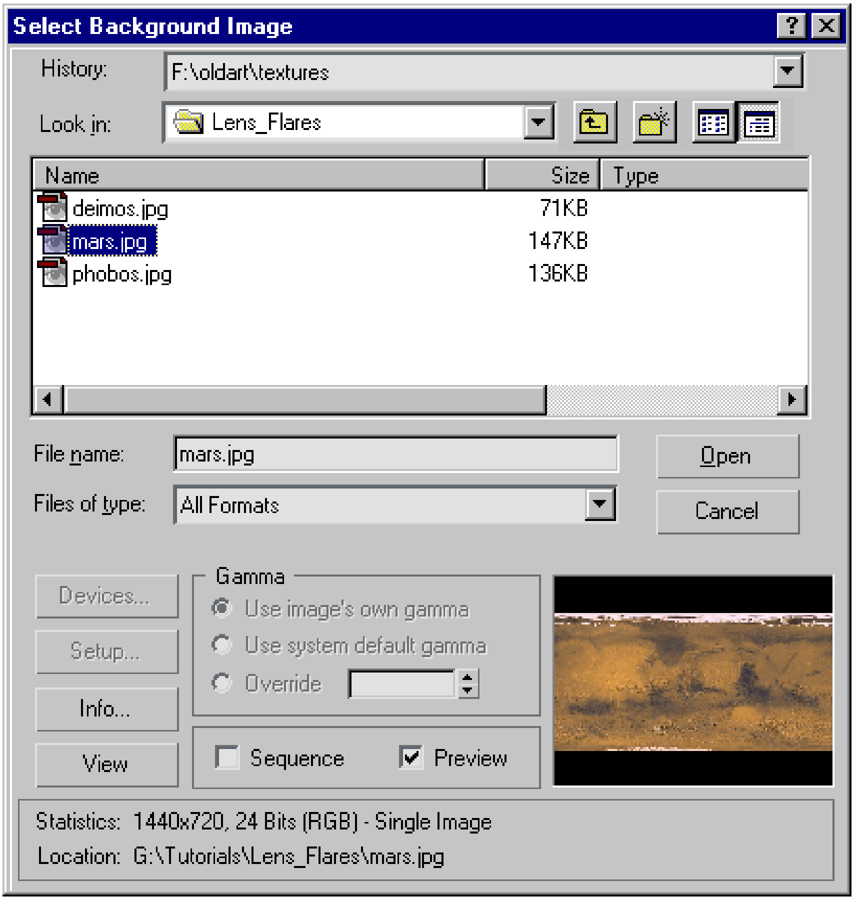
---

6 In the Image File List Control dialog, use the Browse button to set the Target Path to a directory on your hard disk. Do not set this path to a CD-ROM drive, because you cannot save the file there.

7 Choose the options you want, and then click OK.

The Image File List (IFL) file is saved to the target directory.

Interface

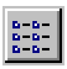


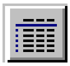
History Displays a list of the directories most recently searched.

Look In Opens a navigation window to move to other directories or drives.

 **Up One Level** Move up a level in the directory structure.

 **Create New Folder** Lets you create a new folder while in this dialog.

 **List** Displays the contents of a directory by file name.

 **Details** Displays the contents of a directory including all the file details.

List Window When Details is on, the contents of the directory are displayed with Name, Size, Type, Date Modified, and Attributes. You can sort the files by clicking the label of each parameter.

File Name Displays the name of the file selected in the list.

Files of Type Displays all the file types that can be displayed. This serves as a filter for the list.

Open Selects the highlighted file and closes the dialog.

Cancel Cancels the selection and closes the dialog.

Devices Lets you select a background image from a digital device. (No device is supported by the default 3ds Max installation.)

Setup Displays the Image File List Control dialog on page 7342 to create an IFL file. Available only when Sequence is on and there are sequentially numbered files in the displayed directory.

Info Displays expanded information about the file, such as frame rate, compression quality, file size, and resolution. The information here is dependent on the type of information that is saved with the file type.

View Displays the file at its actual resolution. If the file is a movie, the Media Player is opened to play the file.

Gamma group

Selects the type of gamma to be used for the selected file. Available only when Enable Gamma Selection is turned on in the Gamma panel on page 7758.

Use Image's Own Gamma Uses the gamma of the incoming bitmap.

Use System Default Gamma Ignores the image's own gamma and uses the system default gamma instead, as set in the Gamma panel on page 7758.

Override Defines a new gamma for the bitmap that is neither the image's own, nor the system default.

Sequence Creates an "Image File List" to your specifications. Each selected image is checked to see if a valid IFL sequence can be created. If the selected image doesn't yield a list, this option is still available, but doesn't do anything.

Preview Displays the image as a thumbnail in the Image Window.

Image Window Displays a thumbnail of the selected file if Preview is on.

Statistics Displays the resolution, color depth, file type and number of frames of the selected file.

Location Displays the full path for the file. With this information at the bottom of the dialog, you always know exactly where you are.

Update Background Image

Views menu > Update Background Image (available only when a viewport background is displayed)

This command updates the background image displayed in the active viewport. If the active viewport is not displaying a background image, this command is unavailable.

Use this command to update the background for changes that are not updated automatically, such as the following:

- Reassigning the map, or changing any parameters affecting the map in the Materials Editor, the Environment dialog, or the Viewport Background dialog.
- Changing the rendering resolution and aspect ratio.
  The following changes update the viewport background image automatically:
- Changing the camera view.
- Undo (for views).
- Undo (for objects).
- Assigning a different view type.
- Toggling Safe Frames display on or off.
- Changing the rendering parameters.
- Moving the time slider when the viewport contains an animated background image.

NOTE Viewports can use the current Environment Map (set on the Environment panel on page 6689 of the Environment and Effects dialog) as the background image.

Procedures

To update the background image displayed in a viewport:

1. Activate a viewport that contains a background image.
2. Choose Views menu > Update Background Image.

Reset Background Transform

Views menu > Reset Background Transform (available only when a viewport background image is displayed and Lock Zoom/Pan is turned on)

Reset Background Transform rescales and recenters the current background to fit an orthographic or user viewport. Use this command when you want to reset the background to the new position of your geometry. See Procedure for detailed requirements.

Procedures

To reset the background to fit the viewport:

1. Activate an orthographic or user viewport that has a background image.
2. Press Alt+B.
3. Turn on either Match Bitmap or Match Rendering Output, and then turn on Lock Zoom/Pan.
4. Click OK.
5. Choose Views menu > Reset Background Transform.

   The background image readjusts in the viewport.

Show Transform Gizmo

Views menu > Show Transform Gizmo

Keyboard > X

Show Transform Gizmo toggles the display of the Transform gizmo axis tripod on page 950 for all viewports when objects are selected and a transform is active.

Additional controls for the Transform gizmo are found on the Gizmo Preferences settings on page 7778.

When the Transform gizmo is turned off, Show Transform Gizmo controls the display of the axis tripod on selected objects.

The state of Transform gizmo is saved in *3dsmax.ini*, so it's maintained between scenes and sessions.

The related entries in the *3dsmax.ini* file are:

- INI: Transformgizmo=1 (for Transform Gizmo visibility, controlled by Preferences)
- INI: ShowAxisIcon=1 (for Axis Icon visibility, controlled in Views menu)

The visibility of the Axis tripod overrides the visibility of the Transform Gizmo. If you turn off the Transform Gizmo in Preferences, the Axis tripod remains on the selected object. If you then turn off the Show Transform Gizmo in the Views menu, it actually turns off the Axis tripod. When the Axis tripod is disabled, the Transform Gizmo is also hidden.

TIP The converse is not true. If the transform gizmo is turned off, turning on the axis tripod visibility does not display the transform gizmo.

Procedures

To scale the transform gizmo, do one of the following:

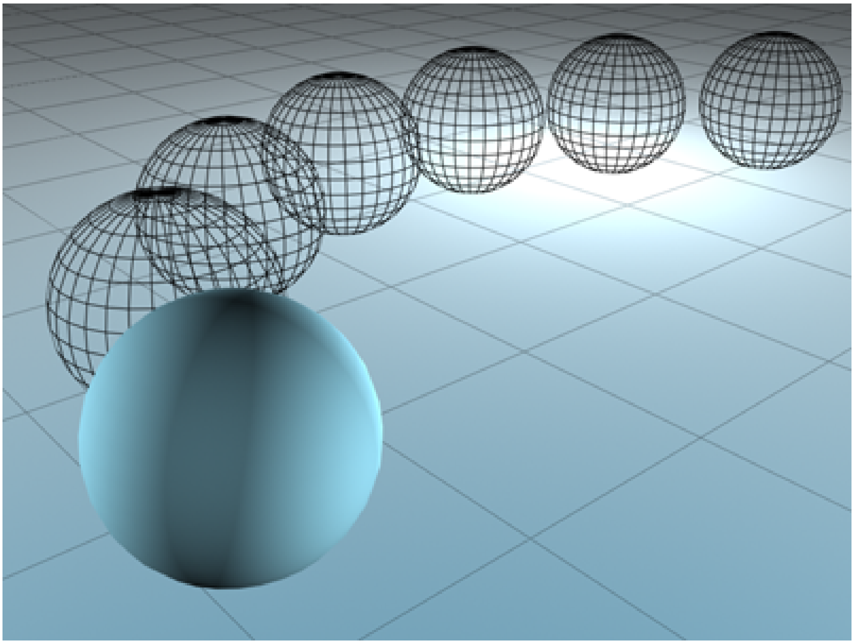
1. Press – (hyphen) to shrink the Transform gizmo.
2. Press = (equal sign) to enlarge the Transform gizmo.

Show Ghosting

Views menu > Show Ghosting

Ghosting is a method of displaying wireframe "ghost copies" of an animated object at a number of frames before or after the current frame. Use it to analyze and adjust your animation. Ghosts that overlap indicate slower motion; ghosts that are spread further apart show faster motion.

When this command is active, ghosting is displayed for selected objects in the scene. Only currently selected objects display the ghosting.



Ghosting helps to visualize animation.

To change Ghosting parameters choose Customize > Preferences. On the Viewport panel of the Preferences dialog you can determine the number of ghosting frames, whether to ghost before or after the current frame, or both, and you can also show frame numbers with the ghosts.

Procedures

To show wireframe ghost copies of an animated object:

- Choose Views menu > Show Ghosting.

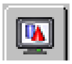
Show Key Times

Select an object with animation. > Views menu > Show Key Times

Key Times shows the frame numbers along a displayed animation trajectory on page 3121. Key times correspond to the settings in Time Configuration on page 7565 for Frames or SMPTE on page 8131. By default, key times are shown as frame numbers.

Procedures

To display trajectory time values in the viewport:

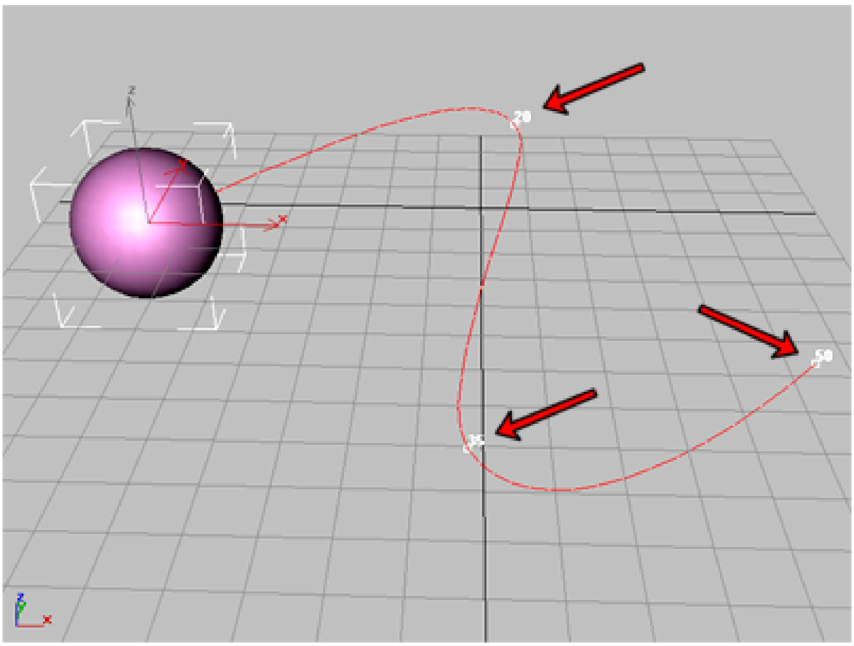
1. Select an object with animation.
2.  On the Display panel > Display Properties rollout, turn on Trajectory.

   TIP If the rollout controls are unavailable, right-click the object in the active viewport, choose Properties, and in the Display Properties group, click By Layer to change to By Object. This will make the Trajectory option become available.

   ---

3. Choose Views menu > Show Key Times.

   The time values are displayed as white numbers along the trajectory. They remain displayed in red when the animated object is deselected.



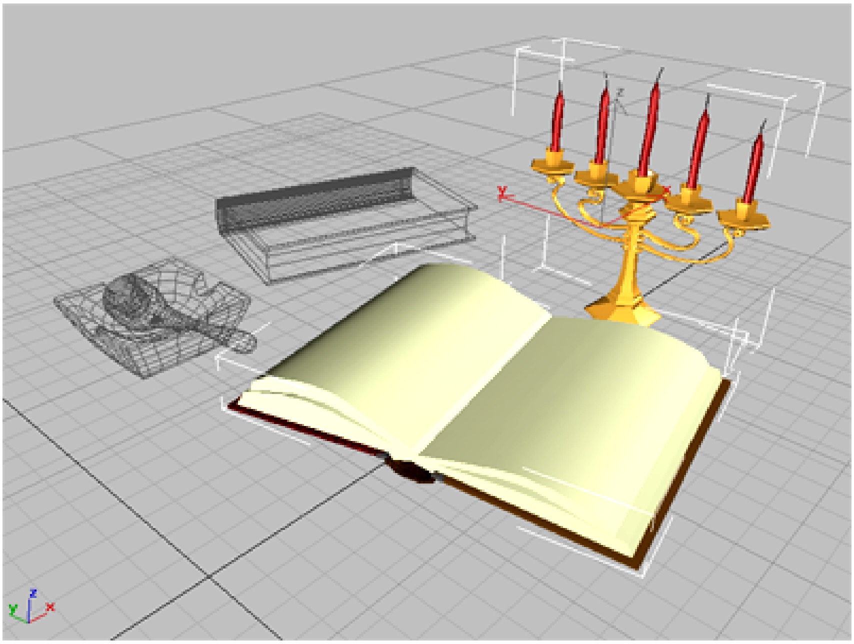
Keyframes with frame number shown on a trajectory.

Shade Selected

Select an object to be shaded. > Views menu > Shade Selected

Shade Selected shades only the selected objects in the scene when the viewport is set to Wireframe or Other. When Smooth + Highlights is on, all objects are shaded whether they are selected or not.

Shade Selected lets you work with a wireframe scene and shade only the selected objects when you want to visualize them more clearly. All other objects in the scene will appear in wireframe.



Selected objects shaded in a wireframe viewport.

Procedures

To shade only selected objects in a scene:

1. Choose Views menu > Shade Selected.
2. Right-click the viewport label and choose Wireframe.
3. Select the object.

Only the selected object is shaded.

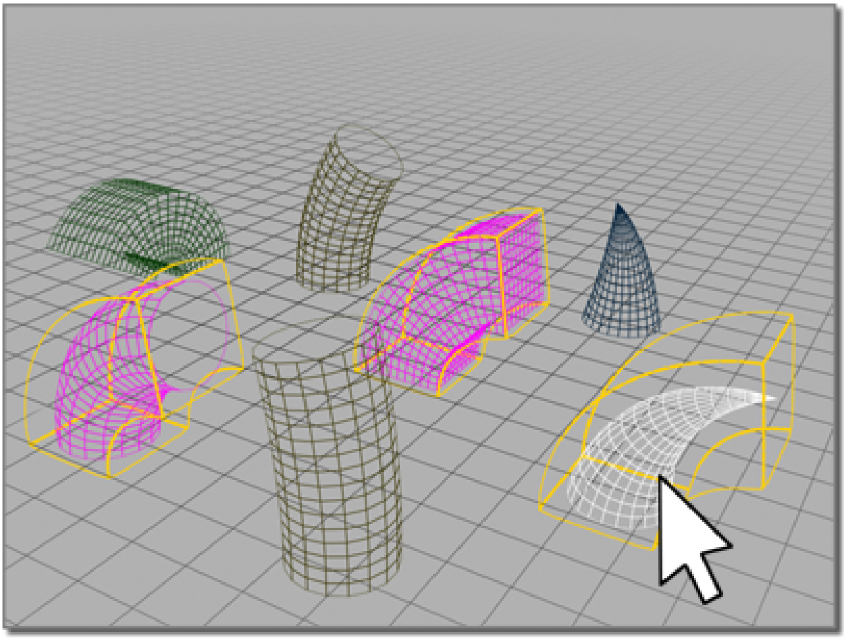
Show Dependencies

Views menu > Show Dependencies

While you are using the Modify panel, this command toggles viewport highlighting of objects dependent on the currently selected object.

When Show Dependencies is on and the Modify panel is active, any object that is dependent upon the currently selected object in any way appears magenta. This includes instances on page 8014, references on page 8106, and shared modifiers on page 8048. Default=off.

You can also see similar dependencies in Schematic View on page 7407.



Procedures

To show dependencies between objects:

1 Select an object with an instanced modifier on page 1110.

2  On the Modify panel, choose the instanced modifier in the modifier stack.

3 Choose Views menu > Show Dependencies

   Other objects with instances of the same modifier appear in a different color.

Example: To use Show Dependencies when animating with Linked XForm:

1 Select the sub-object geometry you want to animate, and apply a Linked XForm modifier on page 1511.

2 On the Parameters rollout, click Pick Control Object.

3 Click an object to be the control object. Choose a dummy object if you want to keep the control hidden in final rendering.

4 The chosen object is now linked as parent to the sub-object selection and its name is listed on the Parameters rollout.

5 Choose Views menu > Show Dependencies to make the link visible when the control object is selected.

6 Use any of the transforms to animate the control object.

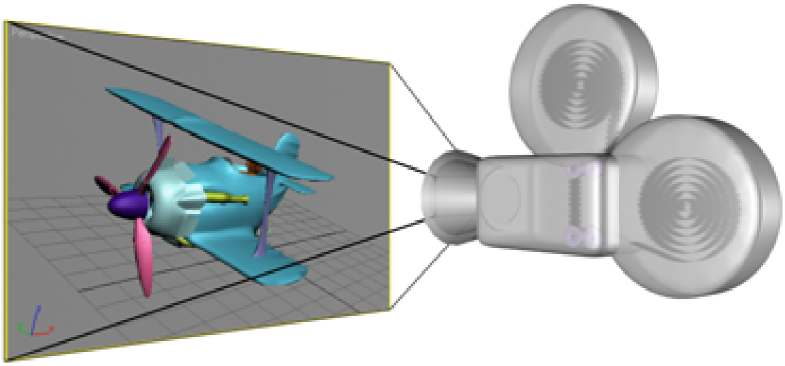
   The selection is animated in parallel with the control object.

Create Camera From View

Activate a Perspective viewport. > Views menu > Create Camera From View

Activate a Perspective viewport. > Create menu > Cameras > Create Camera From View

Activate a Perspective viewport. > Keyboard > Ctrl+C



Create Camera From View creates a Target camera on page 5205 whose field of view matches an active Perspective viewport. At the same time, it changes the viewport to a Camera viewport on page 7604 for the new camera object, and makes the new camera the current selection.

Alternatively, if the scene already contains a camera and the camera is selected, then Create Camera From View does not create a new camera from the view. Instead, it simply matches the selected camera to the active, Perspective viewport. This functionality was adopted from the Match Camera to View command, which is now available only as an assignable main user interface shortcut (see Keyboard Shortcuts on page 7857).

NOTE Create Camera From View is available only when a Perspective viewport is active.

To create a camera from a view, assuming any existing cameras are unselected:

1 Activate a Perspective viewport.

2 If necessary, adjust the viewport using Pan, Zoom, and Orbit, or the ViewCube, until you have a view you like.

3 Leaving the viewport active, on the Views menu choose Create Camera From View or press Ctrl+C.

   3ds Max creates a new camera, matching its view to that of the Perspective viewport, and then switches the Perspective viewport to a Camera viewport, showing the view from the new camera.

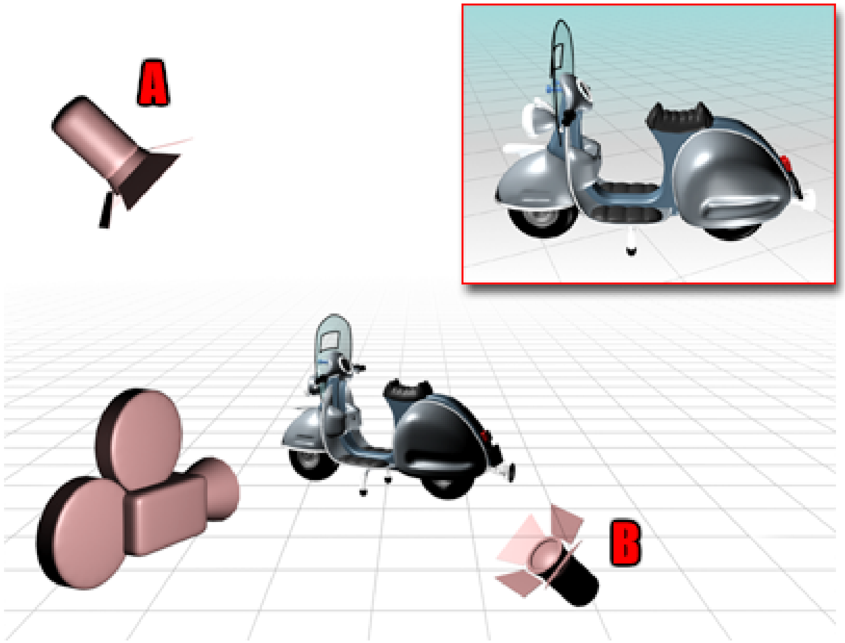
Add Default Lights to Scene

Right-click a viewport label. > Configure > Viewport Configuration dialog > Rendering Method panel > Rendering Options group > Turn on Default Lighting and choose 2 Lights (to activate the Add Default Lights To Scene menu item) > Click OK. > Create menu > Lights > Standard Lights > Add Default Lights To Scene

This command opens the Add Default Lights To Scene dialog, which provides options that let you convert the default scene lighting into actual light objects on page 4970.

The default lighting for viewports consists of a *key* light, positioned in front and to the left of the scene, which behaves as an omni light on page 5063..

This command is unavailable unless you use the Viewport Configuration dialog on page 7817 to configure the active viewport to use two lights. When viewports use two lights, and you invoke this command, the lights are added to the scene as omni lights. You can add either the key light, the fill light, or both.



Two default lights are placed opposite to each other.

A, the key light is in front of the object, on the upper left side, while B, the fill light is behind on the lower right side.

You can add either the key light, the fill light, or both. The omni light objects have the names *DefaultKeyLight* and *DefaultFillLight*.

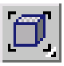
If you have already added one or both default lights, a warning prompts you to rename or delete the previous default light object before you add another.

Procedures

To add the default lights as objects:

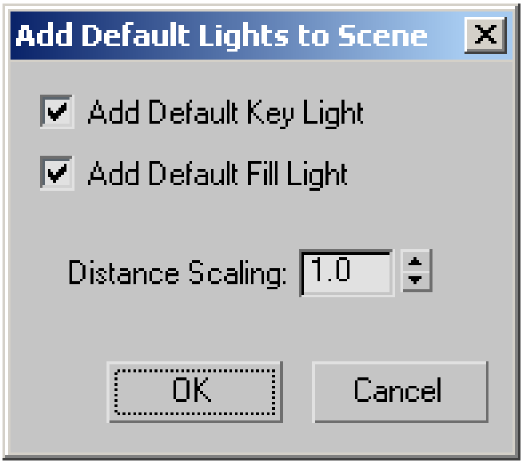
1. Right-click a viewport label, and click Configure.
2. On the Viewport Configuration dialog > Rendering Method tab, in the Rendering Options group, turn on Default Lighting and choose 2 Lights. Click OK to close the dialog.
3. Choose Create menu > Lights > Standard Lights > Add Default Lights To Scene.

4 On the Add Default Lights To Scene dialog, toggle Add Default Key Light, Add Default Fill Light, or both. Click OK.

5  Activate the Top viewport, and on the status bar, click Zoom Extents.

The lights are now visible in the viewport.

Interface



Add Default Key Light When on, adds the default key light to the scene. The key light is in front of the scene and to the left. The key light becomes an omni light on page 5063 with the name, DefaultKeyLight. Default=on.

Add Default Fill Light When on, adds the default fill light to the scene. The fill light is behind the scene and to the right. The fill light becomes an omni light on page 5063 with the name, DefaultFillLight. Default=on.

Distance Scaling Affects how far the lights are placed from the origin (0,0,0). The default value leaves the scene's lighting unchanged. Larger values move the lights farther away, dimming the scene, and smaller values move them closer, brightening the scene. Default=1.0. Range=0.0 to 1000.0.

Redraw All Views

Views menu > Redraw All Views

Keyboard > ` (accent grave)

Redraw All Views refreshes the display in all viewports. When you move, rotate, scale, or otherwise manipulate geometry, the viewports may display the scene with some irregularities, or with objects or parts of objects missing. Use Redraw All Views to redisplay your scene with all lines and shading restored.

Show Materials in Viewport As

Views menu > Show Materials in Viewport As > Standard Display

Views menu > Show Materials in Viewport As > Standard Display with Maps

Views menu > Show Materials in Viewport As > Hardware Display

Views menu > Show Materials in Viewport As > Hardware Display with Maps

Material Editor > Material menu > Show Materials in Viewport As > Standard Display

Material Editor > Material menu > Show Materials in Viewport As > Standard Display with Maps

Material Editor > Material menu > Show Materials in Viewport As > Hardware Display

Material Editor > Material menu > Show Materials in Viewport As > Hardware Display with Maps

These commands turn on and off display of all maps in the viewports, using software or hardware rendering. Software rendering applies to all maps and materials, but tends to be less accurate. Hardware rendering is highly accurate, showing highlights very close to the way they render, but applies primarily to the Arch & Design material on page 5544, although it also supports the Standard material on page 5395.

NOTE Hardware viewport rendering is not available on computers with older graphics systems. Also, hardware viewport rendering is supported only by the Direct3D display driver on page 7802.

NOTE These commands do not apply to XRef materials on page 5765, including materials from XRef objects on page 6936 and XRef scenes on page 6959.

See also:

- Show Standard/Hardware Map in Viewport on page 5350

Interface

Standard Display Uses the legacy software shader, which works on a per-face basis, and turns off viewport display of all maps at the material level. This applies only to materials used in the scene.

Standard Display with Maps Uses the legacy software shader, which works on a per-face basis, and turns on viewport display of all maps.

Hardware Display Uses the hardware viewport shader, which works on a per-pixel basis, and turns off viewport display of all maps at the material level.

The hardware shader applies only to the Standard on page 5395 and Arch & Design on page 5544 materials; when it's active, 3ds Max still uses the software shader to display all other materials.

Hardware Display with Maps Uses the hardware viewport shader, which works on a per-pixel basis, and turns on viewport display of all maps at the material level.

This switch applies only to the Standard on page 5395 and Arch & Design on page 5544 materials; when it's active, 3ds Max still uses the software shader to display all other materials. Also, it doesn't affect viewport display of maps for materials other than those two.

Update During Spinner Drag

Views menu > Update During Spinner Drag

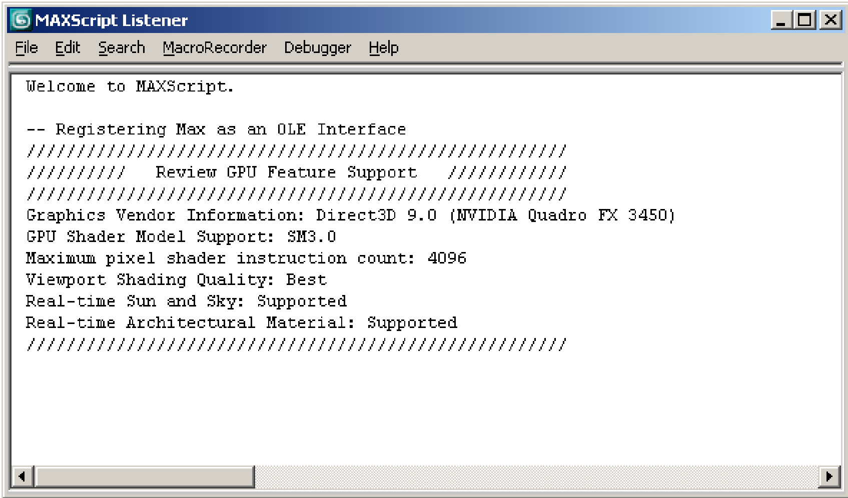
When Update During Spinner Drag is on, dragging a spinner (such as a Radius spinner for a sphere) updates the effects in real time in the viewports. Default=on.

When Update During Spinner Drag is off, the effect is updated after the drag, when you release the mouse. Use this option when you're adjusting processor-intensive controls.

Diagnose Video Hardware

Help menu > Diagnose Video Hardware

This command runs a MAXScript script that displays information about your system’s graphic display configuration. This can be useful for telling you whether some interactive viewport features such as shadows on page 4991 are supported.



Expert Mode

Views menu > Expert Mode

Keyboard > Ctrl+X

When Expert mode is on, the title bar, toolbar, command panel, status bar, and all of the viewport navigation buttons are removed from the display, leaving only the menu bar, time slider, and viewports. Use Expert mode when you need to view your composition alone without the rest of the interface.

With the ability to customize the user interface in 3ds Max, you can create your own versions of Expert mode by hiding whatever you want item-by-item. Expert mode is only a quick way to hide everything that can be hidden at once.

You can assign keyboard shortcuts to hide and unhide the command panel, toolbars, and so on and then use these while in Expert mode. You can also use the quad menu to access tools quickly in Expert mode as well.

Procedures

To turn on Expert mode, do one of the following:

- Choose Views menu > Expert Mode.
- Press Ctrl+X.

To turn off Expert mode and return to full display, do one of the following:

- Click the Cancel Expert Mode button to the right of the time slider.
- Press Ctrl+X.
- Choose Views menu > Expert Mode.

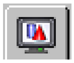
Controlling Object Display

You use the Display panel, layers on page 7438, and Scene Explorer on page 7379 to control how objects are displayed in viewports, and to hide or freeze objects. This section covers usage of the Display panel controls.

TIP You can also use the Isolate Selection command on page 219 to hide everything except your selection set.

See also:

- Object Display Properties on page 309

Display Color Rollout 

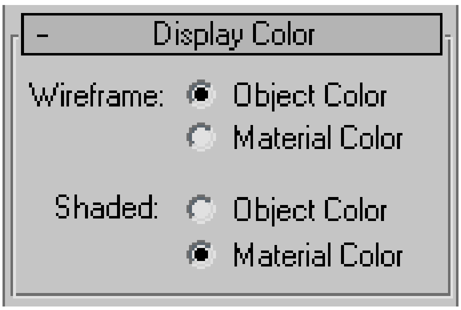
Display panel > Display Color rollout

The Display Color rollout specifies whether 3ds Max displays objects using their object colors or their diffuse material colors on page 7955, when the objects have their display properties on page 305 set to By Object. If the display properties of an object is set to By Layer, the layer color will be used for the display. You can choose one method for wireframe display and a different one for shaded display. In each shading mode you can specify whether the material or the object color is used.

As a default, all new objects have their display properties set to By Layer. The default can be changed in Customize > Preferences > Preferences dialog > General panel > Layer Defaults group. If you turn off Default To By Layer For New Nodes, all new objects created in 3ds Max will display in the viewports based on the settings in the Display Color rollout. You can switch individual objects between By Object and By Layer by setting the Display Properties in the Object Properties dialog on page 305, accessible by right-clicking any selected object.

If the object color box displays black and white rectangles, this indicates that the object has its display properties set to By Layer.

Interface



Wireframe Controls the color of objects when the viewport is in wireframe display mode.

Object Color Displays the wireframes in object color.

Material Color Displays the wireframes using the material color.

Shaded Controls the color of the object when the viewport is in any shaded display mode.

Object Color Displays the shaded objects using the object color.

Material Color Displays the shaded objects using the material color.

Hide By Category Rollout

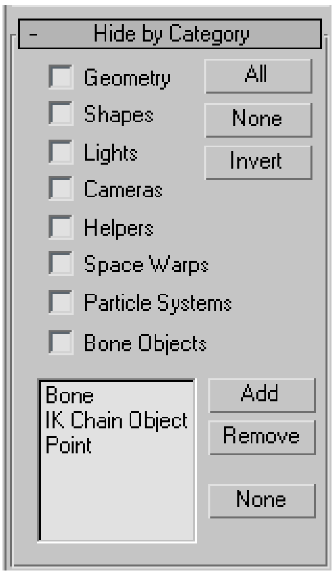
Display panel > Hide By Category rollout

The Hide By Category rollout toggles the display of objects by category (objects, cameras, lights, and so on).

By default, 3ds Max displays all objects in the scene. Objects hidden by category aren't evaluated in the scene, so hiding objects by category improves performance.

You can use any of the default display filters provided, or add new display filters for fast selection of objects to hide.

Interface



Turn on the check boxes to hide objects of that category. You can use the All, None, and Invert buttons to quickly change the settings of the check boxes.

The Display Filter box gives you finer control in creating categories to hide. Click the Add button to display a list of display filters. Hold down the Ctrl key and click the filter name to select whatever category you'd like to hide.

Geometry Hides all geometry in the scene.

Shapes Hides all shapes in the scene.

Lights Hides all lights in the scene.

Cameras Hides all cameras in the scene.

Helpers Hides all helpers in the scene.

Space Warps Hides all space warps in the scene.

Particle Systems Hides all particle systems in the scene.

Bone Objects Hides all bones in the scene.

Bones Hides all bones in the scene.

IK Chain Hides all IK chains in the scene.

Point Hides all points in the scene.

All Hides everything in the scene.

None Unhides everything in the scene

Invert Hides everything that is visible and unhides everything currently hidden.

Add Adds a display filter category to the list.

Remove Removes a display filter category.

None Deselects all highlighted display filters in the list.

Hide Rollout

Display panel > Hide rollout

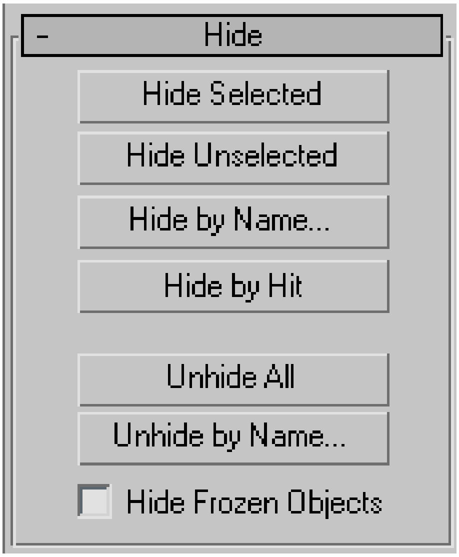
The Hide rollout provides controls that let you hide and unhide individual objects by selecting them, regardless of their category.

You can also hide and unhide objects using the Display Floater on page 7666.

See also:

- Hide By Category Rollout on page 178

Interface



Hide Selected Hides the selected objects.

Hide Unselected Hides all visible objects except the selected ones. Use this to hide all objects except the one you're working on. Objects hidden by category aren't affected.

Hide by Name Displays a dialog you use to hide objects you choose from a list. SeeSelect From Scene on page 228, which describes nearly identical controls.

Hide by Hit Hides any object you click in the viewport. If you hold the Ctrl key while selecting an object, that object and all of its children are hidden. To exit Hide by Hit mode, right-click, press Esc, or select a different function. This mode is automatically turned off if you hide all objects in the scene.

Unhide All Unhides all hidden objects. The unhide buttons are available only when you have specifically hidden one or more objects. They won't unhide objects hidden by category.

NOTE If you click Unhide All in a scene with hidden layers, a dialog will pop up prompting you to unhide all layers. You cannot unhide an object on a hidden layer.

Unhide by Name Displays a dialog you use to unhide objects you choose from a list. See Select From Scene on page 228, which describes nearly identical controls.

NOTE If you select an object on a hidden layer, a dialog will pop up prompting you to unhide the object's layer. You cannot unhide an object on a hidden layer.

Hide Frozen Objects Hides any frozen objects. Turn it off to display hidden frozen objects.

Freeze Rollout

Display panel > Freeze rollout

The Freeze rollout provides controls that let you freeze or unfreeze on page 7989 individual objects by selecting them, regardless of their category.

Frozen objects remain on the screen, but you can't select, transform, or modify them. By default, frozen objects turn dark gray. Frozen lights and cameras, and their associated viewports, continue to work as they normally do.

You can choose to have frozen objects retain their usual color or texture in viewports. Use the Show Frozen In Gray toggle in the Object Properties dialog on page 305.

Interface

Freeze Selected Freezes the selected object(s).

Freeze Unselected Freezes all visible objects except the selected ones. Use this to quickly freeze all the objects except the one you're working on.

Freeze by Name Displays a dialog that lets you choose objects to freeze from a list. See Select From Scene on page 228, which describes nearly identical controls.

Freeze by Hit Freezes any object you click in a viewport. If you press Ctrl while selecting an object, that object and all of its children are frozen. To exit Freeze by Hit mode, right-click, press Esc, or select a different function. This mode is automatically turned off if you freeze all objects in the scene.

Unfreeze All Unfreezes all frozen objects.

NOTE If you click Unfreeze All in a scene with frozen layers, a dialog opens prompting you to unfreeze all layers. You cannot unfreeze an object on a frozen layer.

Unfreeze by Name Displays a dialog that lets you choose objects to unfreeze from a list. See Select From Scene on page 228, which describes nearly identical controls.

NOTE If you unfreeze by name an object on a frozen layer, a dialog opens prompting you to unfreeze the object's layer. You cannot unfreeze an object on a frozen layer.

Unfreeze by Hit Unfreezes any object you click in the viewport. If you press Ctrl while selecting an object, that object and all of its children are unfrozen.

If you select an object on a frozen layer, a dialog will pop up prompting you to unfreeze the object's layer. You cannot unfreeze an object on a frozen layer.

Display Properties Rollout

Display panel > Display Properties rollout

The Display Properties rollout provides controls for altering the display of selected objects.

See also:

- Link Display Rollout on page 188

Procedures

To display trajectories using the Display panel:

1. Select one or more animated objects.
2. Right-click the selection, and choose Properties.
3. In the Display properties group, click By Layer to change it to By Object, and then click OK.
4. Expand the Display Properties rollout in the Display panel.
5. Turn on Trajectory.

   By default, object trajectories appear with the following properties:

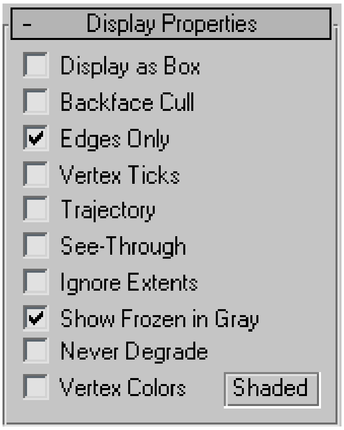
   - The trajectory curve is drawn in red.
   - Frame increments display as white dots on the curve.
   - Position keys display as red boxes surrounding the appropriate frame dot on the curve. The boxes are white when the object is selected.
   - If Views > Show Key Times is turned on, the keyframe numbers are displayed along side the keys on the trajectory.

Trajectories can also be displayed through Object Properties. Right-click any object and choose Properties, then in the Display properties group change By Layer to By Object. Turn on Trajectories when it becomes available in the Display Properties group.

You can change the colors for these items on the Colors panel on page 7712 of the Customize User Interface dialog.

You can also use object properties to display trajectories: right-click any object and choose Properties, then turn on Trajectory.

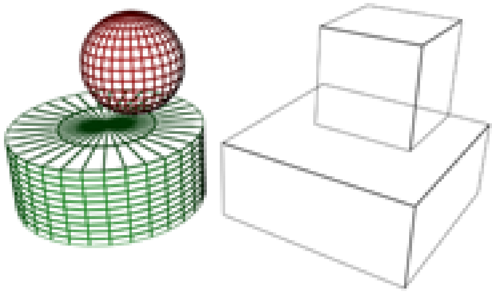
Interface



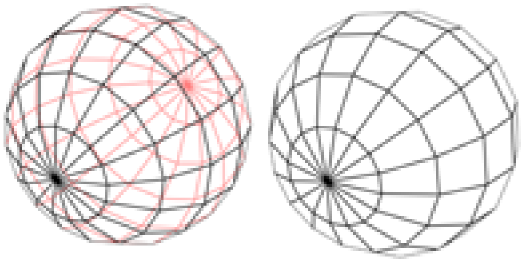
The first three options reduce the displayed geometric complexity of selected objects in a scene, resulting in faster response time because the computer has less to calculate. These options are also available in the Display Properties group of the Object Properties dialog > General panel on page 305 and the Display floater on page 7666.

Display as Box Toggles the display of selected objects, including 3D objects and 2D shapes, as bounding boxes on page 7932. Produces minimum geometric complexity.

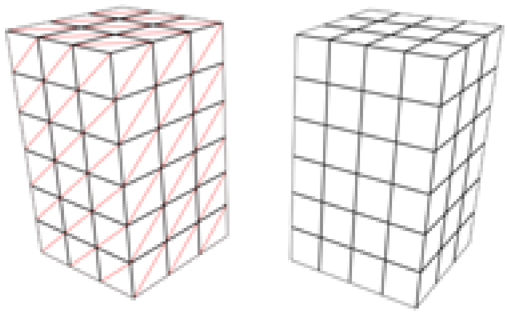
Particle systems appear as bounding boxes when adaptive degradation takes effect. Because particle systems naturally exist in world space, their bounding box is always oriented parallel to the world planes.



Backface Cull Toggles the display of faces, edges, and vertices with normals on page 8059 pointing away from the point of view. When off, all entities are visible. Default=off.

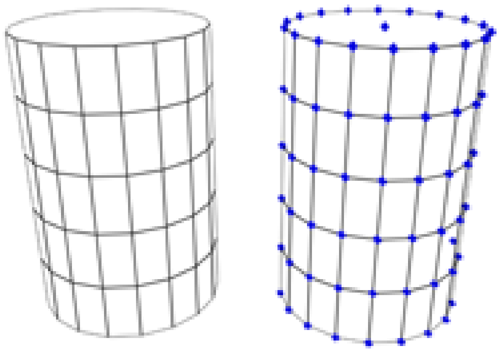


Edges Only Toggles the display of hidden edges and polygon diagonals on page 7953. When on, only outside edges appear. When off, all mesh geometry appears. Applies to Wireframe viewport display mode, as well as other modes with Edged Faces turned on.

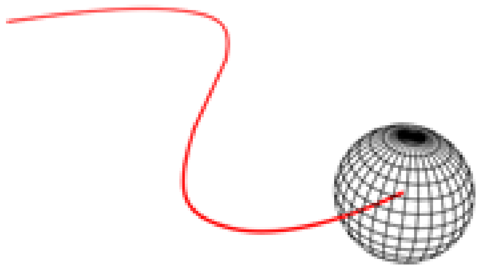


Vertex Ticks Displays the vertices in the selected geometry as tick marks.

If the current selection has no displayed tick marks, the check box is clear. If some of the vertices in the current selection display tick marks, the check box contains a gray X. If all vertices in the current selection display tick marks, the check box contains a black X.



Trajectory Toggles trajectory on page 8154 display for the selected object so its trajectory is visible in viewports.

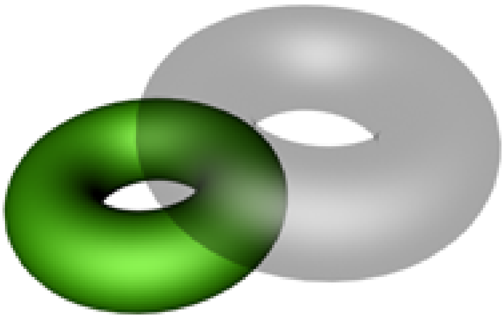


See-Through Makes the object or selection translucent in viewports. This setting has no effect on rendering: it simply lets you see what's behind or inside an object in a crowded scene, and is especially useful in adjusting the position of objects behind or inside the See-Through object. This is very handy when you have objects within other objects in your scene.

This option is also available from Object Properties dialog on page 305 and the Tools > Display Floater on page 7666.

You can customize the color of see-through objects by using the Colors panel on page 7712 of the Customize > Customize User Interface dialog on page 7697.

Keyboard shortcut (default): Alt+X



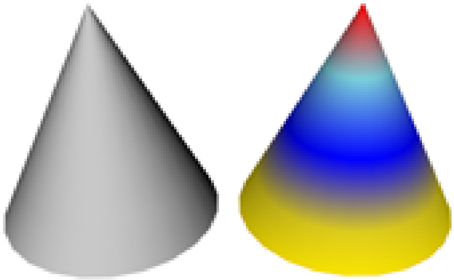
Ignore Extents When turned on, the object is ignored when you use the display control Zoom Extents. Use this on distant lights.

Show Frozen in Gray When on, the object turns gray in viewports when you freeze it. When off, viewports display the object with its usual color or texture even when it is frozen. Default=on.

Never Degrade When on, the object is not subject to adaptive degradation on page 7900.

Vertex Colors Displays the effect of assigned vertex colors. You assign vertex colors using the Assign Vertex Color utility, or the VertexPaint modifier. Once vertex colors have been assigned they can also be edited in the Vertex Properties rollout in the editable mesh or editable poly in vertex or face sub-object level.

The Shaded button determines whether the object with the assigned vertex colors appears shaded in the viewport. When this button is off, the colors are unshaded and appear in their pure RGB values, looking a little like self-illuminated materials. When the Shaded button is on, the colors appear like any other assigned color in the viewports.



Link Display Rollout

Display panel > Link Display rollout

The Link Display rollout provides controls that alter the display of hierarchical linkages on page 8002.

Interface

Display Links Displays a wireframe representation of any hierarchical links affecting the selected object.

NOTE Display Links must be turned on in order to see Joint Limits on a inverse kinematics chain.

Link Replaces Object Replaces the selected object with the wireframe representation of the hierarchical link. This option offers another way to reduce the geometric complexity of selected objects in a scene. See also Display Properties rollout on page 183.

The Draw Links As Lines option on the Viewports panel on page 7753 of the Preference Settings dialog further reduces the display of links to a single line.

6 Selecting Objects

In most cases, before you can perform an action on an object or objects in your scene, you must first *select* them. Thus, the act of selection is an essential part of the modeling and animation process.

3ds Max provides you with a variety of selection tools, which are covered in this chapter. Besides the basic techniques of selecting single and multiple objects using mouse and keyboard, the topics here discuss the use of named selection sets and other features that help you manage object selection, such as hiding and freezing objects and layers. Also included is an introduction to sub-object selection, essential to working with an object's underlying geometry.

Lastly, a technique for grouping objects is presented. Grouping lets you create more permanent selections that have many of the characteristics of independent objects.

This section presents the following topics:

Using Groups on page 266

Introducing Object Selection

3ds Max is an object-oriented program. This means that each object in the 3D scene carries instructions that tell the program what you can do with it. These instructions vary with the type of object.

Because each object can respond to a different set of commands, you apply commands by first selecting the object and then selecting the command. This is known as a noun-verb interface, because you first select the object (the noun) and then select the command (the verb).

Identifying the Selection Interface

In the user interface, selection commands or functions appear in the following areas:

- Main toolbar
- Edit menu
- Quad menu (while objects are selected)
- Tools menu
- Track View
- Display panel
- Schematic View

The buttons on the main toolbar provide a direct means of selection. The Select From Scene dialog on page 228 is easy to use, while the Edit menu provides more general selection commands, plus methods of selecting objects by property. Perhaps the most powerful selection tool is Scene Explorer on page 7379, which lets you select objects by various methods, and also edit object hierarchies and properties. Track View and Schematic View let you select objects from a hierarchical list.

Selecting From the Quad Menu

The quickest way to select an object is from the Transform quadrant of the quad menu, where you can easily switch among the Move, Rotate, Scale, and Select modes. Choose any of these and click the object you want to select in the viewport.

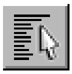
Selecting by Name

Another quick way to select an object is to use the keyboard shortcut for the Select By Name command. Press H on the keyboard to open the Select From Scene dialog on page 228 and then select the object by name from the list. This is a foolproof way to ensure you select the correct object when the scene contains many overlapping objects.

Selection Buttons

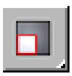
Another way to select an object is to click one of these buttons, and then click the object.

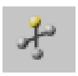
 Select Object

 Select by Name

 Select And Move

 Select And Rotate

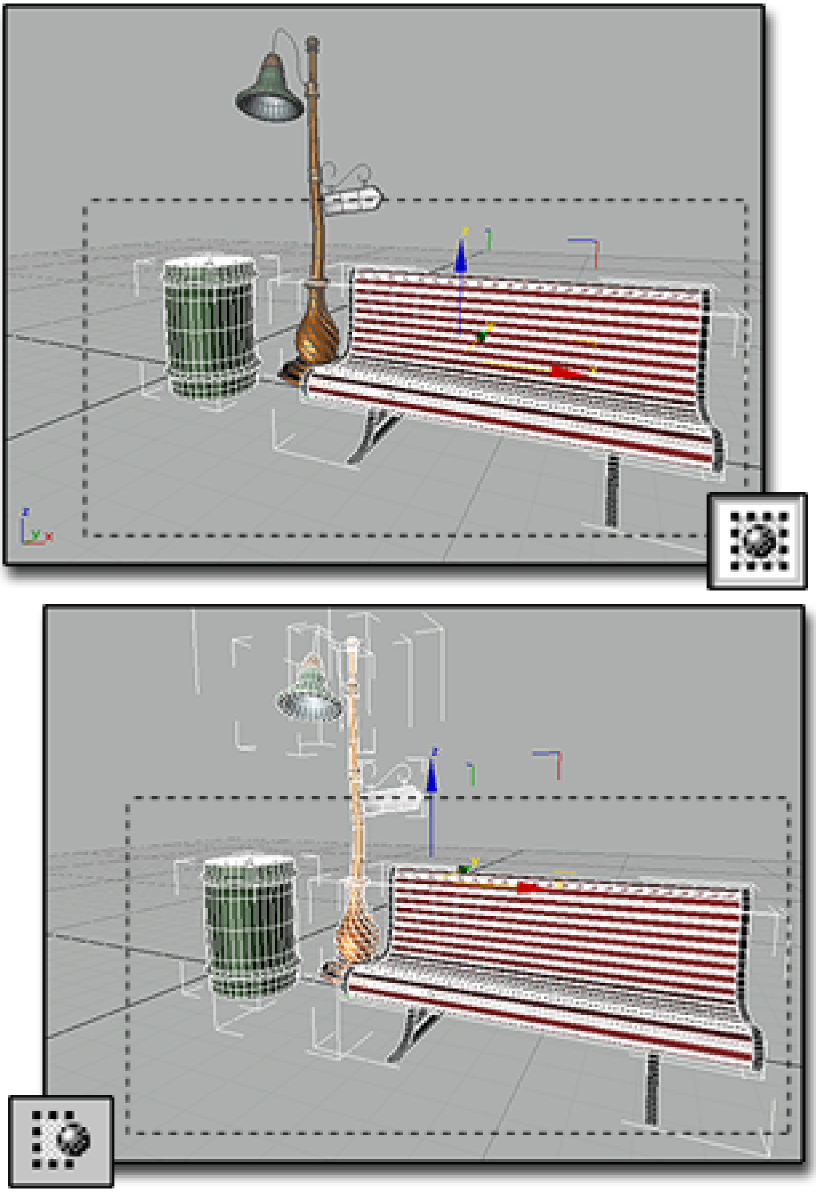
 Select And Scale

 Select And Manipulate

The main toolbar has several selection-mode buttons. When any of the selection buttons is active, the program is in a state where you can select objects by clicking them.

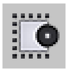
Of the selection buttons, you use Select Object when you want selection only. The remaining buttons let you both select and transform or manipulate your selection. Use transforms to move, rotate, and scale your selection. See Moving, Rotating, and Scaling Objects on page 929 and Select and Manipulate on page 2613.

Crossing Versus Window Selection



Above: Window selection selects the trash can and bench, but not the streetlight.

Below: Crossing selection selects all three: trash can, bench, and streetlight.

 One way to select multiple objects simultaneously is to drag a region, such as a rectangle, around them. The Window/Crossing toggle, available from the main toolbar, switches between Window and Crossing modes when you select by region. In Window mode, you select only the objects within the selection. In Crossing mode, you select all objects within the region, plus any objects crossing the boundaries of the region.

Edit Menu Commands

The Edit menu contains selection commands that operate globally on your objects.

Edit menu selection commands include:

Select All on page 249

Select None on page 249

Select Invert on page 250

Select Similar on page 250

Select By > Color on page 251

Select By > Name on page 251 (also a toolbar button)

Select By > Layer on page 252

Selection Region > Rectangular Region on page 252

Selection Region > Circular Region on page 253

Selection Region > Fence Region on page 254

Selection Region > Lasso Region on page 255

Selection Region > Paint Selection Region on page 256

Selection Region > Window on page 258 (also a toolbar button)

Selection Region > Crossing on page 260 (also a toolbar button)

Edit Named Selection Sets on page 241

Tools Menu Commands

The Tools menu provides the Scene Explorer commands as well as access to modeless on page 8045 selection dialogs or "floaters." You can place them anywhere on the screen, or minimize them by right-clicking the title bar and choosing Minimize:

- **New Scene Explorer (and related commands)** See Scene Explorer on page 197.
- **Display Floater** Provides options for hiding and freezing selections as well as some display options. See Display Floater on page 7666.
- **Layer Manager** This modeless dialog lets you select objects individually and by layer, and change display properties such as Hide and Freeze. See Layer Manager on page 7441.

Scene Explorer

Scene Explorer on page 7379 gives you a modeless dialog for selecting and linking objects as well as changing object properties such as the name and display characteristics. You can manipulate hierarchical relationships via drag-and-drop techniques, and use various search methods including a powerful Boolean editor to fine-tune your selection. You can also edit properties for multiple objects at once simply by selecting them and then changing one of them.

Track/Schematic View Selection

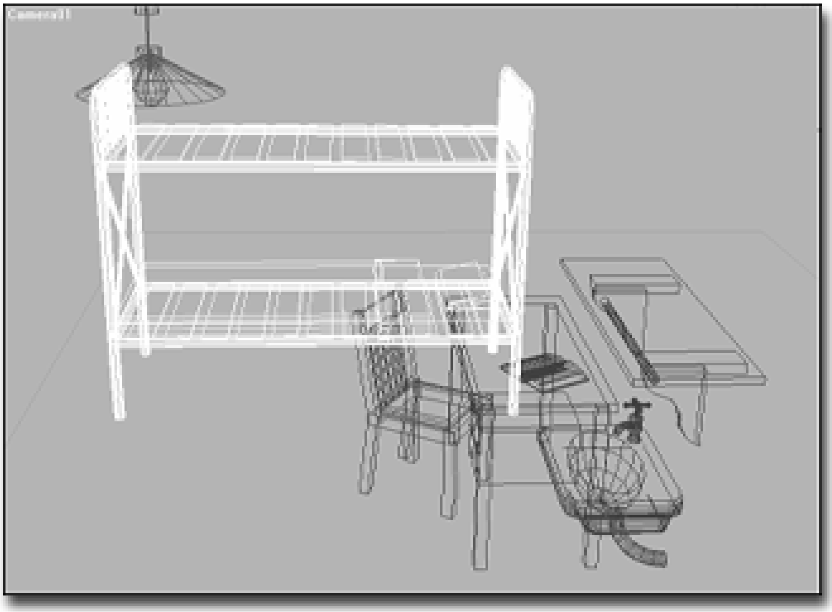
Track View on page 3503 is primarily designed as an animation tool, but you can also use its Hierarchy List window as an alternative method of selecting objects by name and hierarchy. This works in both the Curve Editor and Dope Sheet modes of Track View.

Schematic View on page 7407 is specifically designed to let you navigate your scene efficiently, presenting a hierarchical view and letting you select objects and their properties by name.

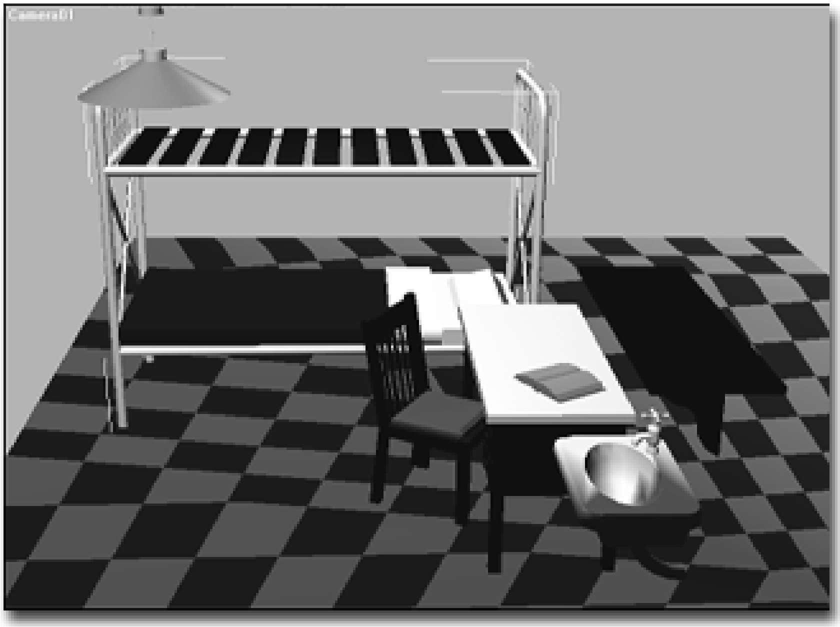
Display Panel Selection

The Display panel provides options for hiding and freezing objects. These techniques exclude objects from other selection methods, and are useful in simplifying complex scenes. Frozen objects are still visible, but hidden objects are not.

Basics of Selecting Objects



Bed selected in wireframe



Bed selected in smooth and shaded view

The most basic selection techniques use either the mouse, or the mouse in conjunction with a keystroke.

Procedures

To select an object in the viewport:

1. Click one of the selection buttons on the toolbar: Select Object, Select And Move, Select And Rotate, Select And Scale, or Select And Manipulate.

   Alternatively, right-click in a viewport to open the quad menu, and from the Transform menu choose Move, Rotate, Scale, or Select.

2. In any viewport, move the cursor over the object you want to select.

   The cursor changes to a small cross when it's positioned over an object that can be selected.

   The valid selection zones of an object depend on the type of object and the display mode in the viewport. In shaded mode, any visible surface of an object is valid. In wireframe mode, any edge or segment of an object is valid, including hidden lines.

3 While the cursor displays the selection cross, click to select the object (and to deselect any previously selected object).

A selected wireframe object turns white. A selected shaded object displays white brackets at the corners of its bounding box.

To select all objects do one of the following:

- Choose Edit menu > Select All.
This selects all objects in your scene.
- On the keyboard press Ctrl+A.

To invert the current selection do one of the following:

- Choose Edit menu > Select Invert.
This reverses the current selection pattern. For example, assume you begin with five objects in your scene, and two of them are selected. After choosing Invert, the two are deselected, and the remaining objects are selected.
- On the keyboard press Ctrl+I.

To extend or reduce a selection:

- Hold down Ctrl while you click to make selections.
This toggles the selection state of the objects you select. Use this method to select or deselect objects. For example, if you have two objects selected and Ctrl+click to select a third, the third object is added to the selection. If you now Ctrl+click any of the three selected objects, that object is deselected.

TIP You can also hold down Alt while you click to remove objects from selections.

To lock a selection:

1 Select an object.

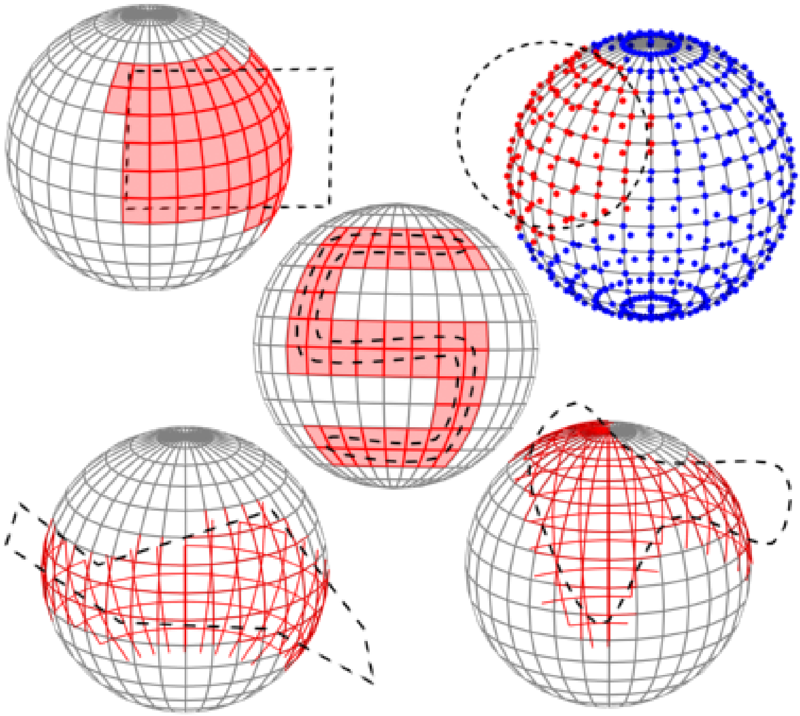
2  Click the Selection Lock Toggle on page 7539 on the status bar to turn on locked selection mode.

While your selection is locked, you can drag the mouse anywhere on the screen without losing the selection. The cursor displays the current selection icon. When you want to deselect or alter your selection, click the Lock button again to turn off locked selection mode. The keyboard toggle for locked selection mode is Spacebar.

To deselect an object, do any of the following:

- Hold down the Alt key, and either click an object, or drag a region around the object to deselect it.
- Hold down the Ctrl key and click to deselect a selected object. This is a toggle; it also selects non-selected objects.
- To deselect all objects in the scene, choose Edit menu > Select None, or click an empty area of a viewport anywhere outside the current selection.

Selecting by Region



Top Left: Selecting face sub-objects with a rectangular region

Top Right: Selecting vertex sub-objects with a circular region

Center: Selecting face sub-objects with a painted region

Bottom Left: Selecting edge sub-objects with a fence region

Bottom Right: Selecting edge sub-objects with a lasso region

The region-selection tools let you use the mouse to select one or more objects by defining an outline or area.

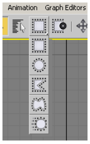
Region Selection

By default, when you drag the mouse a rectangular region is created. When you release the mouse all objects within and touched by the region are selected.

The remainder of this topic describes how you can change each of these settings.

NOTE If you hold down Ctrl while specifying a region, the affected objects are added to the current selection. Conversely, if you hold down Alt while specifying a region, the affected objects are removed from the current selection.

Setting Region Type



The type of region you define when you drag the mouse is set by the Region flyout button to the right of the Select By Name button. You can use any of five types of region selection:

- **Rectangular Region** Dragging the mouse selects a rectangular region. See Rectangular Selection Region on page 252.
- **Circular Region** Dragging the mouse selects a circular region. See Circular Selection Region on page 253.
- **Fence Region** Draw an irregular selection-region outline by alternating between moving the mouse and clicking (begin with a drag). See Fence Selection Region on page 254.
- **Lasso Region** Dragging the mouse outlines an irregular selection region. See Lasso Selection Region on page 255.
- **Paint Region** Drag the mouse over objects or sub-objects to be included in the selection. See Paint Selection Region on page 256

Setting Region Inclusion

This option lets you specify whether to include objects touched by the region border. It applies to all region methods.

Choose Edit menu > Region to display a submenu of the following two items. Only one can be active at a time. The option is also available on the main toolbar.

- **Window** Selects only objects that are completely within the region. See Select Region Window on page 258.

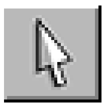
- **Crossing** Selects all objects that are within the region and crossing the boundaries of the region. This is the default region. See Select Region Crossing on page 260.

The Window/Crossing toggle on page 261 on the main toolbar also switches between these two modes.

You can set up a preference to switch automatically between Window and crossing based on the direction of your cursor movement. See Auto Window/Crossing by Direction on page 7748 in General Preferences.

Procedures

To make a region selection using defaults:

1  Click Select Object on page 227.

2 Drag the mouse to define a region.

   A rubber-band rectangle appears.

3 Release the mouse button to select all objects within or touching the region.

   The selected objects turn white.

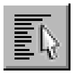
You can also use the Select and Transform buttons on the main toolbar to select by region. You must start defining the region over an unselectable area of the viewport. Otherwise, you'll transform the object beneath your mouse when you begin to drag.

Using Select By Name

The Select By Name command opens the Select From Scene dialog, which lets you select objects by their assigned names without having to click in the viewports.

Procedures

To select objects by name:

1. Do one of the following:
   -  On the main toolbar, click Select By Name.
   - Choose Edit menu > Select By > Name.
   - Press H.
     The Select From Scene dialog opens. By default, this dialog lists all objects in the scene. Any selected objects are highlighted in the list.
2. Choose one or more objects in the list. To select multiple objects, drag vertically in the list or use Ctrl to add to the selection.
3. Click Select to make the selection.

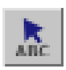
   The dialog closes and the objects are selected.
4. Alternatively, to select a single object and close the dialog at the same time, double-click the object name.

For more information, seeSelect From Scene on page 228.

Using Named Selection Sets

You can assign a name to the current selection, and then later reselect those objects by choosing their selection name from a list.

Named Selection Sets

 You can also edit the contents of named sets from the Named Selection Sets dialog on page 241.

Editing Named Selections

As you model and create a scene, you're likely to rearrange the objects making up your named selection sets. If you do, you'll need to edit the contents of those sets.

Procedures

To assign a name to a selection set:

1. Select one or more objects or sub-objects using any combination of selection methods.
2. Click in the Named Selection field on the main toolbar.
3. Enter a name for your set. The name can contain any standard ASCII characters, including letters, numerals, symbols, punctuation, and spaces.

   NOTE Names are case-sensitive.

4. Press Enter to complete the selection set.

You can now select another combination of objects or sub-objects and repeat the process to create another named selection set.

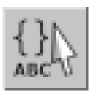
To retrieve a named selection set:

1. In the Named Selection field, click the arrow.

   NOTE If you're working with a sub-object selection set, you must be at the same level at which you created the selection set (for example, editable mesh > vertex) for it to appear on the list.

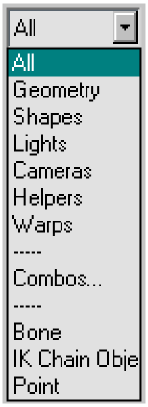
2. On the list, click a name.

To edit named selection sets:

- 

On the main toolbar, click NaEdit med Selection Sets to open the Named Selection Sets dialog.

Using Selection Filters



You can use the Selection Filter list on the main toolbar to deactivate selection for all but a specific category of object. By default, all categories can be selected, but you can set the Selection Filter so that only one category, such as lights, can be selected. You can also create combinations of filters to add to the list.

For greater ease of use while working with animations, you can choose filters that let you select only Bones, objects in IK chains, or Points.

Using Combos

The Combos feature allows you to combine two or more categories into a single filter category.

Procedures

To use the selection filter:

- Click the Selection Filter arrow and click a category from the Selection Filter list.
  Selection is now limited to objects defined in this category. The category remains in effect until you change it.

  The following categories are available:

  All
  All categories can be selected. This is the default setting.

  Geometry

Only geometric objects can be selected. This includes meshes, patches, and other kinds of objects not specifically included in this list.

Shapes
Only shapes can be selected.

Lights
Only lights (and their targets) can be selected.

Cameras
Only cameras (and their targets) can be selected.

Helpers
Only helper objects can be selected.

Warps
Only space warps can be selected.

Combos
Displays a Filter Combinations dialog on page 236 that lets you create custom filters.

Bone
Only bones objects can be selected.

IK Chain
Only objects in IK chains can be selected.

Point
Only point objects can be selected.

To create a combination category:

1. From the drop-down list, choose Combos to display the Filter Combinations dialog on page 236.

   All single categories are listed.

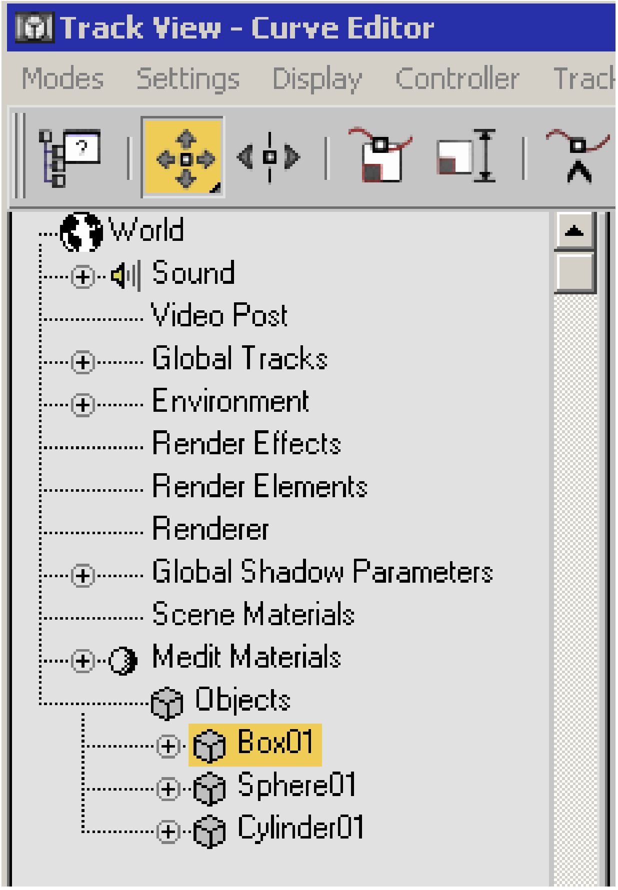
2. Select the categories you want to combine.

3. Click Add.

   The combination appears in a list to the right, abbreviated by the first letter of each category. Click OK.

   For example, if you selected Geometry, Lights, and Cameras, the Combo would be named GLC. This name appears below Combo on the drop-down list. For more information, see Selection Filters List on page 235.

Selecting with Track View



Track View provides sophisticated methods to edit your animation tracks. In addition, its Hierarchy list displays all objects in the scene by name and hierarchy. Using Track View, you can select any object in the scene by clicking its object icon in the Hierarchy list.

Procedures

You can use Track View selection functionality in both the Curve Editor on page 3518 and the Dope Sheet on page 3519. This procedure illustrates usage of the Curve Editor; the same methods work in the Dope Sheet.

To open Track View and display and select objects:

1.  On the main toolbar, click Curve Editor (Open).
2. Click any cube icon in the list to select the named object.

You can make the following kinds of selections:

- Select several adjacent objects in the list. Click the first object, hold down Shift, and click another object elsewhere in the list.
- Modify the selection by pressing Ctrl while clicking. Ctrl lets you toggle individual items on and off without deselecting others in the list.
- Select an object and all its descendants. Press and hold Alt, right-click the object's cube icon (keep the right mouse button held down), and choose Select Children from the menu.

You can open a Track View window for the sole purpose of selecting objects by name. Shrink the window until only a portion of the Hierarchy appears, and then move the window to a convenient area on your screen.

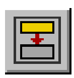
Selecting with Schematic View

Schematic view is a window that displays the objects in your scene in a hierarchical view. It gives you an alternate way to select and choose the objects in your scene and navigate to them.

When the Modify panel is open, double-clicking an object modifier in Schematic view navigates the modifier stack to that modifier for quick access to its parameters.

Procedures

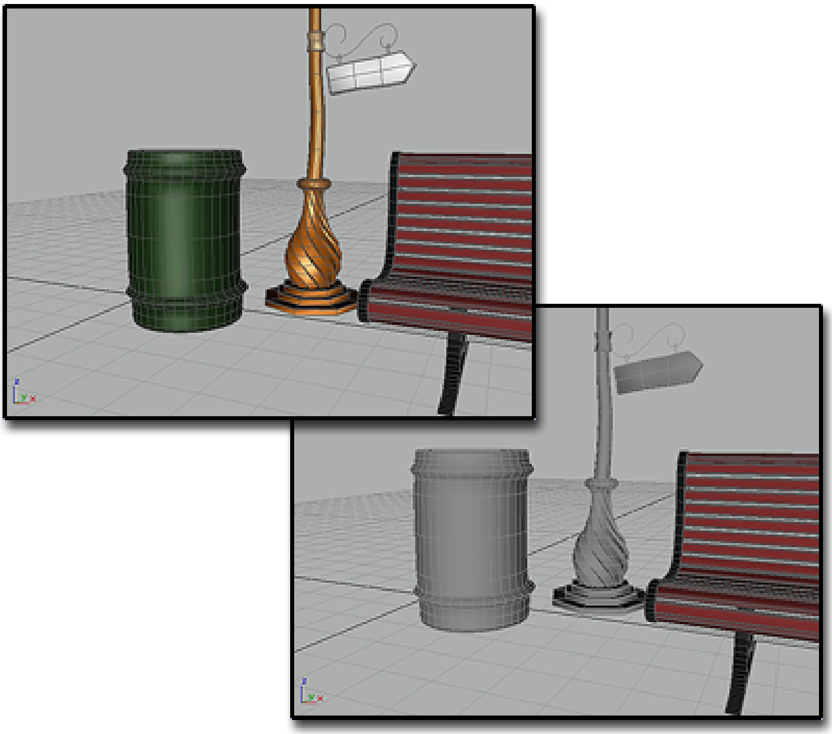
To open Schematic View and display and select objects:

1.  Click Open Schematic View on the main toolbar.
2. Click the rectangle containing the name of your object.

You can select any number of objects in Schematic View using standard methods, including dragging a region. For more information, see Using Schematic View on page 7411.

Freezing and Unfreezing Objects

You can freeze any selection of objects in your scene. By default, frozen objects, whether wireframe or rendered, turn a dark gray. They remain visible, but can't be selected, and therefore can't be directly transformed or modified. Freezing lets you protect objects from accidental editing and speeds up redraws.



Above: No layers frozen

Below: Trash can and streetlight are frozen, and displayed in gray

You can choose to have frozen objects retain their usual color or texture in viewports. Use the Object Properties dialog > General panel > Display Properties > Show Frozen In Gray toggle on page 312.

Frozen objects are similar to hidden objects. Linked, instanced, and referenced objects behave when frozen just as they would if unfrozen. Frozen lights and cameras and any associated viewports continue to work as they normally do.

For more information, see Freeze Rollout on page 181.

Freezing Objects

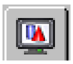
You can freeze one or more selected objects. This is the usual method to put objects "on hold."

You can also freeze all objects that are not selected. This method lets you keep only the selected object active, useful in a cluttered scene, for example, where you want to be sure no other objects are affected.

IMPORTANT Objects on a frozen layer cannot be unfrozen. If you try to unfreeze an object on a frozen layer (with Unfreeze All or Unfreeze By Name), you are prompted (by default on page 7749) to unfreeze the object's layer.

Procedures

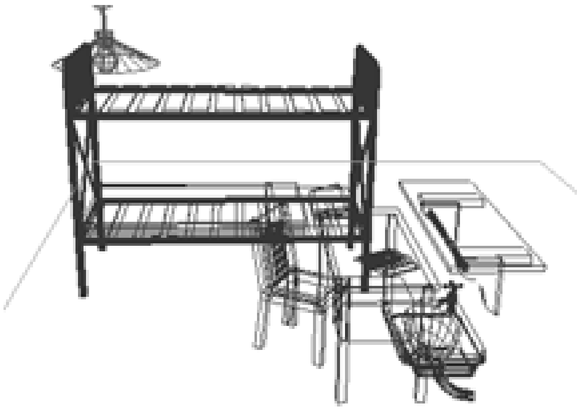
To access Freeze options, select one or more objects and then do one of the following:

- Open a scene explorer on page 7379 and use the check boxes in the Frozen column to freeze and unfreeze objects.
-  Open the Display panel and then expand the Freeze rollout.
- Choose Tools menu > Display Floater. This modeless dialog has the same options as the Freeze rollout. It also contains Hide options.
- Access the Object Properties dialog on page 305 from either the right-click (quad) menu or the Edit menu. Turn on Hide and/or Freeze.
- In the Layer Manager, click in the Freeze column to freeze/unfreeze each layer in the list.
- Right-click in the active viewport and choose a Freeze or Unfreeze command from the quad menu > Display quadrant.

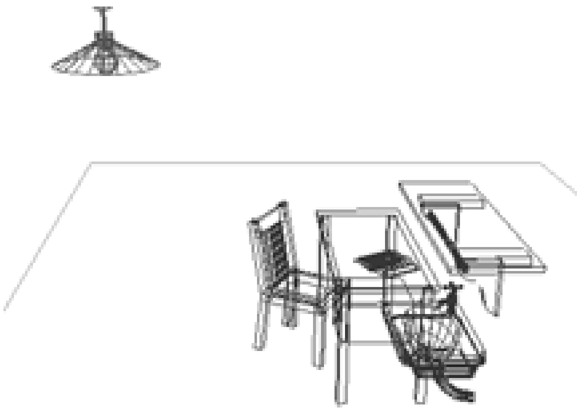
Hiding and Unhiding Objects by Selection

You can hide any selection of individual objects in your scene. They disappear from view, making it easier to select remaining objects. Hiding objects also speeds up redraws. You can then unhide all objects at once or by individual object name. You can also filter the list contents by category, so only hidden objects of a certain type are listed.

NOTE Hiding a light source doesn't alter its effect; it still illuminates the scene.



Original scene



Scene with bed hidden

Hiding objects is similar to freezing objects. Linked, instanced, and referenced objects behave when hidden just as they would if unhidden. Hidden lights and cameras and any associated viewports continue to work normally.

For more information, see Hide Rollout on page 179.

Hiding Objects

Hiding objects is similar to freezing objects. See Freezing and Unfreezing Objects on page 212. You can hide one or more selected objects. You can also hide all objects that are not selected.

Another option is to hide objects by category. See Hiding and Unhiding Objects by Category on page 216.

Unhiding Objects

You can unhide objects in either of two ways:

- Use Unhide All to unhide all objects at the same time.
- Use All On to display all objects at the same time.
- Use Unhide By Name to unhide object selectively. When you click Unhide By Name, the same dialog is displayed as for hiding, now called Unhide Objects.

The Unhide buttons are unavailable when no object in the scene is hidden.

Objects that were first hidden by selection and then hidden by category do not reappear. Although they are unhidden at the selection level, they are still hidden at the category level. For details, see Hiding and Unhiding Objects by Category on page 216.

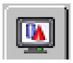
IMPORTANT Objects on a hidden layer cannot be unhidden. If you try to unhide an object on a hidden layer (with Unhide All or Unhide By Name), you are prompted (by default on page 7749) to unhide the object's layer.

Procedures

To access Hide options, do one of the following:

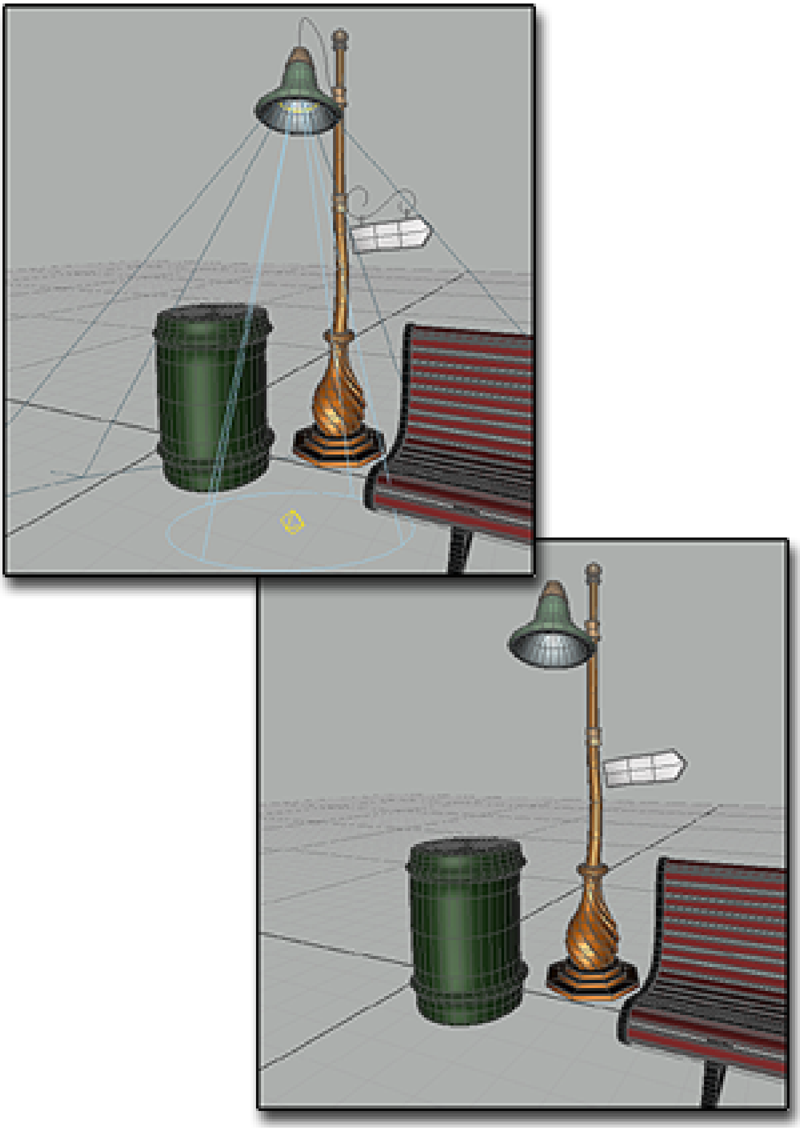
- Open a scene explorer on page 7379 and use the check boxes in the Hidden column to hide and unhide objects.
- Open the Layer Manager on page 7441.

In the Layer Manager, you can easily hide groups of objects or layers.

-  Open the Display panel. Click Hide, if necessary, to expand the rollout.
- Choose Tools menu > Display Floater. This modeless dialog has the same options as the Hide rollout. It also contains Freeze options.
- Access the Object Properties dialog on page 305 from either the right-click (quad) menu or the Edit menu. Turn on Hide, Freeze, or both. If the button is unavailable because By Layer is turned on, click By Layer to change it to By Object.
- Right-click in the active viewport and choose a Hide or Unhide command from the quad menu > Display quadrant.

Hiding and Unhiding Objects by Category

You can hide objects by category, the basic types of objects. For example, you can hide all lights in your scene at one time, or all shapes, or any combination of categories. By hiding all categories, your scene appears empty. Hidden objects, while not displayed, continue to exist as part of the geometry of your scene but cannot be selected.



Above: All objects displayed

Below: Lights and shapes are hidden

Hiding Geometry and Particle Systems

Geometry and particle systems have separate categories, even though particle systems are also geometry.

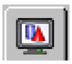
- Selecting Geometry hides all geometry in the scene, including particle systems. The option for particle systems becomes unavailable.
- Selecting Particle Systems hides only these objects, leaving the other geometry unaffected.

Effects of Hiding by Category

- If you create an object in a category that is hidden, the software turns off hiding for that category and unhides the objects within the category.
- Unhiding by category has no effect on objects hidden with the controls on the Hide rollout (see Hiding and Unhiding Objects by Selection on page 214). These objects remain hidden. You need to use the controls on that rollout to unhide them.
- Unhiding by category has no effect on objects that are on a layer that is turned off. These objects remain invisible. You need to turn on their layer to display them.
- Unhiding by selection does not return a hidden object to the scene if the category of the object is hidden. The Unhide All and Unhide By Name controls continue to work, but the effect is not seen until the category is cleared.
- Lights hidden by category continue to shine. Views through cameras and targeted lights are still active.
- Linked, instanced, and referenced objects behave when hidden just as they would if visible.

Procedures

To hide a category of objects:

1.  Open the Display panel.
2. Click Hide by Category, if necessary, to expand the rollout. By default, all categories are off (unhidden) on this rollout.

3 Choose the category you want to hide. All objects of that category disappear from your scene as soon as you make the choice.

The same Hide By Category options appear on the Object Level panel of the Display Floater (Tools menu > Display Floater).

To unhide a category of objects:

- Deselect the category.
  All objects in the category reappear, unless some have been hidden by selection. See “Effects of Hiding by Category”.

Isolate Selection

Tools menu > Isolate Selection

Right-click to open the quad menu. > Display (upper-right) quadrant > Isolate Selection

Keyboard > Alt + Q

The Isolate Selection tool lets you edit a single object or selection set of objects while hiding the rest of the scene on a temporary basis. This guards against selecting other objects while working on a single selection. It allows you to focus on the objects you need to see, without the visual distraction of the surroundings. It also reduces the performance overhead that can come from displaying other objects in the viewports.

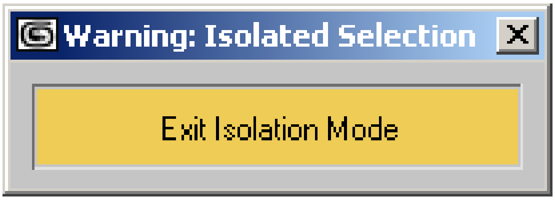
When you turn on Isolate Selection, the active viewport (Perspective and axonometric on page 7918 only) performs the Zoom Extents on page 7588 action on the isolated objects. When you exit Isolate Selection mode, Perspective viewports return to the previous zoom level, but axonometric viewports do not.

When an isolated selection includes multiple objects, you can select a subset of these, and choose Isolate Selection once again. This isolates the subset. However, clicking Exit Isolation unhides the entire scene. You can’t “step back” through individual levels of isolation.

NOTE Isolate Selection works only at the object level. You can’t choose it while at the sub-object level. If you go to a sub-object level while working with an isolated object, you can click Exit Isolation, but you can’t isolate sub-objects.

Interface

While the Isolate tool is active, a dialog labeled Warning: Isolated Selection appears.



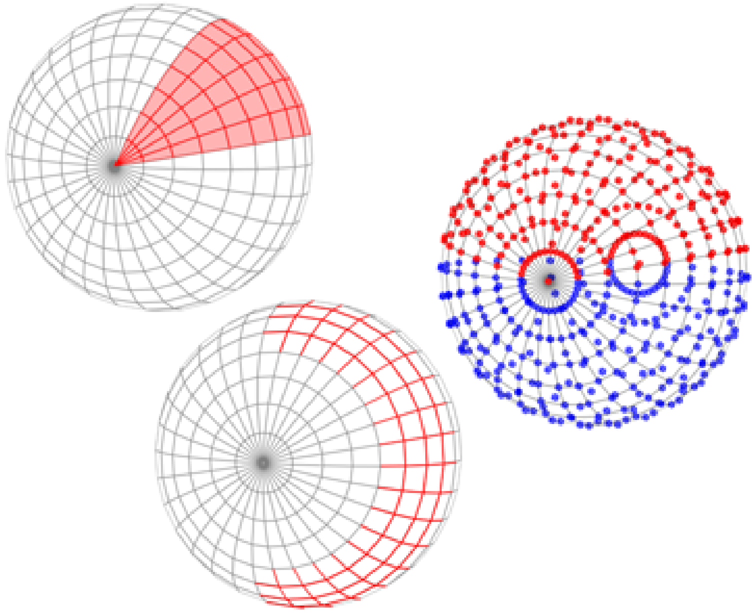
Exit Isolation Mode Click to end isolation, close the dialog, and unhide the rest of the scene.

The views are restored to what they showed before you chose Isolate Selection.

Introduction to Sub-Object Selection

This is a general introduction to sub-object selection. For specific information, see Editable Mesh on page 2075, Editable Patch on page 2019, Editable Poly on page 2123, and Editable Spline on page 659; for a discussion of NURBS sub-object selection, see Sub-Object Selection on page 2249.

When you model an object, often you edit a portion of its underlying geometry, such as a set of its faces or vertices. Or when you are working with a model, you may want to apply mapping coordinates to a portion of its underlying geometry. Use the methods described in this topic to make sub-object selections.



Left: A selection of face sub-objects

Middle: A selection of edge sub-objects

Right: A selection of vertex sub-objects

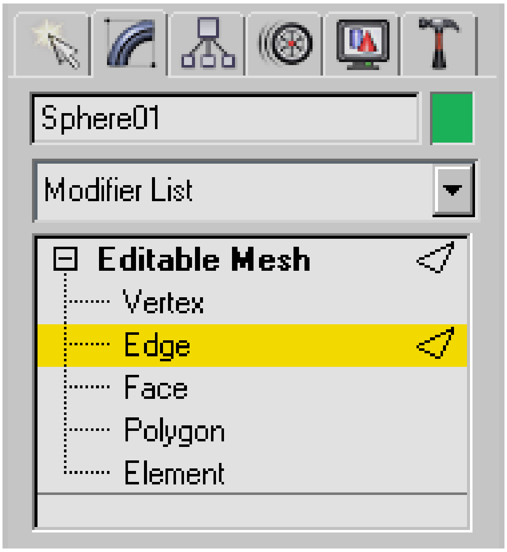
You can access sub-object geometry through a variety of methods. The most common technique is to convert an object into "editable" geometry such as a mesh, spline, patch, NURBS, or poly object. These object types let you select and edit geometry at the sub-object level.

If you have a primitive object and want to retain control of its creation parameters, you can apply a modifier such as Edit Mesh on page 1353, Edit Poly on page 1363, Edit Spline on page 1447, Edit Patch on page 1360, or Mesh Select on page 1527.

The Line Spline on page 620 and NURBS curves and surfaces are the exception: you can edit their sub-objects as soon as you create these kinds of objects.

 You choose a sub-object level in the stack display. Click the plus sign that appears next to the name of an object that has sub-objects. This expands the hierarchy, showing the available sub-object levels. Click a level to choose it. The name of the sub-object level highlights in yellow, and the icon for that

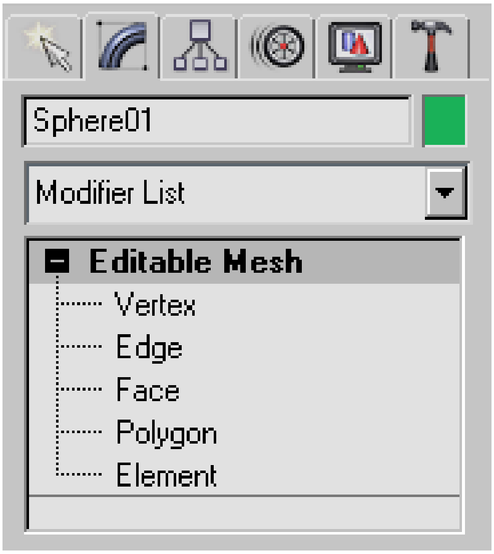
sub-object level appears to the right of both its name and the name of the top-level object.



Stack display shows the sub-object hierarchy, letting you choose a sub-object level.

Editing at the Sub-Object Level

When you edit an object at the sub-object level, you can select only components at that level, such as vertex, edge, face sub-objects, and so on. You can't deselect the current object, nor can you select other objects. To leave sub-object editing and return to object-level editing, click the top-level name of the object in the modifier stack, or click the highlighted sub-object level.



Click the top-level object name to exit sub-object editing.

TIP You can also access sub-object levels from the buttons at the top of the Selection rollout on the Modify panel.

Procedures

To make a sub-object selection:

These methods assume the object has sub-object levels. If the object has no sub-object levels (for example, a primitive such as a sphere), the + icon is not present. In that case, you need to collapse the object or apply an Edit modifier before you can edit its sub-object geometry.

1. Select the object you want to edit.
2. If the object doesn't already have sub-object levels, apply an Edit ... modifier such as Edit Mesh.
3.  Open the Modify panel.
4.  On the modifier stack display, click the + icon to expand the object's hierarchy.

5 On the stack display, click a sub-object level such as Vertex, Edge, or Face.

TIP For some kinds of objects, such as editable meshes, shaded viewports don't display sub-object selections. If this is the case, right-click the viewport label and choose Wireframe or Edged Faces view.

TIP For a detailed selection, you might want to zoom in on the object.

---

6 Click one of the toolbar selection buttons, and then use the same selection methods you'd use on objects to select the sub-object components. Or from the quad menu > Transform quadrant, choose one of the selection methods and select the sub-object components.

There are two alternative ways to go to a sub-object level:

-  Select the object and go to the Modify panel. Then right-click the object, and use the quad menu > Tools 1 (upper-left) quadrant > Sub-objects submenu.
-  Choose the selection level using buttons on the Modify panel's Selection rollout, if one is present for the type of object you're editing.

TIP Once you're at a sub-object level, the Insert key cycles through the levels of other kinds of sub-objects.

To exit a sub-object level, do one of the following:

- In the stack display, click the highlighted sub-object name or the top-level name of the object.
- If the object has a Selection rollout, click the button of the active sub-object level to turn it off.
- Right-click the object, and then in the Tools 1 (upper-left) quadrant of the quad menu, choose Top-level.
- Access a different command panel. This turns off sub-object editing.

If you think you've turned off sub-object editing but top-level object selection is still not restored, it might be due to the following reasons:

-  Your selection is locked. Click the Lock Selection Set button on the prompt line to turn it off.
- You've set the Selection Filter on page 208 on the main toolbar to a specific category of object, so you can't select any of the other categories. To fix this, select All in the Selection Filter list.

Selection Commands

Selection commands appear on the quad menu, on the main toolbar, on the Edit menu, and on the status bar.

The simplest method of selection is to turn on Select Object mode on page 227, and then click an object in a viewport (or drag to surround the object). While the method is simple, it is not effective for selecting multiple objects, especially in a crowded scene. Other tools let you select objects by name, filter out the kinds of objects you want to select, and to create named selection sets you can select repeatedly.

See also:

- Introducing Object Selection on page 192
- Basics of Selecting Objects on page 199
- Isolate Selection on page 219
- Scene Explorer on page 7379
- Select From Scene on page 228

Selection Commands on the Main Toolbar

The following selection commands appear by default on the Main toolbar.

Select Object on page 227

Select From Scene on page 228

Rectangular Selection Region on page 252

Circular Selection Region on page 253

Fence Selection Region on page 254

Lasso Selection Region on page 255

Paint Selection Region on page 256

Selection Filter List on page 235

Window/Crossing Selection Toggle on page 261

Named Selection Sets on page 239

The Window/Crossing toggle determines how the region selection options (on the toolbars) behave.

Selection Commands on the Edit Menu

The Edit menu contains selection commands that operate globally on your objects.

Edit menu selection commands include:

Select All on page 249

Select None on page 249

Select Invert on page 250

Select Similar on page 250

Select By > Color on page 251

Select By > Name on page 251 (also a toolbar button)

Select By > Layer on page 252

Selection Region > Rectangular Region on page 252

Selection Region > Circular Region on page 253

Selection Region > Fence Region on page 254

Selection Region > Lasso Region on page 255

Selection Region > Paint Selection Region on page 256

Selection Region > Window on page 258 (also a toolbar button)

Selection Region > Crossing on page 260 (also a toolbar button)

Edit Named Selection Sets on page 241

Selection Command on the Status Bar

The Selection Lock Toggle on page 7539 is located on the status bar. Locking a selection is useful when you are doing a lot of editing on a selection, and don't want to select something else by mistake.

Select Object 

Main toolbar > Select Object

Right-click to open quad menu. > Transform quadrant > Select

Select Object lets you select an objects and sub-objects for manipulation.

Object selection is affected by several other controls:

- The active Selection Region type: Rectangular on page 252, Circular on page 253, Fence on page 254, Lasso on page 255, or Paint on page 256.
- The active selection filter on page 235 (All, Geometry, Shapes, Lights, and so forth).
- The state of the crossing selection tool (which determines whether completely surrounded objects or surrounded and crossing objects are selected).

You can also select objects by name with the Select From Scene dialogSelect From Scene on page 228 list; press the H key to access the dialog.

A number of objects selected together is called a selection set on page 206. You can name selection sets in the Named Selection Sets field on the main toolbar and then recall them for later use.

NOTE The Smart Select command activates the Select Object function and, with repeated invocations, cycles through the available Selection Region methods. By default, Smart Select is assigned to the Q key; you can use Customize User Interface on page 7697 to assign it to a different keyboard shortcut, a menu, etc.

Procedures

To add or remove individual objects from a selection set:

1. Hold down the Ctrl key and select the objects to add or remove.

2 Hold down the Alt key and select objects to remove from the current selection set.

NOTE Adding and removing objects doesn't change a named selection set.

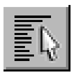
To toggle the selected/deselected state of multiple objects in the selection set:

- Hold down the Shift key and drag to region-select the objects to toggle.

To select objects *and* move, rotate, or scale them:

-  Use Select And Move, Select And Rotate, or Select And Scale, available from the main toolbar and the quad menu > Transform quadrant.
  When you rotate a selection set, the pivot of rotation depends on which option is selected on the Use Center flyout on page 975 on the toolbar.

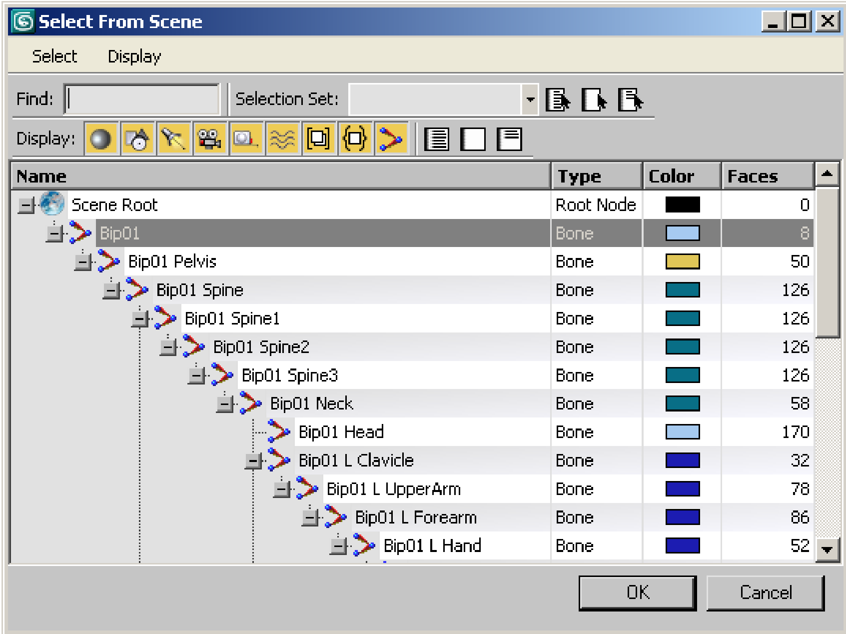
  These tools are restricted to a specific axis or plane, which you can choose from the Axis Constraints toolbar on page 7503 or specify with the transform gizmo on page 934.

Select From Scene 

main toolbar > Select By Name

Keyboard > H

Edit menu > Select By > Name



This dialog, named Select From Scene or Select Objects in most contexts, lets you select or designate objects by choosing them from a list of all objects currently in the scene. Select From Scene is a modal on page 8045, read-only version of Scene Explorer on page 7379; you can't use it to change object properties such as name and color. Other differences between Select From Scene and Scene Explorer include:

- No hierarchy manipulation; you can't link or unlink objects.
- Hidden and frozen objects don't appear in the list.
- Because the dialog is modal, you must close it before continuing.
- To select an object and close the dialog, double-click the object's list entry.
- All toggle settings such as Select Dependents, Display > Children, and the Display buttons persist. That means they survive Reset operations and even quitting and restarting the software. This also applies to the position and size of the dialog. To return all dialog settings to their defaults, delete this file: *[program folder]\plugcfg\DefaultModalSceneExplorer.ini.*

NOTE The Select From Scene dialog name and functionality are context dependent. When a transform such as Select And Move is active, the dialog lets you choose from all objects in the scene. But when certain modes are active, the choices in the dialog are more limited. For example, when Select and Link on page 3343 is active, the dialog is entitled Select Parent, and shows linkable objects but not the child object already selected. Similarly, if Group > Attach is active, the dialog is named Attach To Group and lists groups but not solitary objects.

TIP If you prefer to use the legacy Select Objects dialog instead of Select From Scene, it's available as an option. Open the *CurrentDefaults.ini* file (see Market-Specific Defaults on page 7694), find the [Scene Explorer] section, and change SelectByNameUsesSceneExplorer setting. If set to 1, then Select By Name and related commands use the Select From Scene dialog. If set to 0, thenSelect By Name and related commands use the legacy Select Objects dialog. The latter's functionality is essentially the same as the Selection Floater on page 231, except that it's modal, not modeless.

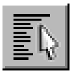
See also:

- Selection Floater on page 231

Procedures

To select objects by name:

1 Do one of the following:

-  Click the Select By Name button on the main toolbar.
- Choose Edit menu > Select By > Name.
- Press H.
  The Select From Scene dialog opens. By default, it lists all objects in the scene, displaying any hierarchies as collapsible branches. Currently selected objects are highlighted in the list.

2 Choose one or more objects in the list by doing one of the following:

- To select a single object and close the dialog, double-click the object name.
- Drag, or click and then Shift+click to select a contiguous range of objects and Ctrl+click to select noncontiguous objects.

- In the field above the list, type a search phrase. As you type, all matches for the current phrase are highlighted in the list. To highlight only objects whose case matches the search phrase exactly, turn on Find Case Sensitive (from the Select menu) .

NOTE In some cases, such as when linking objects, you can select only one object.

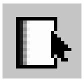
---

3 Click Select.

The selection is made as the dialog closes.

To highlight a single item from among multiple highlighted items:

Clicking one list item among several highlighted items does not unhlighlight the rest. When several items are highlighted, but you want to highlight only one of them, do either of the following:

- If any items are not highlighted, click one of them to remove highlighting from the rest, and then highlight the one you want.
-  If all items are highlighted, the preceding method isn't practical. In that case, on the upper toolbar click Select None, and then highlight the one you want.

Selection Floater

[Available only as a CUI action]

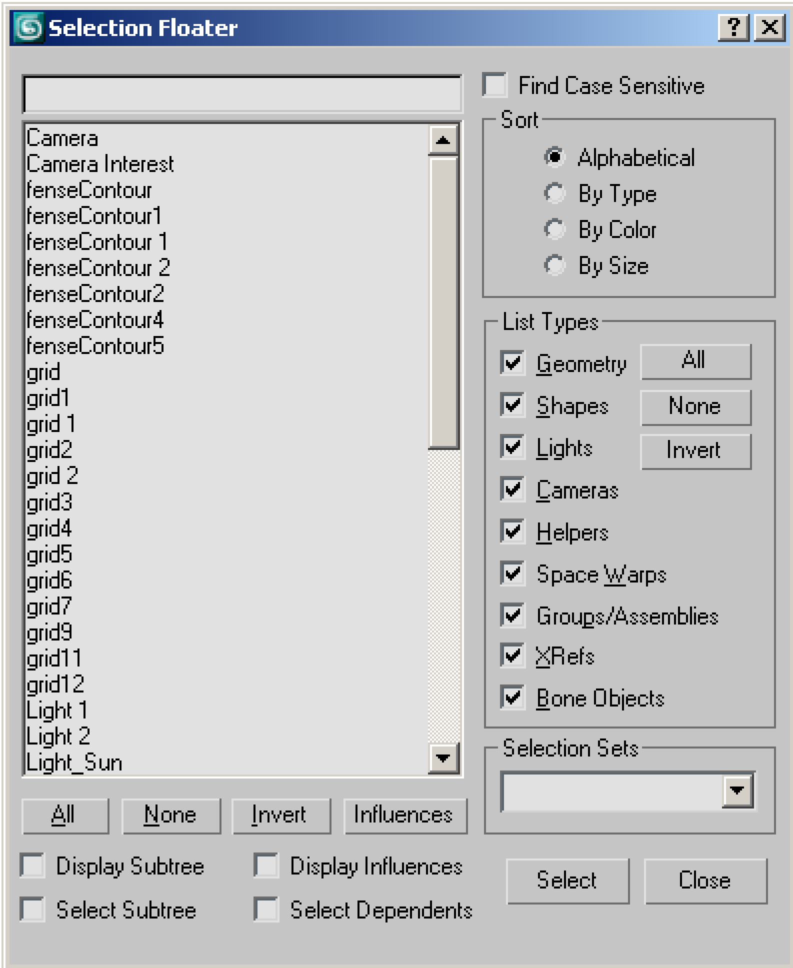
This modeless dialog lets you select objects in the scene. You can keep the dialog open while you work in your scene, making it easier to select objects.

IMPORTANT The Selection Floater command is available only as a Customize User Interface on page 7697 action; to use it you must first add it explicitly to the user interface.

See also:

- Scene Explorer on page 7379

Interface



[select objects field] Enter a name to highlight objects in the list whose names begin with the text you specify.

Find Case Sensitive When on, the select objects field above the list is case-sensitive. For example, if the list contains objects named **apple** and **Apple** and Find Case Sensitive is on, typing "a" will highlight only the **apple** entry. Also, sorts the list so uppercase names come before lowercase.

[objects list] Lists objects according to the current Sort and List Types choices. Does not display hidden and frozen objects.

To highlight an object name in the list, click with the mouse. To highlight multiple object names, drag, or click and then Ctrl+click or Shift+click, or use the search field above the list. To select highlighted objects, click the Select button.

Alternatively, you can highlight and select a single object in the list by double-clicking its name.

After selecting objects, the dialog remains open until you close it explicitly.

All/None/Invert These buttons alter the pattern of selection in the list window.

Influences When you highlight an object in the list window and then click the Influences button, the selected object's influences on page 8012 are highlighted as well.

Display Subtree Displays each item in the list so that its hierarchical branch on page 8002 is included (for example, Thigh/Shin/Foot). Hierarchical branches are indented.

Display Influences When this is on and you select an item in the list window, all of its influences are shown in blue. If you want to highlight these influences, click Influences.

Select Subtree When this is on and you select an item in the list window, all of its hierarchical children are selected as well.

Select Dependents When this is on and you select an item in the list window, all of its dependent on page 7951 objects are selected as well.Dependents include instances, references, and objects sharing a common modifier (the same objects that appear green when Show Dependencies is on in the View menu).

When both Select Subtree and Select Dependents are on, the subtree of any newly selected node is first selected, and then the dependents are selected. In other words, dependents of the subtree are selected, but not the subtrees of all dependents.

Sort group

Lets you choose the sort order of the items displayed in the list. This option is unavailable when Display Subtree is on; in that case, sorting is always alphabetical.

- **Alphabetical**
  Sorts from numeric characters at the top, then A to Z at the bottom. When Find Case Sensitive is on, all upper-case names come before lower-case names.

- **By Type**
  Sorts by category, using the same order as the check boxes in the List Types group.

- **By Color**
  Sorts by object wireframe color. The sorting order is arbitrary; the value of this option is that objects of the same color are grouped together.

- **By Size**
  Sorts based on the number of faces in each object. The object with the least number of faces is listed first, followed by objects with successively greater number of faces.

List Types group

Determines the types of objects to display in the list.

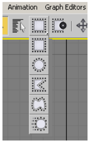
All/None/Invert These buttons alter the pattern of activation of the List Types options.

Selection Sets group

Lists any named selection sets on page 239 that you have defined in the scene. When you choose a selection set from the drop-down list, 3ds Max highlights its component objects in the main list.

Selection Region Flyout

Main toolbar > Selection Region flyout



Selection Region flyout

The Selection Region flyout provides access to five methods you can use to select objects by region. Clicking the Selection Region button displays a flyout containing the Rectangle on page 252, Circular on page 253, Fence on page 254, Lasso on page 255, and Paint on page 256 Selection Region buttons.

For the first four methods, you can select either objects that are completely within the selection region (window method), or objects that are within or touched by the selection shape (crossing method). Toggle between the window and crossing selection methods by using the Window/Crossing Selection button on page 260 on the main toolbar.

NOTE If you hold down Ctrl while specifying a region, the affected objects are added to the current selection. Conversely, if you hold down Alt while specifying a region, the affected objects are removed from the current selection.

NOTE The Smart Select command activates the Select Object on page 227 function and, with repeated invocations, cycles through the available Selection Region methods. By default, Smart Select is assigned to the Q key; you can use Customize User Interface on page 7697 to assign it to a different keyboard shortcut, a menu, etc.

Procedures

To select using a region (general method):

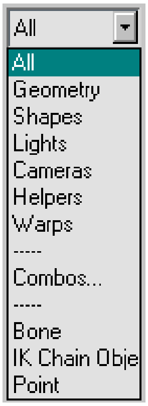
1. Choose a Selection Region method from the flyout.
2. Drag in a viewport, then release the mouse. The first location you click is one corner of the rectangle, and where you release the mouse is the opposite corner.

   IMPORTANT If you're using Select Object on page 227, you can start dragging anywhere to select a region: on an object or off. However, if you're using one of the transform tools, such as Select and Move on page 959, start the drag operation away from an object; that is, in an empty part of the viewport. Otherwise, if you start dragging on an object, most likely the software will assume you intend to select where you click and will begin the transform operation immediately.

   To cancel the selection, right-click before you release the mouse.

Selection Filter List

Main toolbar > Selection Filter



The Selection Filter list lets you restrict to specific types and combinations of objects that can be selected by the selection tools. For example, if you choose Cameras, you can select only cameras with the selection tools. Other objects do not respond. When you need to select objects of a certain type, this is useful as a quick method of freezing all other objects.

Use the drop-down list to select a single filter. Choose Combos from the drop-down list to use multiple filters from the Filter Combinations dialog on page 236.

Filter Combinations Dialog

Main toolbar > Selection Filter list > Combos > Filter Combinations dialog

Use the Filter Combinations dialog to create your own custom combinations of categories to add to the Selection Filters list on page 235.

You can also add specific types of objects, or Class IDs, to the list. For example, you can set a filter that lets you select only Sphere primitives.

Procedures

To create a combination filter:

1. Open the Selection Filter list and choose Combos.

   The Filter Combinations dialog appears.

2. Turn on one or more of the check boxes in the Create Combination group.

3 Click the Add button.

The specified combination appears in the Current Combinations list to the right as a combination of the first letters of each selected category.

4 Click OK.

The new combo item appears at the bottom of the Select Filter list.

Combos are stored in the *3dsmax.ini* *on page 83* file, so they remain in effect for all scenes through all sessions.

To delete a combination filter:

1 Open the Selection Filter list and choose Combos.

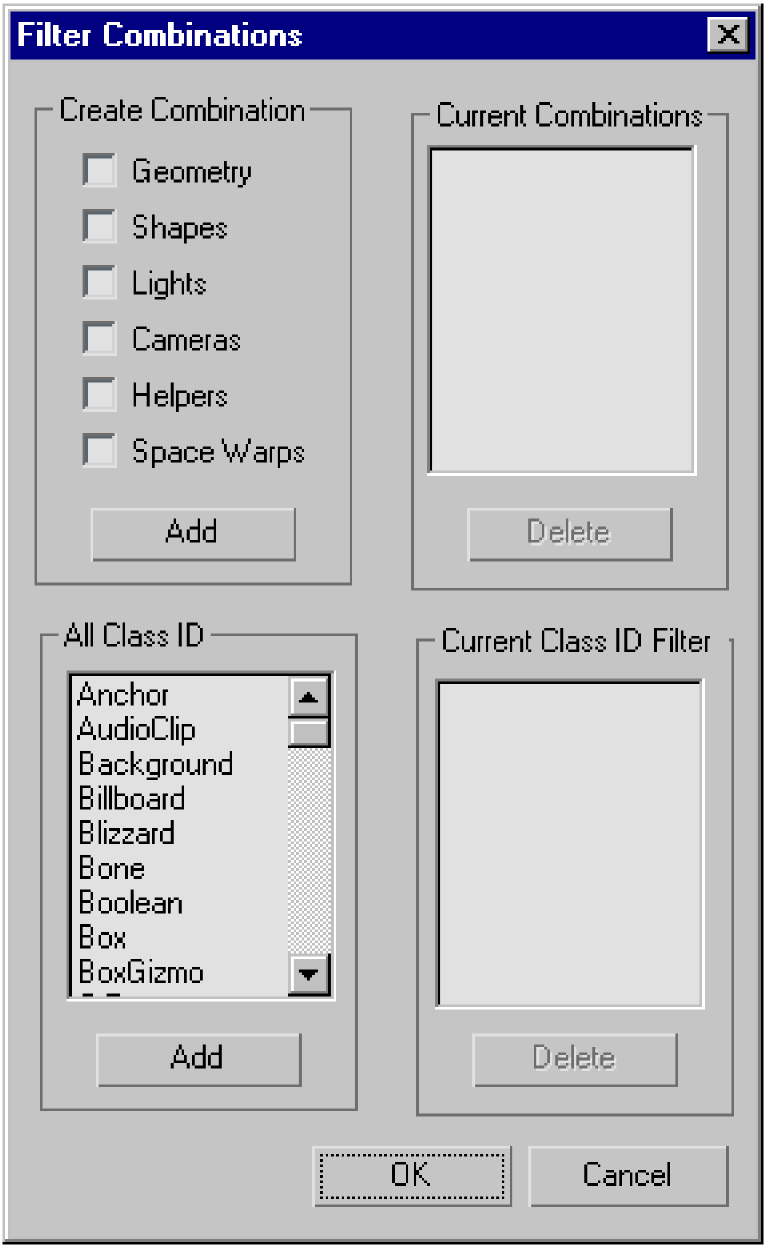
The Filter Combinations dialog appears.

2 Choose one or more of the combos in the Current Combinations list.

3 Click the Delete button.

4 Click OK.

Interface


Filter Combinations
Create Combination
Geometry
Shapes
Lights
Cameras
Helpers
Space Warps
Add
Current Combinations
Delete
All Class ID
Anchor
AudioClip
Background
Billboard
Blizzard
Bone
Boolean
Box
BoxGizmo
Add
Current Class ID Filter
Delete
OK
Cancel

Create Combination group

Geometry, Shapes, Lights, Cameras, Helpers, Space Warps Choose the category or categories you want included in the combination.

Add After choosing the categories to include in a combination, click this button to place the categories, labeled with the categories' initials, in the Current Combinations list, as well as at the bottom of the Selection Filter list.

Current Combinations group

Current Combinations list Lists current combinations. To delete one or more combinations, choose them, and then click Delete.

Delete After choosing one or more combinations in the Current Combinations list, click this button to delete them.

All Class ID group

Class ID list Lists all the available categories that can be added to custom filters for display and selection. Highlight a category to add, then click Add.

Add After choosing a class to include in the filter list, click this button to place the class in the Current Class ID Filter list, as well as at the bottom of the Selection Filter list.

Current Class ID Filter group

Class ID list Lists current classes to filter. To delete a class, choose it, and then click Delete.

Delete After choosing a class in the Current Class ID Filter list, click this button to delete the class.

Named Selection Sets

Main toolbar > Named Selection Sets

The Named Selection Sets list allows you to name a selection set and recall the selection for later use. It supports selection sets both at the object level and at sub-object levels. You edit named object-level selection sets with the

Named Selections Sets dialog on page 241 and sub-object level sets with the Edit Named Selections dialog on page 246.

A named selection set is removed from the list if all of its objects have been deleted from the scene, or if all of its objects have been removed from the named set in the Named Selections Sets dialog.

Selection set names are case sensitive at both the object level and at sub-object levels.

You can transfer sub-object named selections from one level in the stack to another. The Copy and Paste buttons let you copy named selections from one modifier to another.

While at a specific sub-object level, such as Vertex, you can make selections and name those selections in the Named Selection Sets field of the toolbar. The named sets are specific to both the selection level and the level on the stack.

Keep in mind the following restrictions:

- You can transfer named selections only between the same type of sub-object level. In other words, you can transfer named selections from vertex sub-object to another vertex sub-object, but you can't transfer it to face or edge sub-object level.
- You must transfer the selection between modifiers that handle like geometry. You can copy and paste between an editable mesh and a mesh select modifier, but you can't copy and paste between a mesh select modifier and an editable spline.
- You can copy and paste between two modifiers in two different objects, as long as you're at the same level and both modifiers handle the same type of geometry.
- If you change the topology of a mesh after creating a named selection (such as deleting some vertices), the named selections will probably no longer select the same geometry.

Procedures

To create a named selection set:

1. Select the objects you want to be in a set.
2. Type the name of the set in the Named Selection Set field and press Enter.

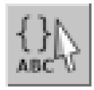
3 Whenever you want to access the selection, choose its name from the Named Selection Sets list.

To select a named selection set, do one of the following:

1 To select a single item, click it in the list.

2 To select more than one item in the list, select one, and then select others while holding down the Ctrl key.

3 To deselect single items after you've selected multiple items, hold down the Alt key.

Named Selection Sets Dialog 

Edit menu > Edit Named Selections

Main toolbar > Named Selection Sets

The Named Selection Sets dialog, available from the Edit menu, is a modeless dialog on page 8045 that lets you create named selection sets or select objects to add to (or remove from) a selection set, directly from the viewport. The dialog also lets you organize your current named selection sets, browse their members, delete or create new sets, or identify which named selection sets a particular object belongs to.

NOTE This dialog applies to *objects* only. For editing *sub-object* named selection sets, see Edit Named Selections Dialog on page 246.

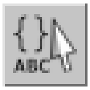
See also:

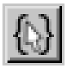
- Named Selection Sets on page 239
- Using Named Selection Sets on page 206
- Edit Named Selections Dialog on page 246

Procedures

To create a named selection:

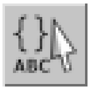
1 In the viewport, select the objects you want to gather as a selection set.

2  Click the main toolbar Edit Named Selection Sets button or choose Edit > Edit Named Selections.

3  On the Named Selection Sets dialog, click Create New Set.

4 Enter a name for the new selection set.

To add objects to a named selection set:

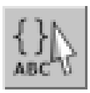
1  Click the toolbar Named Selection Sets button or choose Edit > Edit Named Selections.

2 Choose the named selection set in the dialog.

3 Select one or more objects in the viewport.

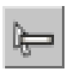
4  In the dialog, click Add Selected Objects.

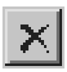
To remove objects from a named selection set:

1  Click the toolbar Named Selection Sets button or choose Edit > Edit Named Selections.

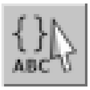
2 Choose the named selection set in the dialog.

3 In the viewport, select the objects you want to remove.

4  In the dialog, click Subtract Selected Objects.

 Note: You can also remove objects by selecting them in the Named Selection Sets dialog, then clicking Remove or pressing Delete.

To move an object from one set to another:

1  Click the toolbar Named Selection Sets button or choose Edit > Edit Named Selections.

2 In the Named Selection Sets dialog, expand the selection sets.

3 Drag an object from one set to another.

   The object is moved into the second set. If you use Ctrl+drag, the object will be *copied* into the second set.

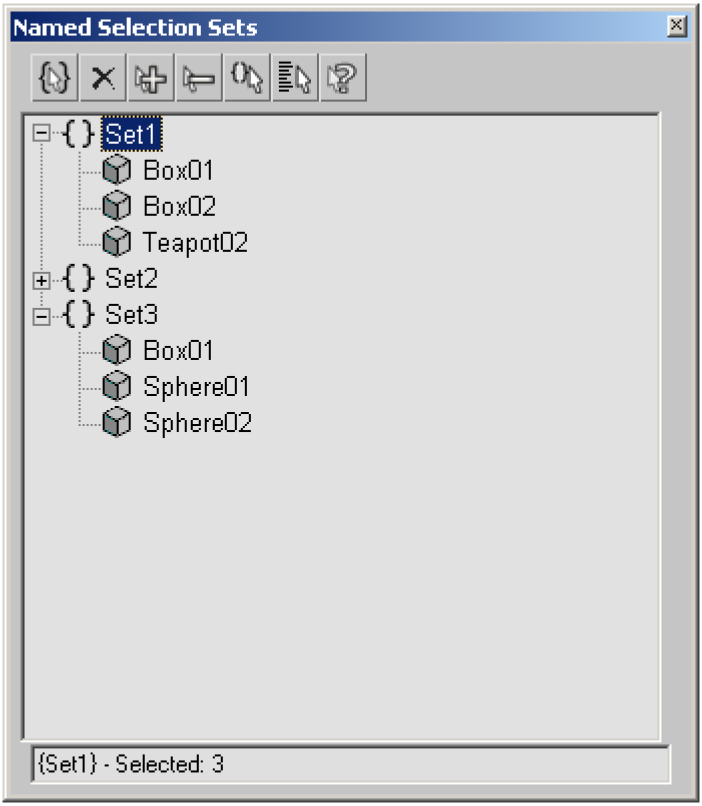
   TIP You can also copy the contents of an entire set into another, by dragging them into the desired selection set.

To select objects in a set:

1 Highlight the set in the Named Selection Sets dialog.

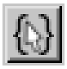
2  Click Select Objects In Set to select all of the objects in the highlighted set.

Interface

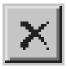


In the Named Selection Sets dialog, all of the current named selection sets are displayed. By clicking the plus (+) or minus (-) icon, you can expand or collapse (respectively) the object list for each set.

The buttons along the top of the dialog let you create or delete sets, add or remove objects from a set, select objects (independently or as a selection set), and see which named selection set(s) a particular object belongs to.

 **Create New Set** Creates a new selection set, including any currently selected objects as members.

NOTE If no objects are selected, an empty set is created.

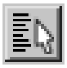
 **Remove** Removes the selected object or selection set.

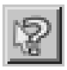
NOTE Deleting an object or its selection set does not delete the object; it only destroys the named set.

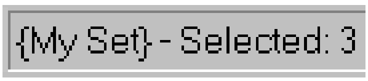
 **Add Selected Objects** Adds the currently selected objects to the selected named selection set.

 **Subtract Selected Objects** Removes currently selected objects from the selected named selection set.

 **Select Objects in Set** Selects all members of the current named selection.

 **Select Objects by Name** Opens the Select Objects dialog on page 228, where you can select a group of objects. The selected objects can then be added to or removed from any named selection set.

 **Highlight Selected Objects** Highlights all of the named selection sets that contain the current scene selection.



Status Bar Displays the current named selection set, as well as what's currently selected in the scene. If more than one object is selected, the number of selected objects is displayed.

Right-click menu

Additional commands are available when you right-click in the Named Selection Sets dialog.

Rename Lets you rename the selected set or object.

TIP You can rename objects or sets by pressing F2.

Cut Removes the selected object or set and stores it in a buffer for reuse with the Paste command, similar to the Cut command in Windows.

TIP You can cut an object or set by pressing Ctrl+X.

Copy Copies the selected object or set and stores it in a buffer for reuse with the Paste command, similar to the Copy command in Windows.

TIP You can copy an object or set by pressing Ctrl+C.

Paste Adds a Cut or Copied object or set into another set.

TIP You can paste an object or set by pressing Ctrl+V.

Collapse All Collapses all expanded selection sets.

Expand All Expands all collapsed selection sets.

Create New Set Creates a new selection set, including any currently selected objects as members.

Remove Removes the selected object or selection set.

Add Selected Objects Adds currently selected objects to the selected named selection set.

Subtract Selected Objects Removes currently selected objects from the selected named selection set.

Select Objects in Set Selects all members of the current named selection.

Select Objects by Name Opens the Select Objects dialog on page 228, and adds all objects selected there to the current named selection set.

Highlight Selected Objects Highlights all of the named selection sets that contain the current scene selection.

Find Next Toggles through selection sets containing the selected object, when used in collaboration with the Highlight Selected Objects command.

TIP You can use Ctrl+G to toggle through the sets.

Edit Named Selections Dialog

Make a sub-object selection. > Edit menu > Edit Named Selections

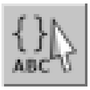
Make a sub-object selection. > Main toolbar > Named Selection Sets

Edit Named Selections displays the Edit Named Selections dialog, letting you manage named selection sets of *sub-objects on page 220*. Unlike the Named Selection Sets dialog on page 241, which applies to *objects only*, it is a modal dialog, which means that you must close it in order to work in other areas of 3ds Max. Also, you can work only with existing named sub-object selections; you cannot use the dialog to create new selections.

Procedures

To edit named sub-object selections:

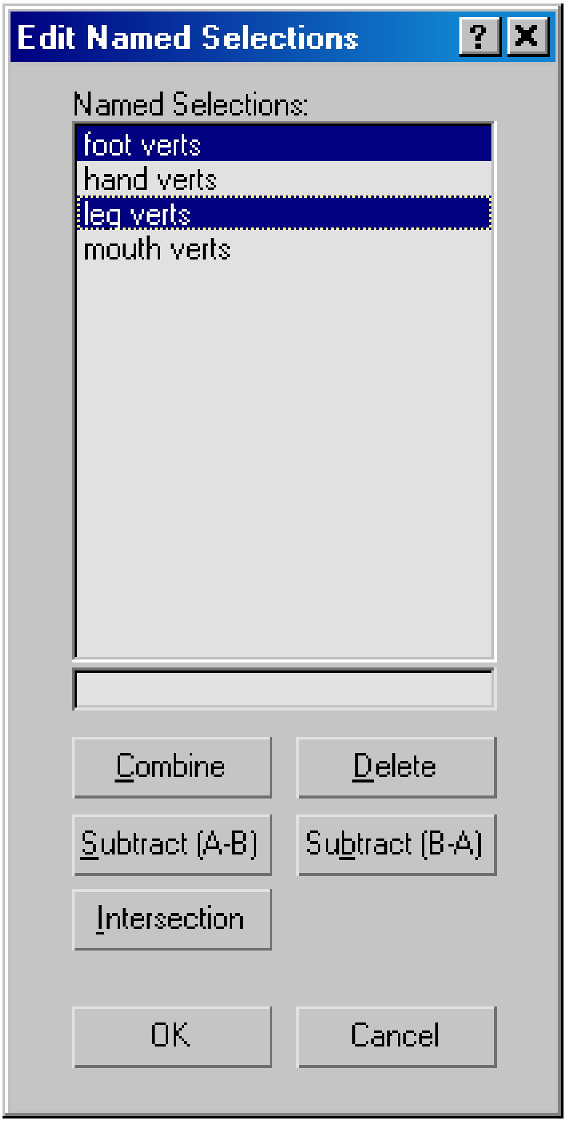
1 At a sub-object level, create one or more named selection sets on page 239.

2  Click the toolbar Named Selection Sets button or choose Edit > Edit Named Selections.

The Edit Named Selections dialog opens, listing all named selection sets for the current sub-object level.

3 Use the dialog controls to edit the named selection sets.

Interface



The dialog window lists all named selections at the current sub-object level. The buttons beneath the windows let you delete, merge, and edit the listed items. Use standard mouse-plus-keyboard methods (using Ctrl or Shift) to highlight list items and designate them for subsequent operations.

To rename a set, click it in the list, and then edit its name in the one-line window immediately below the list.

Combine Merges all objects from the highlighted selection sets into a single, new selection set. Select two or more selection sets, and then click Combine and enter a new name for the selection set. Use Delete to delete the original sets.

Delete Deletes all highlighted items from the Named Selections window. This affects only selection sets, not the sub-objects they refer to.

Subtract (A-B) Removes the sub-objects contained in one selection set from another. Select one item in the Named Selections window, and then select the other. The top highlighted item in the window is operand A, and the bottom is operand B (regardless of the order of their selection). Click Subtract (A-B) to subtract the sub-objects in the bottom item from those in the top item. There must be some overlap between the two selection sets for this command to have any effect.

Subtract (B-A) Subtracts the sub-objects in the top selected item from those in the bottom item.

Intersection Creates a selection set that consists only of sub-objects that all highlighted selection sets have in common. Highlight two or more items in the Named Selections window, and then click Intersection. In the dialog that appears, enter a new set name and click OK.

Select All

Edit menu > Select All

Keyboard > Ctrl+A

This command selects all objects in the scene matching the current selection filter type on page 235 on the main toolbar.

Select None

Edit menu > Select None

Keyboard > Ctrl+D

This command deselects all objects in the scene conforming to the current selection filter type on page 235 on the main toolbar.

Select Invert

Edit menu > Select Invert

Keyboard > Ctrl+I

This command inverts the current selection set. All objects not currently selected are selected, and all objects currently selected are deselected, respecting the current selection filter type on page 235 on the main toolbar.

Select Similar

Select one or more objects. > Edit menu > Select Similar

Select one or more objects. > Right-click > quad menu > Transform quadrant > Select Similar

Select one or more objects. > Ctrl+Q

Select Similar automatically selects all items that are "similar" to the current selection. In general, this means that the objects must be on the same layer, and have the same material (or no material) applied. The specific functionality depends on the objects' source:

- Selects all items imported in the DWG format in the selected object's or objects' layer that have the same style(s) on page 7054 or categories as defined in AutoCAD Architecture (formerly known as ADT, or Architectural Desktop), or the same families and types as defined in Revit. For example, if you've imported or linked to a DWG file that contains walls in several different styles, such as CMU-8, Concrete-8, and Stud-4, you could select all CMU-8 wall segments in the same layer by selecting one and then invoking Select Similar.

  If you start by selecting multiple objects with different styles, Select Similar will select all objects with those styles.

- This command also applies to objects native to 3ds Max. It selects all objects of the same type. This includes primitives and editable object types. For example, if you add some boxes and cylinders, select one of the cylinders, and then invoke Select Similar, all of the cylinders will be selected, but not the boxes. If you then convert all of the objects to Editable Poly format and repeat the test, all of the objects will be selected. If you then apply a material to one of the objects or move it to a different layer, it's no longer "similar" to the rest and won't be selected by Select Similar.

Select By

Edit menu > Select By

The Select By submenu on the Edit menu provides commands for selecting objects in the scene by color, name, and other characteristics. It also gives quick access to the various Region selection options.

Name on page 251

Layer on page 252

Color on page 251

Select By Color

Edit menu > Select By > Color

Select By Color lets you select all objects having the same color as the selected object. Selection is made by wireframe color (see Object Color Dialog on page 387), rather than by any materials associated with the objects.

After you choose this command, click any object in the scene to determine the color for the selection set.

TIP To select objects by material, use Schematic View on page 7411.

Select By Name (Edit Menu)

Edit menu > Select By > Name

Keyboard > H

Select By Name lets you select objects by choosing them from a list of all objects in the scene.

For a full description of the Select By Name function, see Select From Scene on page 228.

TIP To select objects by material, use Select By Material on page 5340.

Select By Layer

Edit menu > Select By > Select By Layer

Select By Layer lets you select all objects in one or more layers by picking them from a list of all layers in the scene. Choosing this command opens the Select By Layer dialog; use standard methods to highlight one or more layers, and then click OK. The dialog closes, and all objects in the highlighted layer or layers are selected.

Region Selection

Selection Region

Edit Menu > Selection Region

The Selection Region submenu on the Edit menu provides quick access to the various Region selection on page 203 options.

Rectangular Selection Region on page 252

Circular Selection Region on page 253

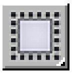
Fence Selection Region on page 254

Lasso Selection Region on page 255

Paint Selection Region on page 256

Window on page 258

Crossing on page 260

Rectangular Selection Region 

main toolbar > Rectangular Selection Region (Selection Region flyout)

Edit menu > Selection Region > Rectangular Region

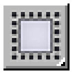
The Rectangular Selection Region option, available from the Selection Region flyout on page 234 and the Edit menu, provides one of five methods you can use to select objects by region. The other methods are Circular on page 253, Fence on page 254, Lasso on page 255, and Paint on page 256.

You can use Rectangular to select either objects that are completely within the selection region (window method), or objects that are either within or touched by the selection shape (crossing method). Toggle between the window and crossing selection methods by using the Window/Crossing Selection button on page 260 on the main toolbar.

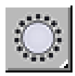
NOTE If you hold down Ctrl while specifying a region, the affected objects are added to the current selection. Conversely, if you hold down Alt while specifying a region, the affected objects are removed from the current selection.

Procedures

To select using a rectangle:

1.  Click the Rectangular Selection Region button.
2. Drag in a viewport, then release the mouse. The first location you click is one corner of the rectangle, and where you release the mouse is the opposite corner.

   To cancel the selection, right-click before you release the mouse.

Circular Selection Region 

main toolbar > Circular Selection Region (Selection Region flyout)

Edit menu > Selection Region > Circular Region

The Circular Selection Region option, available from the Selection Region flyout on page 234 and the Edit menu, provides one of five methods you can use to select objects by region. The other methods are Rectangular on page 252, Fence on page 254, Lasso on page 255, and Paint on page 256.

You can use Circular to select either objects that are completely within the selection region (window method), or objects that are either within or touched by the selection shape (crossing method). Toggle between the window and crossing selection methods by using the Crossing Selection button on page 260 on the main toolbar.

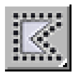
NOTE If you hold down Ctrl while specifying a region, the affected objects are added to the current selection. Conversely, if you hold down Alt while specifying a region, the affected objects are removed from the current selection.

Procedures

To select using a circle:

1.  Click the Circular Selection Region button.
2. Drag in a viewport, then release the mouse. The first location you click is the center of the circle, where you release the mouse defines the circle's radius.

   To cancel the selection, right-click before you release the mouse.

Fence Selection Region 

Main toolbar > Fence Selection Region (Selection Region flyout)

Edit menu > Selection Region > Fence Region

The Fence Selection Region option, available from the Selection Region flyout on page 234 and the Edit menu, provides one of five methods you can use to select objects by region. The other methods are Rectangular on page 252, Circular on page 253, Lasso on page 255, and Paint on page 256.

You can use Fence to select either objects that are completely within the selection region (window method), or objects that are either within or touched by the selection shape (crossing method). Toggle between the window and crossing selection methods by using the Window/Crossing button on page 261 on the main toolbar.

NOTE If you hold down Ctrl while specifying a region, the affected objects are added to the current selection. Conversely, if you hold down Alt while specifying a region, the affected objects are removed from the current selection.

Procedures

To select using a fence:

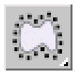
1.  Click the Fence Selection Region button.
2. Drag to draw the first segment of a polygon, then release the mouse button.

   A "rubber-band line" is now attached to the cursor, anchored at the point of release.
3. Move the mouse and click to define the next segment of the fence. You can make as many steps as you want.
4. To complete the fence, either click the first point, or double-click.

   A pair of cross hairs appears when you're near enough to click the first point. This creates a closed fence.

   Double-clicking creates an open fence, which can select objects only by the crossing method.

   To cancel the selection, right-click before you release the mouse.

Lasso Selection Region 

Main toolbar > Lasso Selection Region (Selection Region flyout)

Edit menu > Selection Region > Lasso Region

The Lasso Selection method lets you select multiple objects within a complex or irregular region with a single mouse action.

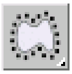
The Lasso Selection Region option, available from the Selection Region flyout on page 234 and the Edit menu, provides one of five methods you can use to select objects by region. The other methods are Rectangular on page 252, Circular on page 253, Fence on page 254, and Paint on page 256.

You can use Lasso to select either objects that are completely within the selection region (window method), or objects that are either within or touched by the selection shape (crossing method). Toggle between the window and crossing selection methods by using the Window/Crossing button on page 261 on the main toolbar.

NOTE If you hold down Ctrl while specifying a region, the affected objects are added to the current selection. Conversely, if you hold down Alt while specifying a region, the affected objects are removed from the current selection.

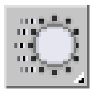
Procedures

To select using a lasso:

1  Click the Lasso Selection Region button.

2 Drag to draw a shape around the object(s) that should be selected, then release the mouse button.

NOTE To cancel the selection, right-click before you release the mouse.

Paint Selection Region 

main toolbar > Paint Selection Region (Selection Region flyout)

Edit menu > Selection Region > Paint Region

The Paint Selection method lets you select multiple objects or sub-objects by dragging the mouse over them. To change the brush size, right-click the Paint Selection Region button, and then, on the Preference Settings dialog > General tab > Scene Selection group, change the Paint Selection Brush Size value on page 7748.

If you hold down Ctrl while specifying a region, the affected objects are added to the current selection. Conversely, if you hold down Alt while specifying a region, the affected objects are removed from the current selection.

TIP You can also create custom tools for changing the brush size; choose Customize menu > Customize User Interface and set keyboard shortcuts or other user interface items for the actions Paint Selection Size Up and Paint Selection Size Down.

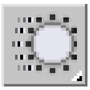
NOTE Paint Selection Region respects the Window/Crossing selection toggle on page 261 setting. If the toggle is set to Select Region Window on page 258 and the brush is smaller than an object or sub-object to be selected, you won't be able to select the item. To resolve this, enlarge the brush or choose Select Region Crossing on page 260.

NOTE With editable poly on page 2123 and Edit Poly on page 1363 objects, you can also paint soft selections on page 2015 and deformation on page 2128.

The Paint Selection Region button, available from the Selection Region flyout on page 234, provides one of five methods you can use to select objects by region. The other methods are Rectangular on page 252, Circular on page 253, Lasso on page 255, and Fence on page 254.

Procedures

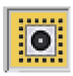
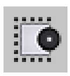
To select by painting a region:

1.  Choose Paint Selection Region from the flyout.
2. Drag over the object(s) to select, then release the mouse button. As you drag, a circle showing the brush radius appears attached to the mouse.

   NOTE To cancel the selection, right-click before you release the mouse.

3. To change the brush size, right-click the Paint Selection Region button, and then, on the Preference Settings dialog > General tab > Scene Selection group, change the Paint Selection Brush Size value on page 7748.

   You can also set keyboard shortcuts for changing the brush size. To do so, use the Paint Selection Size Up and Paint Selection Size Down action items. See Keyboard Panel on page 7698.

Region  

Edit menu > Region

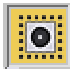
main toolbar > Window Selection or Crossing Selection

When dragging the mouse to select one or more objects, the Region options let you switch between selecting objects within, or crossed by, a window region that you draw with the mouse. Choose the appropriate Selection Region submenu command, or use the Window/Crossing Selection Toggle on page 261 on the main toolbar.

You can automatically switch between Window and Crossing Region Selection based on cursor movement direction. To set this up, choose Customize > Preferences and on the General tab in the Scene Selection group turn on Auto Window/Crossing Selection by Direction.

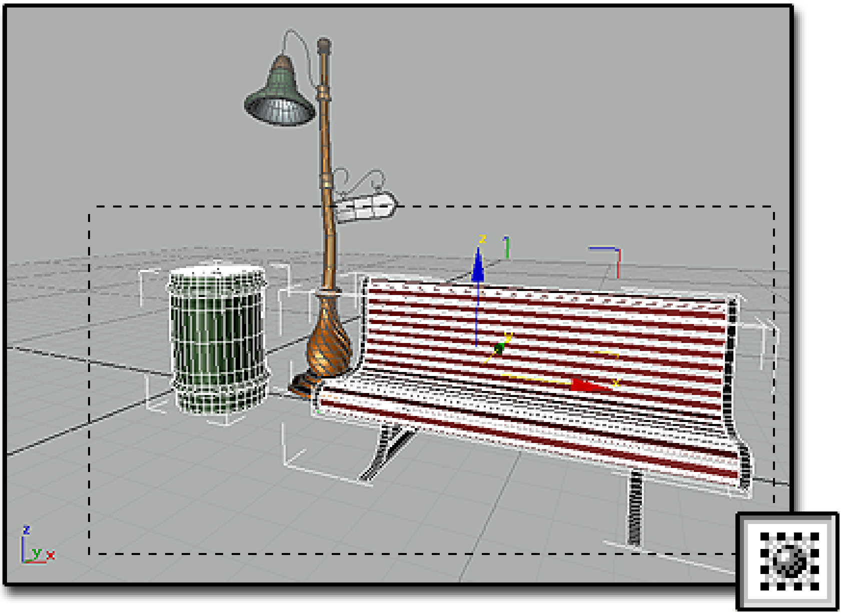
See also:

- Select Region Window on page 258
- Select Region Crossing on page 260

Select Region Window 

Edit menu > Selection Region > Window

main toolbar > Window/Crossing Selection



Select Region Window selects only those objects completely inside the window: the trash can and bench.

Select Region Window selects objects within a selection region on page 234.

After you choose this command, draw a selection region around any objects in the scene. Only those objects that are entirely inside the region boundary are selected.

Procedures

To select objects within a selection region:

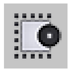
1 Do one of the following:

- Choose Edit > Selection Region > Window.
- Click the Window/Crossing Selection Toggle on page 261 to display the Window icon.

2 On the main toolbar, click the Selection Region flyout on page 234 and choose a method: Rectangular, Circular, Fence, or Lasso Selection region.

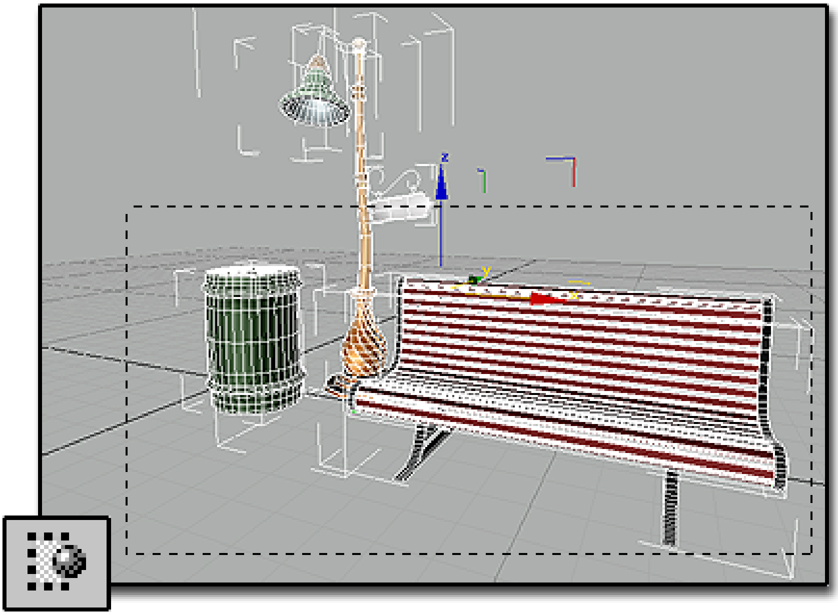
NOTE This setting also applies to Paint Selection Region, but in this case the boundary is that of the brush, not the region. In other words, when painting a selection region, the brush must completely encompass an object or sub-object to select it.

3 Drag to specify the region and select the objects.

Select Region Crossing 

Edit menu > Selection Region > Crossing

main toolbar > Crossing Selection



Select Region Crossing selects objects within the window and also objects it crosses: the trash can, bench, and streetlight.

Select Region Crossing selects objects within and crossed by a selection region on page 234 boundary.

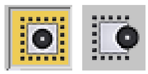
After you choose this command, draw a selection region around or crossing objects in the scene. Objects within the region boundary as well as those that intersect the boundary are selected.

Procedures

To select objects within and crossed by a selection region:

1. Do one of the following:
   - Choose Edit > Selection Region > Crossing.
   - Click the Window/Crossing Selection Toggle on page 261 to display the Crossing icon.
2. From the main toolbar, click the Selection Region flyout on page 234 and choose a method: Rectangular, Circular, Fence or Lasso Selection region.

   NOTE This setting also applies to Paint Selection Region, but in this case the boundary is that of the brush, not the region. In other words, when painting a region in Crossing mode, the brush selects every object or sub-object it touches or encompasses.
3. Drag to specify the region and select the objects.

Window/Crossing Selection Toggle 

main toolbar > Crossing Selection or Window Selection from the Window/Crossing toggle

Edit menu > Region > Window or Crossing

The Window/Crossing Selection toggle switches between window and crossing modes when you select by region.

- In Window mode on page 258, you select only the objects or sub-objects*within* the selection.
- In Crossing mode on page 260, you select all objects or sub-objects within the region, plus any objects or sub-objects *crossing* the boundaries of the region.

TIP If you're making sub-object selections of faces and you select more faces than you want, make sure you're in Window mode.

The Selection Region flyout on page 234 on the toolbar allows you to create different-shaped boundaries.

3ds Max automatically saves the Window/Crossing setting in the *3dsmax.ini on page 83* file.

Edit Commands

These commands on the Edit menu on page 7474 are for basic edit manipulations of selections.

Undo and Redo work as in standard Windows applications. These commands are available on the default main toolbar as well. 3ds Max also provides a history of commands. Right-clicking the Undo or Redo buttons displays a list of commands you can undo or redo. Not all operations are reversible using Undo.

NOTE Viewport changes such as panning and zooming have a separate Undo and Redo. See View-Handling Commands on page 146.

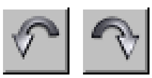
The Hold and Fetch command pair serves as an alternative to Undo and Redo. Hold saves the current state of the scene. After using Hold, you can restore that state at a later point by using Fetch. Sometimes, when you are about to perform a risky operation, an alert prompts you to first use Hold.

3ds Max does not have the Cut or Paste functions found in many Windows applications. The Delete command simply removes the selection from the scene.

Undo/Redo on page 262

Hold/Fetch on page 264

Delete on page 265

Undo/Redo 

Edit menu > Undo or Redo

Main toolbar > Undo or Redo

Keyboard > Ctrl+Z (Undo) or Ctrl+Y (Redo)

The Undo command reverses the last operation, including selection actions and those performed on selected objects. Redo reverses the last Undo operation.

Some actions cannot be undone: for example, applying the Collapse utility or Reset Transform utility, or saving a file, which overwrites the previous version. When you know an action cannot be undone, use Hold on page 264 first. Then if you want to undo it, use Fetch. Hold and Fetch are also commands on the Edit menu on page 7474.

Afer you perform an action that is undoable, the Undo command on the Edit menu shows the name of the function to be undone. After you undo an action, the Redo command Edit menu shows the name of the function you can redo.

Undo and Redo are also available as buttons on the main toolbar. You can right-click the Undo or Redo button to display a box that lists the last operations performed. You can highlight and reverse any number of these operations in sequence with the respective Undo or Redo command. By default, there are 20 levels of Undo. You can change the number of levels with the Customize > Preferences > General tab on page 7744 > Scene Undo group.

Undo/Redo and Object Creation

When you create an object, the Create operation is recorded by 3ds Max and displayed next to the activated Undo command in the Edit menu. If you undo the Create operation, the Create operation appears next to the activated Redo command in the Edit menu. The Undo and Redo commands in the Edit menu are unavailable when no valid operation was performed or recorded.

Procedures

To undo the most recent action:

Do one of the following:

-  Click Undo..
- Choose Edit menu > Undo.
- Press Ctrl+Z.

To undo several actions:

1.  Right-click Undo.
2. From the list, select the level where you want to return. You must choose a continuous selection; you can’t skip over items in the list.
3. Click the Undo button.

   To exit the list without performing an action, click the Cancel button, or click somewhere outside of the list.

To redo an action, do one of the following:

1.  Click Redo.
2. Edit menu > Redo.
3. Press Ctrl+Y.

To redo several actions:

1.  Right-click Redo.
2. From the list, click the action to return to.

   Your selection must be continuous: you can't skip over any items in the list.
3. Click the Redo button.

   To exit the list without performing an action, click the Cancel button or click somewhere outside of the list.

Hold/Fetch

Edit menu > Hold or Fetch

Hold saves the scene and its settings to a disk-based buffer. Fetch restores the contents of the buffer stored by the previous Hold command. The information stored includes geometry, lights, cameras, the viewport configuration, and selection sets.

Use Hold before you perform an operation that might not work as expected, that is new or unfamiliar to you, or that cannot be undone. If the results aren't as expected, you can use Fetch to return to the point where you chose Hold.

TIP Also use Save or Save As before you perform an operation that cannot be undone: for example, applying the Reset Transform utility.

If you experience an unexpected end of operation or crash after you perform Hold, you can retrieve your scene from the buffer with the Fetch command after you restart 3ds Max.

Additional Details

- The Hold buffer is a temporary file (*maxhold.mx*) in the directory specified by the AutoBackup path on the Configure User Paths dialog > File I/O panel on page 7733.
- Fetch also deletes all operations recorded in the Undo and Redo History lists.

Delete

Make a selection. > Edit menu > Delete

Make a selection. > Keyboard > Delete

The Delete command deletes the current selection from the model.

The Undo command on page 262 can restore the deleted selection to the model. (The Undo command is also available as a button on the main toolbar.)

NOTE Actively file-linked objects on page 6987 cannot be deleted.

Groups and Assemblies

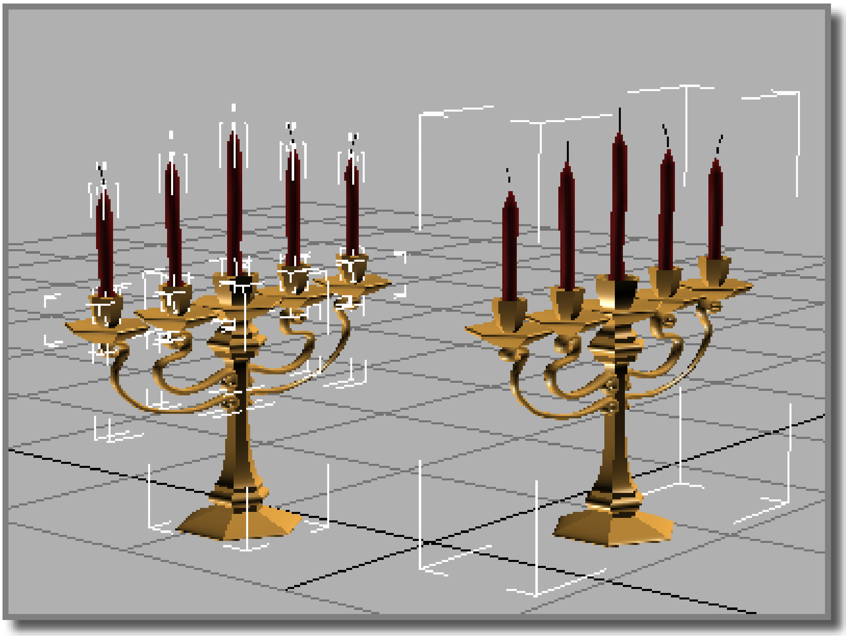
You use groups and assemblies in 3ds Max to combine arbitrary sets of scene entities into a single, non-hierarchical object that you can then manipulate as one. Grouping works with all objects, while assemblies are best used for light fixtures and characters.

For more information about groups, see Using Groups on page 266 and Group Commands on page 281.

For more information about assemblies, see Using Assemblies on page 269 and Assembly Commands on page 286

For more information about character assemblies, see Character Assembly on page 277 and Character Assembly Commands on page 297

Using Groups



Object on the right is a group and treated as a single entity.

Grouping lets you combine two or more objects into a single grouped object. The grouped object is given a name, and then treated much like any other object.

Group names are similar to object names, except that they're carried by the group object. In lists like the one in the Selection Floater on page 231, group names appear in square brackets. For example: [Group01]. In Scene Explorer on page 7379 and related dialogs, the square brackets enclose the group object icon instead.

The commands to manage groups are on the default Group menu on page 7477.

General Features of Groups

Once you group objects, you can treat them as a single object in your scene. You can click any object in the group to select the group object.

When you create a group, all of its member objects are rigidly linked to an invisible dummy object. The group object uses the pivot point and the local transform coordinate system of this dummy object.

Groups can be nested. That is, groups can contain other groups, up to any level.

Transforming and Modifying a Group

You can transform and modify a group as if it were a single object, and you can animate the transforms and the modifiers.

When you apply a modifier to the group, this applies an instance of the modifier to each object in the group. A grouped object retains its modifier instance, even if you later remove it from the group.

When you apply a transform to the group, on the other hand, this applies only to the group as a whole. More precisely, 3ds Max applies transforms to the dummy object that represents the group.

You can transform and animate individual objects within a group independently from the group itself. However, when you transform the group itself, the transform affects all grouped objects equally. The group transform is uniformly added to objects that have independent motions. An analogy is a cage of birds, each flying around on its own, while the cage itself is being moved. In the case of groups, the "cage" (the dummy object) expands to surround all objects in the group, wherever the objects' independent transforms take them.

Accessing Objects in a Group

You can open and close groups to access the individual objects contained in them without dissolving the group. These commands maintain the integrity of the group.

- Open on page 283: Temporarily opens the group so that you can access its member objects. While a group is open, you can treat the objects (or nested groups) as individuals. You can transform them, apply modifiers, and access their modifier stacks.

- Close on page 283: Restores the group when you're finished working with the individual objects.

Dissolving Groups

You can permanently dissolve groups by either ungrouping or exploding them. Both commands dissolve groups, but to different levels.

- Ungroup on page 284: Goes one level deep in the group hierarchy. It separates the current group into its component objects (or groups), and deletes the group dummy object.
- Explode on page 285: Similar to Ungroup, but dissolves all nested groups as well, leaving independent objects.

When you Ungroup or Explode a group, the objects within the group lose all group transforms not on the current frame. However, objects retain any individual animation.

To transform or modify the objects within a group, you must first remove them from the group, either temporarily or permanently. The Open command lets you do this.

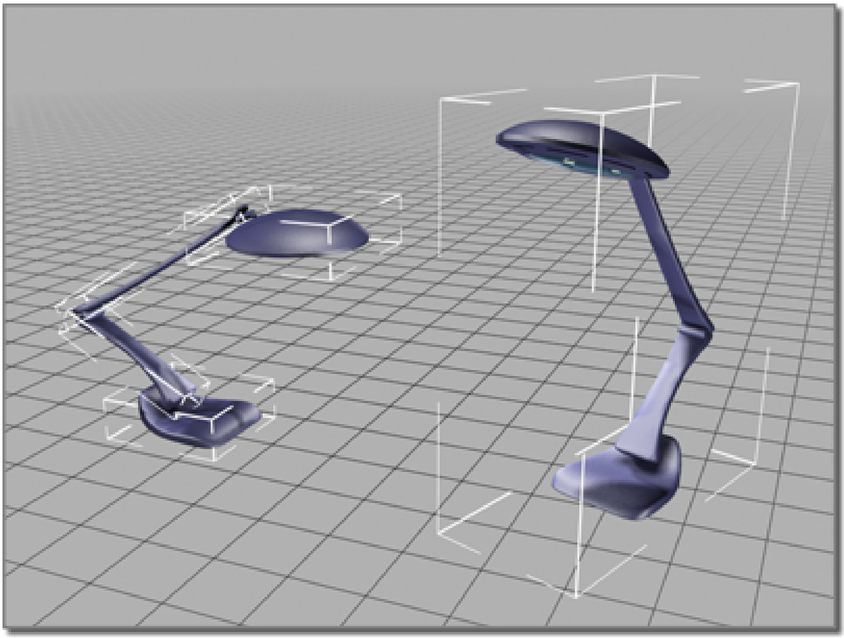
Comparing Groups with Other Selection Methods

Compared to the other methods you can use to combine objects in 3ds Max, grouping is more permanent than selection sets, but less permanent than attaching objects.

- Selection sets on page 199: Form a temporary collection of objects to which you apply the current commands. As soon as you select another object, the selection set is gone.
- Named selection sets on page 206: Let you reselect the same pattern of objects, but the positional relationship between those objects (their transforms) might be different each time you recall the named set.
- Grouped objects: Maintain their positional relationships unless you open the group and rearrange them. A group also keeps its identity as an individual object.
  Each object in a group retains its modifier stack, including its base parameters. At any time, you can open the group to edit an object, and then close the group to restore the group identity.

- Attached objects (see Editable Mesh (Object) on page 2083): Attached objects form a single object. The modifier stacks of the original objects are lost, including their base parameters. You can regain the form of the original objects by detaching them, but they become plain meshes.
- Assemblies on page 269 are useful for creating combinations of geometry and light objects that act as lighting fixtures.

Using Assemblies

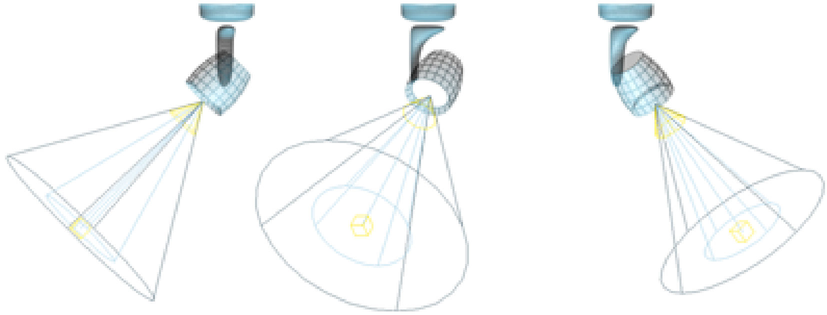


Object on the right is an assembly and is treated as a single entity.

Assemblies are useful for creating combinations of geometry and light objects that act as lighting fixtures; you use them to represent the housing of a lamp and its light source or sources. You can use assemblies to represent lighting fixtures such as simple desk lamps, lighting strips, track systems, wall sconces with fluorescent or incandescent lights, chandelier systems, line voltage cable systems, and so on.

When you create light assemblies, first you create your objects and build a hierarchy, then set joint parameters and assign inverse kinematics (IK) on page 3374. As a final step, you assemble the object hierarchy. The lights you use in the assembly have light-multiplier and filter color controls. You wire on page 3322 the Dimmer and Filter Color parameters of the Luminaire helper object to the parameters of the light sources that are members of the light assembly.

NOTE In order to wire the Luminaire controls to the light parameters, you must first open on page 291 the assembly; then, after wiring, you close on page 292 it.



You can use IK to point a luminaire's beam by simply moving the light's target object.

Assemblies and Groups

Assembly functionality is a superset of grouping on page 266. Like grouping, creating an assembly lets you combine two or more objects and treat them as a single object. The assembled object is given a name, and then treated much like any other object.

The main difference with assemblies is that, when you assemble on page 287 the member objects, you specify a head object on page 294: a Luminaire helper object on page 295. The head object acts as a front end for the assembly, and its parameters appear in the Modify panel when the assembly is selected. You can use these parameters to control the light sources in the assembly via parameter wiring on page 3322. You can create other types of head objects with MAXScript; for further information, open the *MAXScript Reference*, available from the Help menu, and look in Creating MAXScript Tools > Scripted Plug-ins > Scripted Helper Plug-ins.

Assembly names are similar to object names, except that they're carried by the assembly. In lists like the one in the Selection Floater on page 231, assembly names appear in square brackets; for example: [Assembly01]. In Scene Explorer on page 7379 and related dialogs, the square brackets enclose the assembly object icon instead.

TIP After you've created one fixture and assembled the parts, use instancing on page 8014 to copy on page 1023 the fixture, and then distribute them in your scene. That way, if you change the attributes for a light source in an assembly, the change will be reflected in all the instanced light sources. For example, in the early design stages, you might use shadow maps, but later you might want to switch to advanced ray-trace shadows for greater accuracy in rendering. Using instancing makes it easier to change such settings globally.

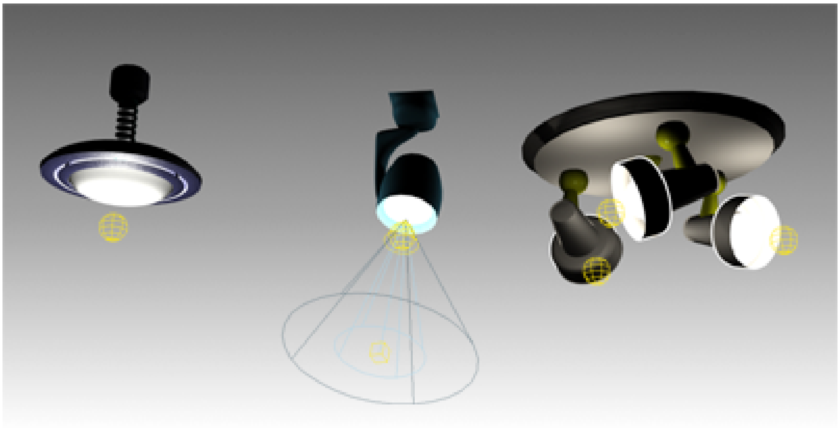
General Features of Assemblies

Once you assemble objects, you can treat them as a single object in your scene. You can click any object in the assembly to select the entire assembly.

When you create an assembly, all of its member objects are rigidly linked to an invisible Luminaire helper object. The assembly uses the pivot point and the local transform coordinate system of this helper object.

You can nest assemblies. That is, assemblies can contain other assemblies (or groups), up to any level.

The head object parameters appear in the Modify panel when the assembly is selected. You can use the 3ds Max Wire Parameters on page 3322 functionality to connect these parameters to those of light objects in the assembly. For a step-by-step procedure, see To wire a head object to a light source on page ?.



Luminaire types

Left: Fixed

Middle: Orientable

Right: Multiple lights

Transforming and Modifying an Assembly

You can transform or modify an assembly as if it were a single object, and you can animate the transforms.

Unlike a group, when you apply a modifier to the assembly, only the luminaire receives the modifier. Thus, deforming modifiers such as Bend don't have any effect on assemblies.

When you apply a transform to the assembly, it applies to the assembly as a whole. More precisely, 3ds Max applies transforms to the dummy object that represents the assembly. To modify member objects, you must first open the assembly, select the objects, and then apply modifiers. Such modifiers do not appear in the modifier stack when the assembly is closed.

You can transform and animate individual objects within an assembly independently from the assembly itself. However, when you transform the assembly itself, the transform affects all assembled objects equally. The assembly transform is uniformly added to objects that have independent motions. An analogy is a cage of birds, each flying around on its own, while the cage itself is being moved. In the case of assemblies, the "cage" (the dummy object) expands to surround all objects in the assembly, wherever the objects' independent transforms take them.

Accessing Objects in an Assembly

You can *open* and *close* assemblies to access the individual objects contained in them without dissolving the assembly. These commands maintain the integrity of the assembly.

- Open on page 291: Temporarily opens the assembly so that you can access its member objects. While an assembly is open, you can treat the objects (or nested assemblies/groups) as individuals. You can transform them, apply modifiers, and access their modifier stacks.
- Close on page 292: Restores the assembly when you're finished working with the individual objects.

Using Make Unique with Assemblies

When you clone assemblies using instancing, and then make the clones unique, it's important to consider how this affects parameter wiring. Consider the following typical usage case:

1. Drag an assembly, such as a light fixture, into the scene.
2. Clone the assembly several times using the Instance option and position the instances in the scene.
3. To make the scene look more realistic, giving the appearance of randomness to the objects in the scene, make some of the assembly instances unique and adjust their parameters to differ from the rest of the instances.

When you clone-instance an assembly, all objects in the assembly, along with all the parameter wires, are instanced. So if you change a wired luminaire parameter, all instanced assemblies are affected.

When the modifier stack displays an assembly head that is an instance or reference, the Make Unique on page 7654 button is active. By clicking it, the assembly head object is made unique with respect to its instances and all the assembly members are also made unique.

The parameter wiring between the unique assembly head and its members is de-coupled from the other instances of the assembly. Changing the parameters of the unique assembly head object affects only the parameters of its own members, not the members of the other instances of the assembly.

When multiple assembly instances are selected, the Make Unique command works the same as when multiple instances of an object are selected. You're asked whether you want to make the selected assemblies unique one with respect to each other.

- If you answer Yes, 3ds Max makes the assemblies unique one with respect to another and parameter wires are reconnected inside each unique assembly. That is, the parameters of each unique assembly head drives only the parameters of its own members, not that of the members in any other assembly instances.
- If you answer No, then the selected assemblies are made unique only with respect to the other assembly instances. The parameters of unique assembly heads drive only the parameters of their members, not the members of the other assembly instances.

NOTE If you chose to instance the controllers when you instanced the assembly, the Modify panel > Make Unique command does not make the controllers unique. You can make them unique by doing the following: Open Track View, select the Transform track for object whose controller you want to make unique, and click the Make Unique button in the Track View toolbar.

Dissolving Assemblies

You can permanently dissolve assemblies by either *disassembling* or *exploding* them. Both commands dissolve assemblies, but to different levels.

- Disassemble on page 293: Goes one level deep in the assembly hierarchy. It separates the current assembly into its component objects (or assemblies/groups), and deletes the assembly head object.
- Explode on page 293: Similar to Disassemble, but dissolves all nested assemblies and groups as well, leaving independent objects.

When you disassemble or explode an assembly, any transform animation applied to the assembly is lost, and objects remain as they were in the frame at which the dissolution is performed. However, objects retain any individual animation.

To transform or modify the objects within an assembly, you must first remove them from the assembly, either temporarily or permanently. The Open command lets you do this.

Comparing Assemblies with Other Selection Methods

Compared to the other methods you can use to combine objects in 3ds Max, assembling is more permanent than selection sets, but less permanent than attaching objects.

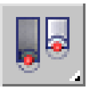
- Selection sets on page 199: Form a temporary collection of objects to which you apply the current commands. As soon as you select another object, the selection set is gone.
- Named selection sets on page 239: Let you reselect the same pattern of objects, but the positional relationship between those objects (their transforms) might be different each time you recall the named set.
- Assembled and grouped on page 266 objects: Maintain their positional relationships unless you open the assembly and rearrange them. An assembly also keeps its identity as an individual object.
  Each object in an assembly retains its modifier stack, including its base parameters. At any time, you can open the assembly to edit an object, and then close the assembly to restore the assembly identity.
- Attached objects (see Editable Mesh (Object) on page 2083): Attached objects form a single object. The modifier stacks of the original objects are lost, including their base parameters. You can regain the form of the original objects by detaching them, but they become plain meshes.

See also:

- Lights on page 4970

Procedures

To insert and place an existing assembly:

1. Turn on AutoGrid on page 2597.
2. Drag the assembly from a Web page (if it's an i-drop object on page 7160) or from your local disk and drop it into your scene, placing it on any existing surface.
3.  On the main toolbar, click Use Pivot Point Center on page 976.
4. Position the assembly as you would any other object to aim it in a specific direction.

5 If necessary, wire on page ? the assembly luminaire to its light source or sources.

6 Select the assembly, and then use the Modify panel settings to adjust the intensity of the light with the Dimmer control.

To create your own luminaire:

1 Create the geometry of the lighting fixture.

2 Create a light source on page 4974 or on the Create panel, click Lights to add a standard or photometric light to the geometry of the lighting fixture you just made.

3 Select all the objects in the assembly, including geometrical objects and lights.

NOTE If using IK, leave the light targets out of the assembly so that you can manipulate them independently.

---

4 Choose Assembly menu > Assemble.

A dialog appears requesting a name for the assembly and that you specify a head object. The only head object type available by default is Luminaire on page 295.

5 Enter a name for the assembly and click OK.

6 Wire on page ? the assembly luminaire to its light source or sources. More information on parameter wiring is available at the link in this step.

If more than one light source is present inside the assembly, create a chain of wired parameters. Then enter the desired relationship expression in the expression text box.

To adjust the pivot location of an assembly:

- When you adjust the pivot point of a closed group or assembly, the pivot point of all group and assembly members are affected, not only the pivot point of the group or assembly head object. Therefore, we recommend that you open the assembly, adjust the pivot of the head object, and then close the assembly.

To use an assembly with radiosity:

- Right-click the Luminaire, choose Properties, and on the Object Properties dialog on page 305 choose the Radiosity tab. You can exclude and control radiosity parameters of the geometry and lights independently.

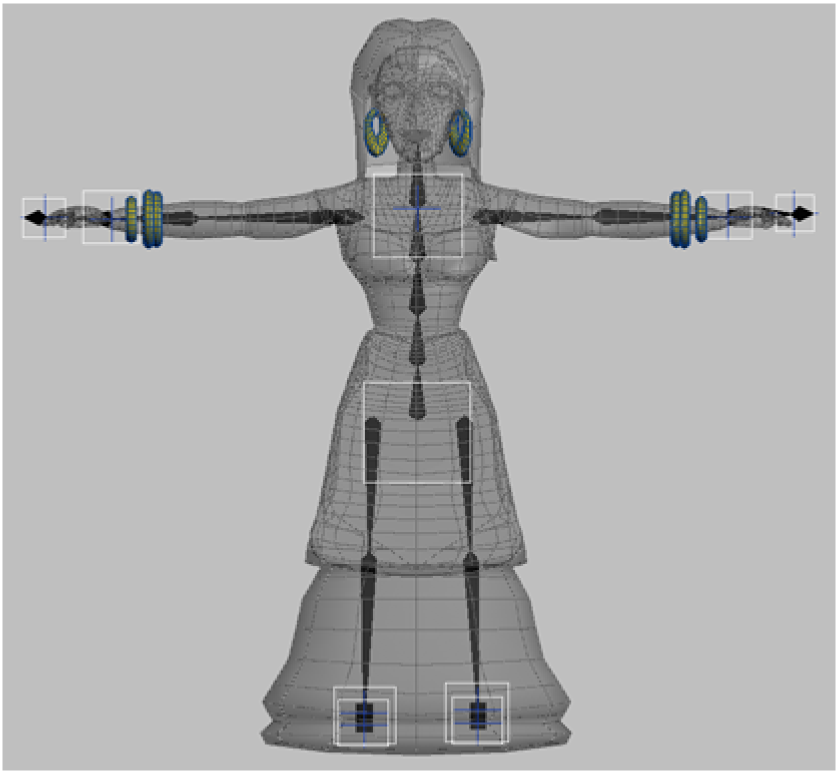
To adjust the properties of an assembly:

1. After wiring the Dimmer and Filter Color parameters, select the Luminaire, and then go to Modify panel to display the luminaire parameters.
2. Adjust the parameters.

   The effect is visible in the viewport.

Character Assembly

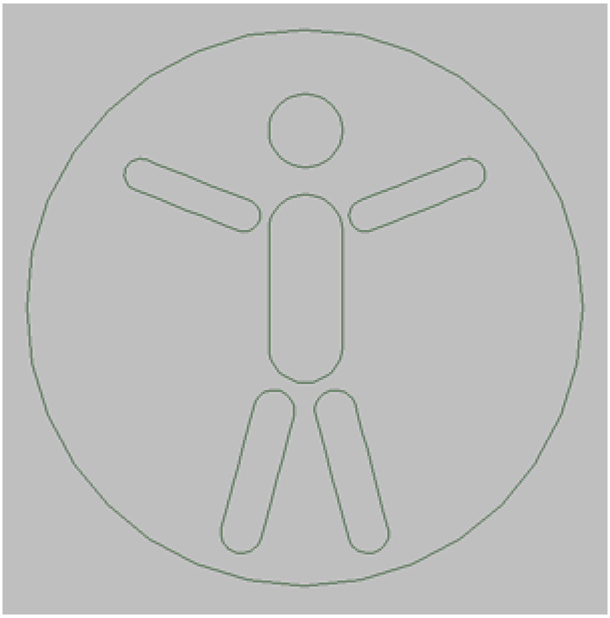
A *character assembly* is a special type of group for objects particular to a character setup: the character mesh, bones, IK chains, helper objects, controllers, and other objects used to animate characters. Once the objects are grouped (assembled), you can perform various functions on the group as a whole, such as saving and loading animation for the entire bone/mesh set.



The objects that make up a typical character assembly

Unlike an ordinary group, there is no need to open a character assembly to work with its individual members.

When a character assembly is created, it is designated by a placeholder object called a *node*, placed near the bottom of the character assembly. Selecting the node gives access to special tools for working with character models and animation.



Character assembly node

A character assembly will not create a character mesh or bone structure for you. The character assembly tool is designed for use on character structures that have already been set up using other tools.

Although the character assembly feature was designed for use with character structures, it will work equally well with any type of hierarchy or related set of objects.

Adding Character Assembly Commands to the UI

By default, the character-assembly commands described here are not part of the 3ds Max user interface. To add them, choose Customize menu > Customize User Interface, click the tab representing the part of the UI to which you'll add the commands (Keyboard, Toolbars, etc.) and then, from the Category drop-down list, choose Characters. Use standard CUI functionality on page 7697 to add the commands.

TIP To add a fully populated Character menu to the menu bar, on the Customize User Interface dialog, scroll the Menus list to Character, and then drag the Character item over to the Main Menu Bar list on the right side of the dialog.

Creating a Character Assembly

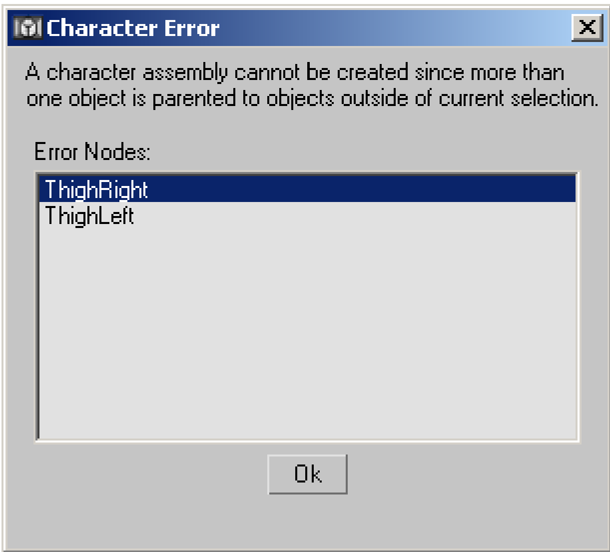
To create a character assembly, first select the objects that will make up the assembly. Next, you need to add any character-assembly commands you wish to use to the user interface; see Adding Character Assembly Commands to the UI on page 279. Last, choose the Create Character command.

All selected objects become members of the assembly, and the character assembly node is created. Other objects can be manually added to the assembly after it has been created.

Once the character assembly has been created, you can work with it in a variety of ways. For information about the Modify panel options available after a character assembly is created, see Create Character on page 298.

Linked Objects in Character Assemblies

Any or all members of the character assembly can be linked to a single object outside the assembly, but no more than one. For example, ThighLeft and ThighRight, which are both part of the assembly, can both be linked to Pelvis, which is not part of the assembly. However, if ThighLeft and ThighRight are each linked to different objects outside the assembly, the creation of the assembly will fail, and will show the following error message:



Parameter Wiring and Animation

If you plan to wire parameters between two objects, create the assembly first and then set up the wiring. Be sure to include both objects in the assembly.

Because wiring should be done after the assembly is created, you will probably find it easiest to create the character assembly before animating the character.

See also:

- Create Character on page 298
- Destroy Character on page 302
- Lock/Unlock Character on page 302
- Save Character on page 303
- Insert Character on page 303
- Skin Pose Commands on page 304
- Merge Animation on page 7063

Group Commands

The commands to manage groups are on the Group menu on page 7477.

Group on page 282

Open Group on page 283

Close Group on page 283

Ungroup on page 284

Attach Group on page 286

Detach Group on page 285

Explode Group on page 285

See also:

- Using Groups on page 266

Group

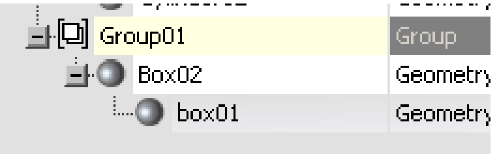
Group menu > Group

The Group command combines a selection set of objects or groups into a single group.

Once you group objects, you can treat them as a single object in your scene. You can click any object in the group to select the group object. You can transform the group as a single object, and you can apply modifiers as if it were a single object.

Groups can contain other groups, up to any level.

Group names are similar to object names, except that they're carried by the group object. In lists like the one in the Selection Floater on page 231, group names appear in square brackets. For example: [Group01]. In Scene Explorer on page 7379 and related dialogs, the square brackets enclose the group object icon instead.



A group in Scene Explorer

If a group is selected, its name will appear in "bolded" text in the Name And Color rollout.

All members of a group inherit the visibility of the parent when a visibility controller is assigned to the parent.

Groups are considered whole objects in the Light Exclude/Include dialog, so you can exclude (or include) all objects in a group by selecting the group in the list. If a group is nested within another group, only the "outer" group is available in the list. To exclude only certain objects in a group, open the group before displaying the Exclude/Include dialog.

Procedures

To define a group:

1 Select two or more objects.

2 Choose Group menu > Group.

A dialog appears requesting a name for the group.

3 Enter a name for the group and click OK.

To define a nested group:

1 Select two or more groups or any combination of groups and objects.

2 Choose Group > Group.
A dialog appears requesting a name for the group.

3 Enter a name for the new group object and click OK.

Open Group

Select one or more groups. > Group menu > Open

The Open command lets you ungroup a group temporarily, and access objects within a group.

You can transform and modify the objects within the group independently from the rest of the group, then restore the original group using the Close command.

Procedures

To open a group:

1 Select one or more groups.

2 Choose Group > Open. A pink bounding box appears, and the objects in the group are now accessible.

To open nested groups:

1 Select the group within the opened group.

2 Choose Group > Open.

Close Group

Select the pink dummy object of an opened group. > Group menu > Close

The Close command regroups an opened group. For nested groups, closing the outermost group object closes all open inner groups.

When you link an object to a closed group, the object becomes a child of the group parent rather than of any member of the group. The entire group flashes to show that you've linked to the group.

Procedures

To close all opened groups nested within a main group:

1. Select the pink bounding box representing the main group.
2. Choose Group > Close.

To close a nested group:

1. Select any object in the nested group or its dummy.
2. Choose Group > Close.

Ungroup

Select one or more groups. > Group menu > Ungroup

Ungroup separates the current group into its component objects or groups.

The Ungroup command ungroups one level, unlike Explode on page 285, which ungroups all levels of nested groups.

When you Ungroup a group, the objects within the group lose all group transforms that were applied on nonzero frames, but they retain any individual animation.

All ungrouped entities remain in the current selection set.

Procedures

To ungroup a group:

1. Select one or more groups.
2. Choose Group > Ungroup.

   All components of the group remain selected, but are no longer part of the group. The group dummy is deleted.

Explode Group

Select one or more groups. > Group menu > Explode

The Explode command ungroups all objects in a group, regardless of the number of nested groups, unlike Ungroup on page 284, which ungroups one level only.

As with the Ungroup command, all exploded entities remain in the current selection set.

WARNING Ungroup and Explode remove all transform animations that have been applied to the group as a whole. As with the Ungroup command, all exploded entities remain in the current selection set.

Procedures

To explode a group:

1. Select one or more groups.
2. Choose Group > Explode.

   All objects in the groups remain selected but no longer belong to groups. All nested groups are exploded. All group dummies in the selection are deleted.

Detach Group

Select a group. > Group menu > Open > Select one or more objects detach. > Group menu > Detach

The Detach command detaches the selected object from its group.

This command becomes active when you open the group by choosing the Open command from the Group menu.

Procedures

To detach an object from a group:

1. Open the group.
2. Choose Group > Detach.

The selected objects are now separate, independent objects, no longer members of the group.

Attach Group

Select one or more objects. > Group menu > Attach

The Attach command makes the selected object part of an existing group.

With an object selected, choose this command, and then click a group in the scene.

Procedures

To attach an object to a group:

1. Select one or more objects to attach.
2. Choose Group > Attach.
3. Click any member of a closed group.

   The selected objects become part of the group, which is now selected.

NOTE To attach an object to an open group, click the pink bounding box.

Assembly Commands

The commands to manage assemblies are available from the Group > Assembly submenu.

Assemble on page 287

Disassemble on page 293

Open Assembly on page 291

Close Assembly on page 292

Attach Assembly on page 294

Detach Assembly on page 294

Explode Assembly on page 293

See also:

- Using Assemblies on page 269

Assemble

Select the objects to assemble. > Group menu > Assembly > Assemble

The Assemble command combines a selection set of objects, assemblies, and/or groups into a single assembly, and adds a Luminaire helper object on page 295 as a head object on page 294.

Once you assemble objects, you can treat them as a single object in your scene. You can click any object in the group to select the entire assembly. You can transform the assembly as a single object, and you can apply modifiers as if it were a single object.

Assemblies can contain other assemblies and/or groups, up to any level.

Assembly names are similar to object names, except that they're carried by the assembly. In lists like the one in the Selection Floater on page 231, assembly names appear in square brackets; for example: [Assembly01]. In Scene Explorer on page 7379 and related dialogs, the square brackets enclose the assembly object icon instead.

Each member of an assembly inherits the visibility of the parent when a visibility controller is assigned to the parent, providing its Object Properties > Rendering Control group > Inherit Visibility check box is turned on, or if its Rendering Control is set to By Layer and Inherit Visibility is turned on for its layer.

Assemblies are considered whole objects in the Light Exclude/Include dialog, so you can exclude (or include) all objects in an assembly by selecting the assembly in the list. If an assembly is nested within another assembly, only the "outer" assembly is available in the list. To exclude only certain objects in an assembly, open the assembly before displaying the Exclude/Include dialog.

See also:

- Using Assemblies on page 269

Procedures

To define an assembly:

1 Select two or more objects.

2 Choose Group menu > Assembly > Assemble.

The Create Assembly dialog appears. It requests you to specify a name for the assembly and a head object on page 294. The default head object type is Luminaire on page 295.

3 Enter a name for the assembly, choose Luminaire on page 295 from the list, and click OK.

The selected objects are assembled. The assembly head object position and orientation is determined as follows:

- If there are multiple immediate children of the assembly head (for example, you're assembling several non-hierarchical objects), the head object is aligned with center of bottom face of the assembly bounding box.
- If there's only one immediate child of the assembly head, the assembly head pivot point is aligned with that object's pivot point. For example, if you're assembling a single hierarchy, the topmost object in the hierarchy would be the single immediate child of the assembly head.

To define a nested assembly:

1 Select two or more assemblies or any combination of assemblies and objects.

2 Choose Group menu > Assembly > Assemble.

The Create Assembly dialog appears. It requests a name for the assembly, and a head object.

3 Enter a name for the new assembly object and click OK.

To wire a head object to a light source:

1 Create a hierarchy of lights and geometrical objects that models a lighting fixture. Sets up all the necessary IK chains and other constraints that make the model behave properly when the user interacts with it (orients, positions, aims, etc.).

IMPORTANT For any photometric lights that you want to control with the head object, be sure to turn on the Multiplier check box on the Intensity/Color/Distribution rollout.

2 Select all objects in the fixture and define them as an assembly.

When the assembly is selected, the luminaire parameters Dimmer and Filter Color appear in the Modify panel.

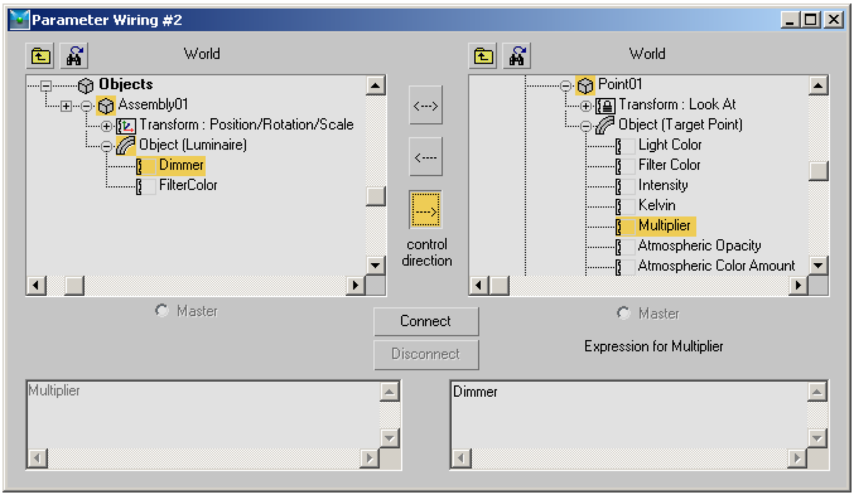
3 From the Animation menu, choose Wire Parameters > Parameter Wire Dialog.

4 The Parameter Wiring dialog appears.

5 On one side of the dialog, find the assembly and expand the branch titled Object (LuminaireHelper). Click the Dimmer item to highlight it.

6 On the other side, find the assembly and expand its hierarchy branch (click the + symbol in the square box). Find and expand the branch for the light source, and then expand its Object branch. Click the Multiplier item to highlight it.

7 Between the two hierarchy lists, click the Control Direction arrow button that points from the selected Dimmer item to the selected Multiplier item.



8 If you're wiring a photometric light, skip this step. If you're wiring a standard light, or any light whose default Multiplier setting is 1.0, do this:

- The Expression box below the selected Multiplier item contains the word "Dimmer." Edit this to read "Dimmer/100". This divides the Dimmer value by 100, giving a 1:1 value ratio between it and the Multiplier setting.

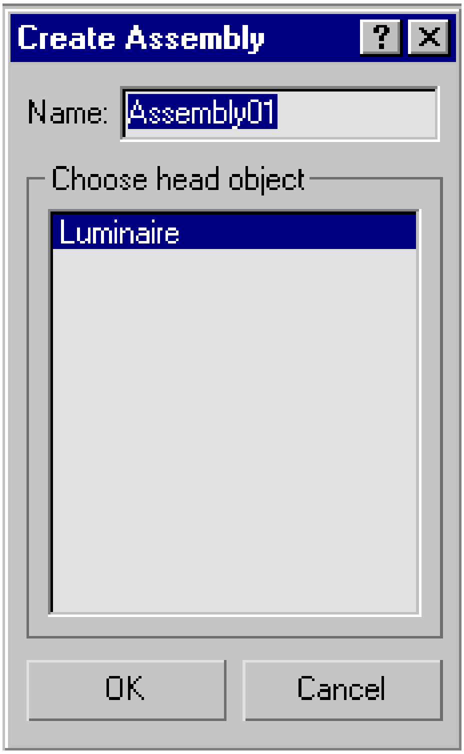
9 Click the Connect button.

Now, when you change the luminaire's Dimmer setting, the light source intensity changes as well.

10 If you like, use the same method to wire the luminaire to any additional light sources in the light fixture.

You can also use this method to wire the luminaire's Filter Color parameter to any light sources' color settings.

Interface



Name Specifies the name of the new assembly. The default name is "Assembly" followed by a two-place number starting with 01 and incremented by one for each new assembly.

Choose Head Object Lets you choose the type of object to serve as the assembly head object.

Open Assembly

Select one or more assemblies. > Group menu > Assembly > Open

The Open command lets you temporarily disassemble an assembly and access its head and member objects individually.

You can transform and modify the head and member objects within the assembly independently from the rest of the assembly, then restore the original assembly using the Close command on page 292.

Procedures

To open nested assemblies:

1. Select an assembly within the opened assembly.
2. Choose Group menu > Assembly > Open.

Close Assembly

Select the luminaire. > Group menu > Assembly > Close

The Close command reassembles an opened assembly. For nested assemblies, closing the outermost assembly object closes all open inner assemblies.

When you link an object to a closed assembly, the object becomes a child of the assembly parent rather than of any member of the assembly. The entire assembly flashes to show that you've linked to the assembly.

Procedures

To close all opened assemblies nested within a main assembly:

1. Select any object in the main assembly or its luminaire head object.

   NOTE If you select an object within an opened inner assembly, using Close will close only that assembly.

   ---

2. Choose Group menu > Assembly > Close.

To close a nested assembly:

1. Select any object in the nested assembly or its luminaire.
2. Choose Group menu > Assembly > Close.

Disassemble

Select one or more assemblies. > Group menu > Assembly > Disassemble

Disassemble separates the current assembly into its component objects or assemblies.

The Disassemble command separates one level, unlike Explode on page 293, which separates all levels of nested assemblies.

When you disassemble an assembly, all components of the assembly remain selected, but are no longer part of the assembly. Any transform animation applied to the assembly is lost, and objects remain as they were in the frame at which the dissolution is performed. However, objects retain any individual animation.

All disassembled entities remain in the current selection set.

NOTE If you have wired the luminaire head to any other parameters, those parameters are still controlled by the wiring setup after disassembly and are not adjustable until you apply a standard controller, such as Bezier Float. Use Track View to do this.

Explode Assembly

Select one or more assemblies. > Group menu > Assembly > Explode

The Explode command separates all objects in an assembly, regardless of the number of nested assemblies and/or groups, unlike Disassemble on page 293, which separates one level only.

When you explode an assembly, all components of the assembly remain selected, but are no longer part of the assembly. Any transform animation applied to the assembly is lost, and objects remain as they were in the frame at which the dissolution is performed. However, objects retain any individual animation.

NOTE If you have wired the luminaire head to any other parameters, those parameters are still controlled by the wiring setup after exploding and are not adjustable until you apply a standard controller, such as Bezier Float. Use Track View to do this.

Detach Assembly

Select an assembly. > Group menu > Assembly > Open > Select one or more objects to detach. > Assembly > Detach

Select one or more objects to detach in an open assembly. > Group menu > Assembly > Detach

The Detach command detaches the selected object from its assembly. If the object is a member of a nested assembly, after you detach it, it is no longer a member of any assembly.

This command becomes active when you open the assembly by choosing Open on page 291 from the Assembly menu.

Attach Assembly

Select one or more objects. > Group menu > Assembly > Attach

The Attach command makes the selected object part of an existing assembly.

With an object selected, choose this command, and then click either a closed assembly in the scene, or the head object of an open assembly.

Procedures

To attach an object to an assembly:

1 Select one or more objects to attach.

2 Choose Group menu > Assembly > Attach.

3 Click any member of an assembly.
   The selected objects become part of the assembly, which is now selected.

Assembly Head Helper Objects

Assembly Head Helper Object

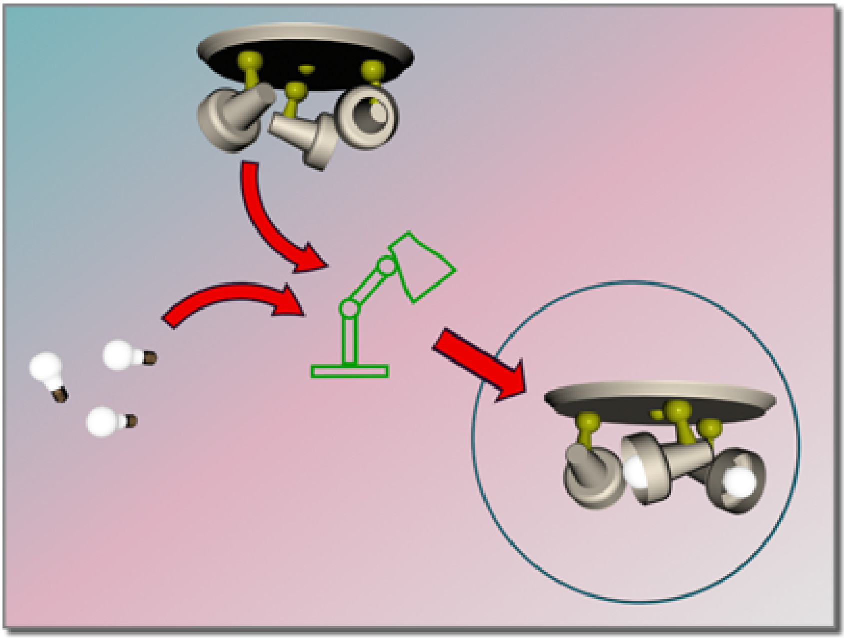
When you create an assembly on page 269 in 3ds Max, the program automatically adds a special type of helper object called a head object, or assembly head. This object serves as the fulcrum of the assembly and also exposes parameters, available in the Modify panel when the assembly is selected, that you can wire to properties of objects inside the assembly. Thus, you can change and animate parameters of assembly member objects without having to open the assembly, as you would with a group.

Luminaire Helper Object

Create panel > Helpers > Assembly Heads > Object Type rollout > Luminaire

The Luminaire helper object serves primarily as a head, or control, object for light fixtures. When you assemble on page 287 a set of objects into a light fixture, you specify that a new luminaire object should be used as the assembly head object. The luminaire's parameters, available from the Modify panel, let you control the light sources in the fixture. See Using Assemblies on page 269 for more information.

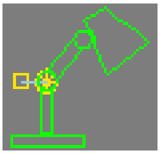
You can also add a Luminaire object separately from the Create panel, but in general it's not necessary.



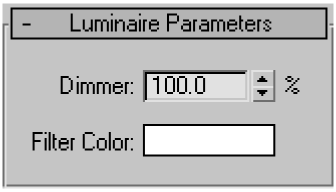
A luminaire object groups and manages the components as a whole.

Interface

When a selected assembly is closed, the Modify panel displays the Luminaire parameters. However, when you open an assembly, 3ds Max shows you the parameters of the whichever object is selected. The Luminaire object provides Dimmer and Filter Color parameters. You wire these to the light objects that are part of the assembly.



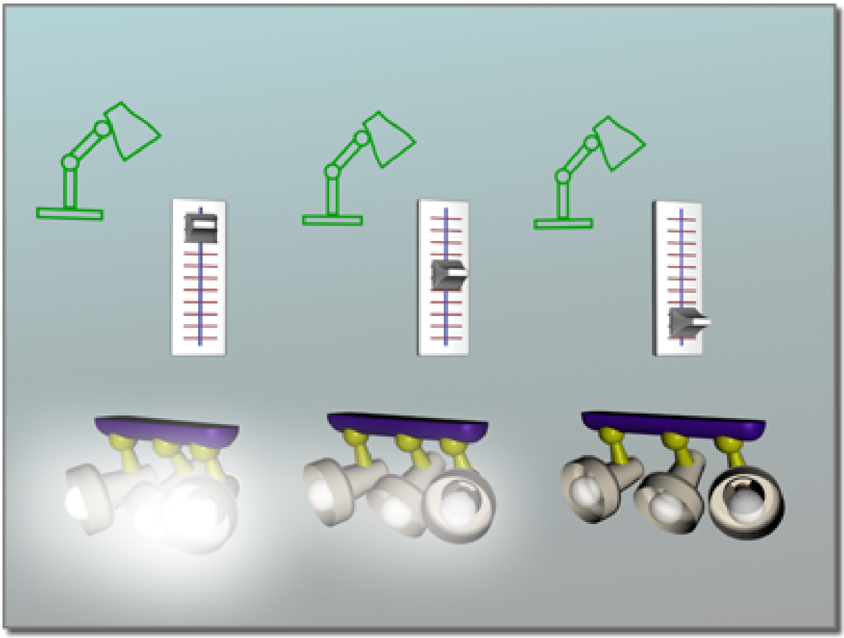
Luminaire icon in the viewport



Luminaire rollout

Dimmer Emulates the dimmer switch of a real-world lighting fixture. The setting determines the percentage of the default light intensity is emitted by the light source of a lighting fixture. You wire this parameter to one or more light sources' Multiplier settings.

Filter Color An RGB color parameter that you link to a light source's color or filter color.



The Dimmer option can control the intensity of all the lights in the luminaire

Character Assembly Commands

By default, the character-assembly commands listed here are not part of the 3ds Max user interface. To add them, choose Customize menu > Customize User Interface, click the tab representing the part of the UI to which you'll add the commands (Keyboard, Toolbars, etc.) and then, from the Category drop-down list, choose Characters. Use standard CUI functionality on page 7697 to add the commands.

TIP To add a fully populated Character menu to the menu bar, on the Customize User Interface dialog, scroll the Menus list to Character, and then drag the Character item over to the Main Menu Bar list on the right side of the dialog.

Create Character on page 298

Destroy Character on page 302

Lock/Unlock Character on page 302

Save Character on page 303

Insert Character on page 303

Skin Pose Commands on page 304

Merge Animation on page 7063

Create Character

See Adding Character Assembly Commands to the UI on page 279.

This command creates a character assembly on page 277.

See also:

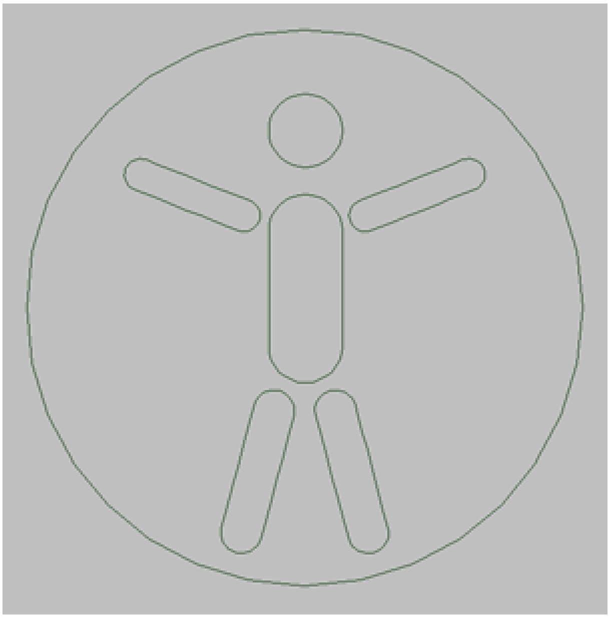
- Destroy Character on page 302
- Lock/Unlock Character on page 302
- Save Character on page 303
- Merge Animation on page 7063

Procedures

To create a character assembly:

1. Create a linked structure of bones or other objects. The structure can have several chains. You can also use the linked structure with the Skin modifier, and/or set up character rigs and controllers as needed.
2. Select all objects that will become members of the assembly.
3. Use this method on page 279 to add the character-assembly commands to the user interface, and then choose the Create Character command.

   The character-assembly node is created at the bottom of the entire selection, as viewed in the Front viewport.



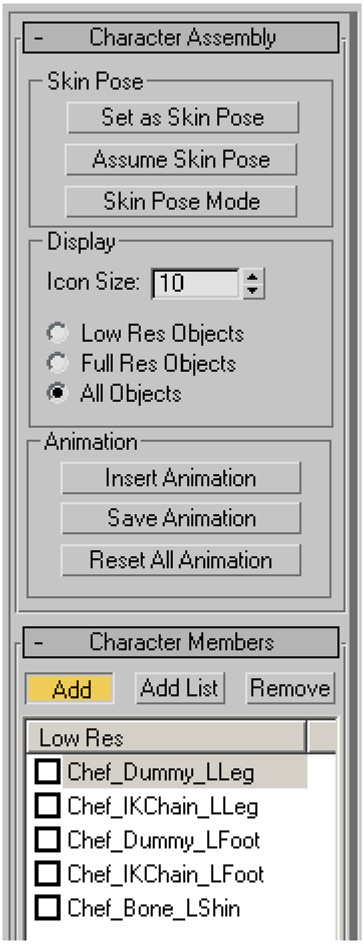
4 On the Modify panel, use the character assembly tools to work with the character structure.

The character assembly is given the default name of Character01, which can be changed. All members of the assembly are listed in the Character Members rollout.

Interface

To work with the character assembly, select the character assembly node and work with the parameters on the Modify panel.

Character Assembly rollout



Skin Pose group

The *Skin pose* is the bone structure pose used by the Skin modifier for associating bones with the mesh. When the Skin modifier is first applied, the current bone structure pose is used as the Skin pose. The Skin pose can sometimes be accidentally altered by animating the bone structure on frame 0. If this occurs, you can use these options to fix the Skin pose.

You can use these options both before and after applying the Skin modifier. You can also invoke these options when any member of the character assembly is selected.

Set as Skin Pose Sets the Skin pose to the current bone structure's pose. The Skin modifier's envelopes and vertex weighting are automatically recalculated to work with the new pose.

Assume Skin Pose Causes the bone structure to take on the Skin pose. This feature can be useful during the animation phase. For example, if you have animated the character on various keyframes and want the character to animate back to its Skin pose at frame 50, you can turn on Auto Key at frame 50 and click Assume Skin Pose.

Skin Pose Mode Poses the character in its Skin pose and allows the Skin pose to be refined. Changes to the bone structure when Skin Pose Mode is on will affect only the Skin pose and not the animation. When Skin Pose Mode is turned off, the bone structure returns to its pose at the current frame.

Display group

When a high resolution character model is animated, redraw time can slow the animation process. To speed up your work, a low resolution version of the model can be used for the animation process, then switched for the full resolution version at render time. Character assembly objects can be designated as Full Res or Low Res on the Character Members rollout.

Low Res Objects Displays only the objects checked in the Low Res display in the Character Members rollout.

Full Res Objects Displays only the objects not checked in the Low Res display in the Character Members rollout.

All Objects Displays all objects in the character assembly.

Animation group

Animation for the character assembly can be saved or reset in this group. Previously saved animation from another character can also be inserted to the current character assembly.

Insert Animation Displays the Merge Animation dialog on page 7063, and prompts for a previously saved animation file.

Save Animation Saves the character assembly animation in an ANM or XML file. Both file types contain the character assembly and its animation. An ANM file is a proprietary format that can be read and saved only by 3ds Max. An

XML file formats the information as XML code, and can be edited with a text editor.

Animation saved as an ANM file loads and saves faster than an XML file. Saving and editing an XML animation file is recommended only for users who are familiar with the XML language, and who have a specific need for editing the file.

Reset All Animation Removes all animation from the character assembly.

Character Members rollout

Add Allows you to select individual objects to add to the character assembly.

Add List Displays the Pick Character Members dialog, where you can select multiple objects from a list and add them to the character assembly.

Remove Removes highlighted object(s) from the assembly. Bones and objects upon which other assembly objects depend, cannot be removed.

Low Res All members of the character assembly are displayed on this list. By default, all members are designated as Full Res objects. To designate a member as Low Res, check the object on the list. The Full Res and Low Res designations are used in conjunction with the Display group selection in the Character Assembly rollout.

Destroy Character

See Adding Character Assembly Commands to the UI on page 279.

Destroying a character deletes the character assembly node.

This command is available only when a character assembly node is selected.

See also:

- Character Assembly on page 277
- Lock/Unlock Character on page 302

Lock/Unlock Character

See Adding Character Assembly Commands to the UI on page 279.

Locking a character assembly prevents the character from being animated. Use these commands when you want to prevent accidental animation of the character, such as when the animation process is complete.

Unlocking a locked character assembly allows you to animate the character.

These commands are available only when a character-assembly node is selected.

See also:

- Character Assembly on page 277

Insert Character

See Adding Character Assembly Commands to the UI on page 279.

Choose this command to insert a previously saved character into the current scene.

You save a character assembly as a CHR file with Save Character on page 303. A CHR file contains the character assembly node, all members of the assembly and any animation on the members.

When a character is inserted into the scene, it is placed at the same world-space location it had when saved.

See also:

- Character Assembly on page 277
- Save Character on page 303

Save Character

See Adding Character Assembly Commands to the UI on page 279.

Saving a character saves a character assembly on page 277 at its current location, including its node, all members, and any animation on its members. Use this command for storing a character assembly to disk prior to inserting it into another scene.

Saving a character with this command saves the assembly as a CHR file. You can then insert the CHR file into a scene with Insert Character on page 303.

This option is available only when a character assembly node is selected.

Skin Pose Commands

Animation menu > Set as Skin Pose

Animation menu > Assume Skin Pose

Animation menu > Skin Pose Mode

When the Skin modifier is first applied to a mesh, the bone structure's current pose is used as the skin pose. Subsequent animation of the bone structure on frame 0 can cause the skin pose to be altered. The skin pose commands allow you to change and set the skin pose either before or after you apply the Skin modifier.

The *skin pose* stores a specific position, rotation and scale for an object. Its intended use is for storing a character assembly's pose for the Skin modifier. However, a skin pose can be used for any object to store its current transforms for later retrieval.

These commands work on any object, regardless of whether the structure is part of a character assembly, or whether the bones have been assigned to a mesh with the Skin modifier.

Set as Skin Pose Stores the selected objects' current position, rotation and scale as the skin pose. If the selected objects are assigned as bones for the Skin modifier, the envelopes and vertex weighting are automatically recalculated to work with the new pose.

Assume Skin Pose Causes the selected objects to take on the stored skin pose. This feature can be useful during the animation phase. For example, if you have animated the character on various keyframes and want the character to animate back to its skin pose at frame 50, you can turn on Auto Key at frame 50 and click Assume Skin Pose.

Skin Pose Mode Poses the character in its skin pose and allows the skin pose to be refined. Changes to the objects when Skin Pose Mode is on will affect only the skin pose and not the animation. When Skin Pose Mode is turned off, the structure returns to its pose at the current frame.

See also:

- Character Assembly on page 277

Object Properties 7

The Object Properties dialog, available from the Edit and right-click menus, lets you view and edit parameters for how selected objects behave in viewports and renderings. Note that you cannot necessarily edit all properties; parameters that apply to renderable geometry are unavailable for non-renderable objects. However, parameters that apply to any object, such as Hide/Unhide, Freeze/Unfreeze, Trajectory, and so on, remain available for these non-renderable objects.

With the Object Properties dialog you can specify settings per object or by layer on page 7933. Object settings affect only the object or objects selected. When an object is set to By Layer, it inherits its properties from the layer settings, which you set with the Layer Properties dialog on page 7451.

The Object Properties dialog panels are:

Object Properties Dialog Panels

General Panel (Object Properties Dialog)

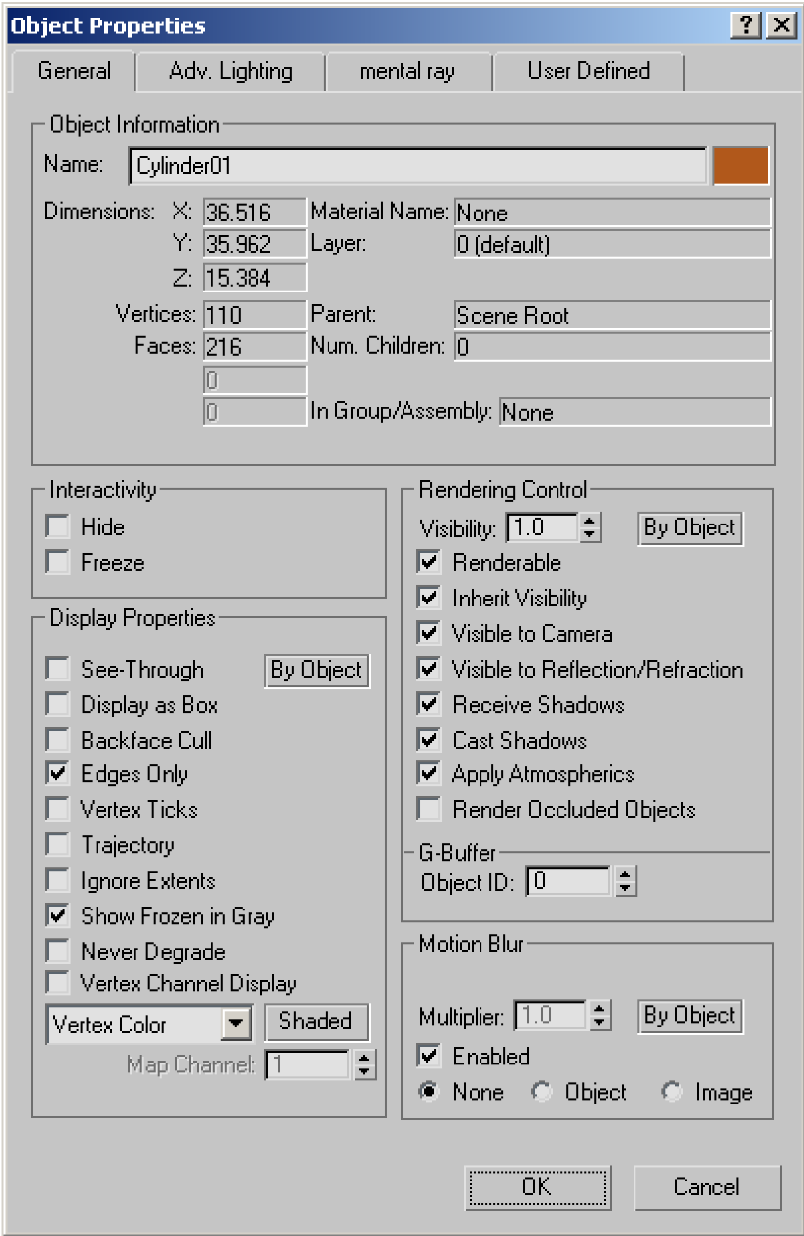
Edit menu > Object Properties > Object Properties dialog > General panel

Select object or objects. > Right-click. > Transform (lower-right) quadrant of the quad menu > Properties > Object Properties dialog > General panel

Layer manager > Click the icon next to an object's name. > Object Properties dialog > General panel

This panel of the Object Properties dialog displays general object information, as well as controls for rendering the object and displaying it in viewports.

Interface


Object Properties
General
Adv. Lighting
mental ray
User Defined
Object Information
Name:
Cylinder01
Dimensions:
X: 36.516
Y: 35.962
Z: 15.384
Material Name: None
Layer: 0 (default)
Vertices: 110
Faces: 216
Parent: Scene Root
Num. Children: 0
0
0
In Group/Assembly: None
Interactivity
Hide
Freeze
Display Properties
See-Through
By Object
Display as Box
Backface Cull
Edges Only
Vertex Ticks
Trajectory
Ignore Extents
Show Frozen in Gray
Never Degrade
Vertex Channel Display
Vertex Color
Shaded
Map Channel: 1
Rendering Control
Visibility: 1.0
By Object
Renderable
Inherit Visibility
Visible to Camera
Visible to Reflection/Refraction
Receive Shadows
Cast Shadows
Apply Atmospherics
Render Occluded Objects
G-Buffer
Object ID: 0
Motion Blur
Multiplier: 1.0
By Object
Enabled
None
Object
Image
OK
Cancel

Object Information group

This group displays information about the selected object, including the following:

Name Shows the name of the object. When a single object is selected, you can edit this field to give the object a new name. When multiple objects are selected, this field shows "Multiple Selected," and cannot be edited.

Color The color swatch shows the object's color. You can click it to display the Object Color dialog on page 387 and select a different color.

Dimensions Displays the X, Y, and Z dimensions of the object's extents on page 7969.

Vertices and Faces Display the number of vertices and faces in the object. For shapes on page 606, these values are the values used if you have made the shape renderable. Faces for renderable shapes are generated only at rendering time.

Shape Vertices and Shape Curves Appear only for shape objects. Shape Vertices is the number of vertices in the shape, and Shape Curves is the number of polygons. (Shape Curves is the value that appeared as "Polygons" in previous releases.)

These values can change over time: they are valid only for the current frame and the current view.

Parent Displays the name of the object's parent in a hierarchy. Shows "Scene Root" if the object has no hierarchical parent.

Material Name Displays the name of the material assigned to the object. Displays "None" if no material is assigned.

Num. Children Displays the number of children hierarchically linked to the object.

In Group/Assembly Displays the name of the group or assembly to which the object belongs. Displays "None" if the object is not part of a group.

Layer Displays the name of the layer which the object is assigned to.

Interactivity Group

Hide Hides the selected object or objects.

Hidden objects exist in the scene, but do not appear in the viewports or rendered images. To unhide hidden objects, use the Display panel on page 7665 or choose Tools > Display Floater on page 7666.

NOTE Objects residing on a hidden layer are automatically hidden, regardless of this setting.

TIP The Layer Manager on page 7441 is the easiest way to hide groups of objects or layers.

Freeze Freezes the selected object or objects.

Frozen objects appear in the viewports, but cannot be manipulated. To unfreeze frozen objects, use the Display panel on page 7665 or choose Tools > Display Floater on page 7666.

NOTE Objects residing on a frozen layer are automatically frozen, regardless of this setting.

TIP The Layer Manager on page 7441 is the easiest way to freeze groups of objects or layers.

Display Properties group

NOTE Most of these options are also available on the Display panel on page 7665 and by choosing Tools > Display Floater on page 7666.

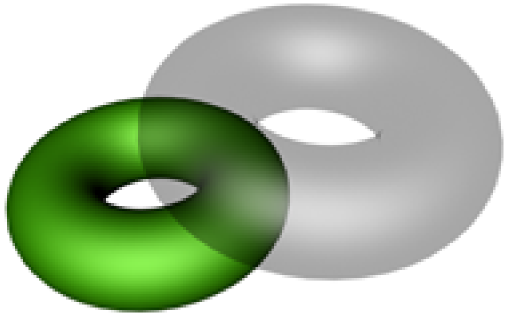
By Object/By Layer Toggles between setting display properties on a per-object basis and at the layer level. When set to By Layer, the object display properties are unavailable; set the layer properties on the Layer Properties dialog on page 7451.

NOTE If multiple objects are selected and have different values for this setting, this button reads "Mixed."

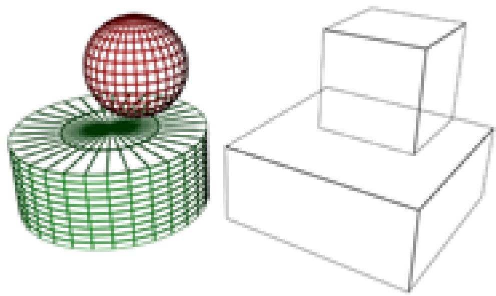
See-Through Makes the object or selection translucent in viewports. This setting has no effect on rendering: it simply lets you see what is behind an object in a crowded scene, and especially to adjust the position of objects behind the see-through object. Default=off.

You can customize the color of see-through objects by using the Colors panel on page 7712 of the Customize > Customize User Interface dialog on page 7697.

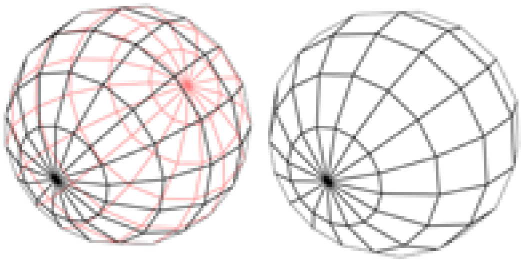
Keyboard shortcut (default): Alt+X



Display as Box Toggles the display of selected objects, both 3D objects and 2D shapes, as bounding boxes on page 7932. Produces minimum geometric complexity for rapid display in viewports. Default=off.

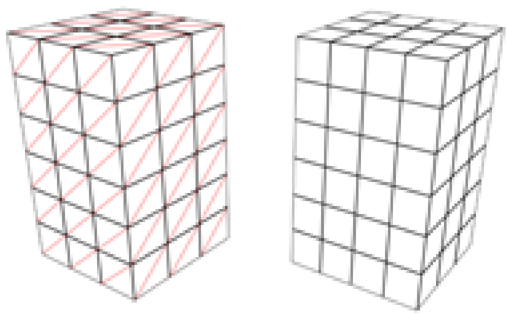


Backface Cull Toggles the display of faces with normals on page 8059 that point away from the view. When on, you see through the wireframe to the backfaces. Applies only to wireframe viewports. Default=on.

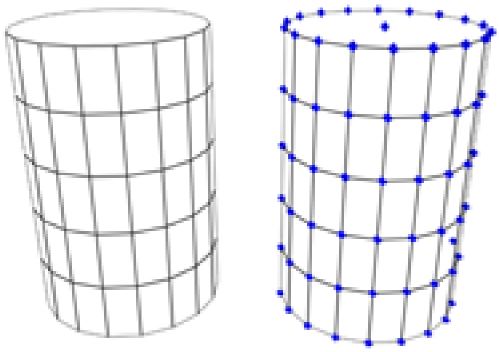


Edges Only Toggles the display of hidden edges and polygon diagonals on page 7953. When on, only outside edges appear. When off, all mesh geometry appears. Applies to Wireframe viewport display mode, as well as other modes with Edged Faces turned on.

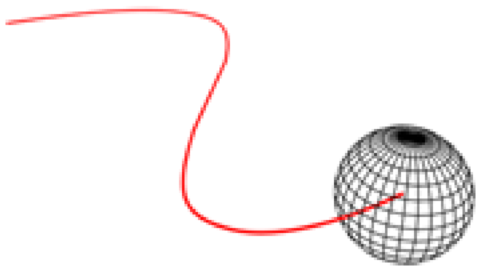
NOTE This option is also available in the Display panel on page 7665 and by choosing Tools > Display Floater on page 7666.



Vertex Ticks Displays the object's vertices as tick marks. Default=off.



Trajectory Displays the object's trajectory on page 8154. Default=off.



Ignore Extents When on, this object is ignored when you use the display controls Zoom Extents on page 7594 and Zoom Extents All on page 7588.

Keyboard shortcut: No default, but you can customize it using the Keyboard panel on page 7698 of the Customize > Customize User Interface dialog on page 7697.

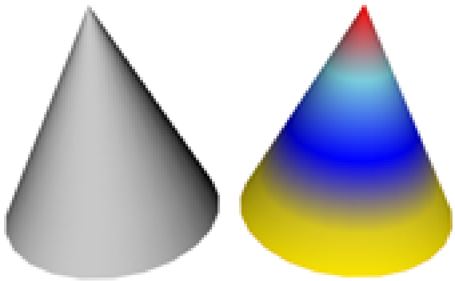
Show Frozen in Gray When on, the object turns gray in viewports when you freeze it. When off, viewports display the object with its usual color or texture even when it is frozen. Default=on.

Never Degrade When on, the object is not subject to adaptive degradation on page 7900.

Vertex Channel Display For editable mesh on page 2075, editable poly on page 2123, and editable patch on page 2019 objects, displays the assigned vertex colors in viewports. The drop-down list lets you choose to display Vertex Color, Vertex Illumination, Vertex Alpha, Map Channel Color, or Soft Selection Color. Default=off.

You can assign vertex colors at all sub-object levels except Edge.

Map Channel Sets the map channel for the vertex color. Available only when the Map Channel Color option is active.



Shaded When on, shaded viewports add shading to the vertex coloring. When off, colors are unshaded. Default=off.

Rendering Control group

By Object/By Layer Toggles between setting rendering controls on a per-object basis and at the layer level. When set to By Layer, the object rendering control settings are unavailable; set the layer properties on the Layer Properties dialog on page 7451.

NOTE If multiple objects are selected and have different values for this setting, this button reads "Mixed."

Visibility Controls the rendered visibility of the object. At 1.0, the object is fully visible. At 0.0, the object is completely invisible when rendered. Default=1.0.

You can animate this parameter. Animating Visibility assigns a visibility controller to the object. By default this is a Bezier float controller on page 3138.

Renderable Makes an object or selected objects appear or disappear from rendered output. Nonrenderable objects don't cast shadows or affect the visual component of the rendered scene. Like dummy objects, nonrenderable objects can manipulate other objects in the scene.

For lights, this is the only available control in the Rendering Controls group. Making a light non-renderable effectively turns it off.

Shape on page 606 objects have the Renderable option turned on by default. In addition, each shape has an Enable In Renderer parameter. When both check boxes are on, the shape appears in rendered output. If the Renderable object properties setting is off, the shape is not renderable, regardless of the state of its local Enable In Renderer check box.

If you apply a modifier that converts the shape into a mesh object, such as a Lathe modifier on page 1501 or Extrude modifier on page 1448, the shape automatically becomes renderable regardless of the state of its local Enable In Renderer setting.

Inherit Visibility Causes the object to inherit a percentage of the visibility of its parent (as determined by the parent's Visibility track in Track View). When a group parent is assigned a visibility track, Inherit Visibility is automatically turned on for all children in the group. The children will have the maximum visibility of the parent. Transparent materials and hidden objects have no effect on this function.

Visible to Camera When on, the object is visible to cameras in the scene. When off, cameras do not view this object; however, its shadows and reflections are rendered. Default=on.

Visible to Reflection/Refraction When on, the object has “secondary” visibility: it appears in rendered reflections and refractions. When off, the object does not appear in rendered reflections or refractions. Default=on.

NOTE An object can have Visible To Camera on but Visible To Reflection/Refraction off, in which case the object renders in the scene but does not appear in reflections or refractions.

Receive Shadows When on, the object can receive shadows. Default=on.

Cast Shadows When on, the object can cast shadows. Default=on.

Apply Atmospherics When on, atmospheric effects are applied to the object. When off, atmospheric effects do not change the rendered appearance of this object. Default=on.

Render Occluded Objects Allows special effects to affect objects in the scene that are occluded by this object. The special effects, typically applied by plug-ins on page 8092 such as Glow on page 6599, use G-buffer on page 7991 layers to access occluded objects. Turning on this control makes the object transparent for the purposes of special effects. This makes no difference when you render to most image files. When you render to either the RLA on page 7364 or RPF on page 7366 file format, however, occluded objects appear with the effect applied on their designated G-buffer layer. Default=off.

G-Buffer group

Allows you to tag an object as a target for a render effect on page 6583 based on the G-buffer on page 7991 channel. Assigning the object a nonzero ID creates a G-buffer channel that can be associated with a render effect.

WARNING The mental ray renderer on page 6230 does not recognize Z-depth with G-buffers. G-buffer data is saved on a single layer. Also, the mental ray renderer does not support the following effects:

- Glow lens effect on page 6599 (rendering effect)
- Ring lens effect on page 6606 (rendering effect)
- Lens effects Focus filter on page 6871 (Video Post)

Object Channel Setting this spinner to a nonzero number means that the object will receive the rendering effects associated with that channel in Render Effects and the post-processing effects associated with that channel in Video Post.

To save the channel data with the rendering, render to either the RLA on page 7364 or RPF on page 7366 file format.

Motion Blur group

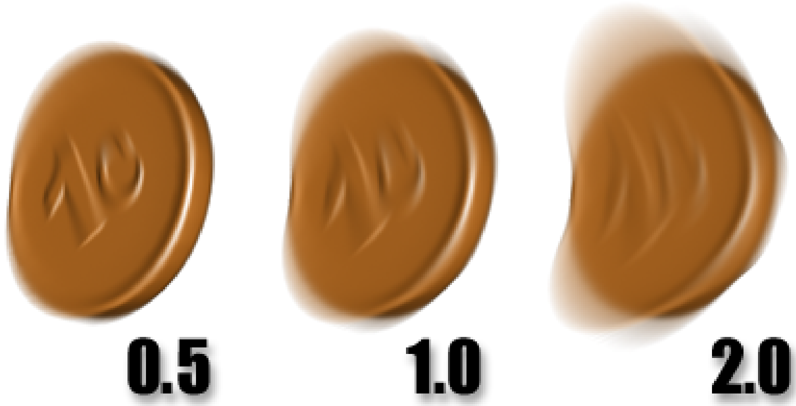
By Object/By Layer Toggles between setting motion blur properties on a per-object basis and at the layer level. When set to By Layer, the motion blur settings are unavailable; set the layer properties on the Layer Properties dialog on page 7451.

NOTE If multiple objects are selected and have different values for this setting, this button reads "Mixed."

Multiplier Affects the length of the motion-blur streak.

If you choose either form of motion blur here in the Object Properties dialog, you must also choose to apply that type of blur in the Render Setup dialog on page 6067.

The rendering speed of object motion blur depends on the complexity of the geometry to which it's assigned. The rendering speed of image motion blur depends on the amount of rendered screen space taken up by the blurring object. In most cases image motion blur renders more quickly. Object motion blur renders more quickly when applied to very simple objects, and image motion blur renders more slowly when the object takes up a lot of screen space, and moves all the way across the screen in a single frame.



Changing the Object Blur Multiplier value.

Enabled When on, enables motion blur for this object. When off, motion blur is disabled regardless of the other blur settings. Default=on.

You can animate the Enabled check box. The main use of animating Enable is to apply motion blur over only a limited range of frames. This can save a tremendous amount of time when you are rendering an animation.

You can enable motion blur for lights and cameras. With the mental ray renderer, moving lights and cameras can generate motion blur. However, they do not generate motion blur with the default scanline renderer.

- **None** Turns off the state of motion blur for the object.
- **Object** Object motion blur on page 8063 provides a time-slice blur effect.
- **Image** Image motion blur on page 8010 blurs the object's image based on the velocity of each pixel.

Advanced Lighting Panel (Object Properties Dialog)

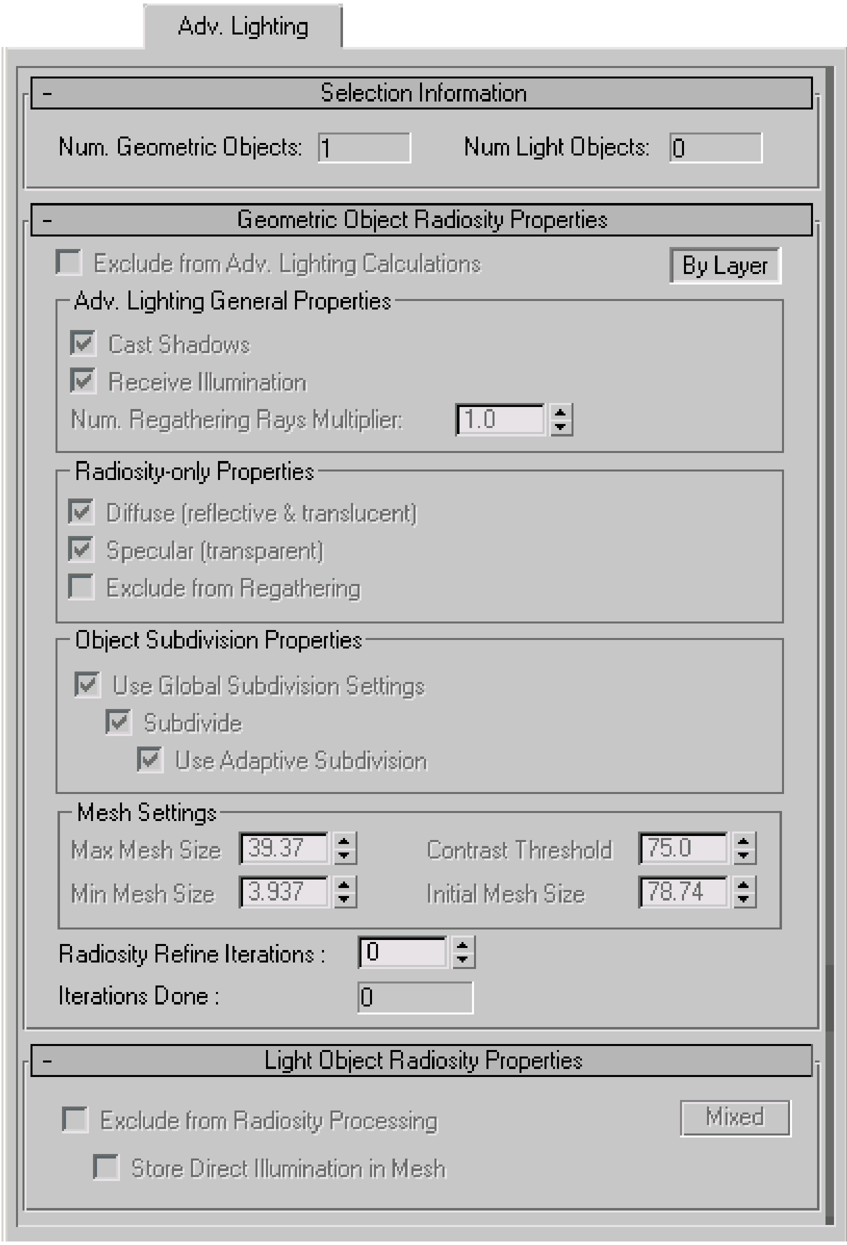
Select object or objects. > Edit menu > Object Properties > Object Properties dialog > Advanced Lighting panel

Select object or objects. > Right-click. > Transform (lower-right) quadrant of the quad menu > Properties > Object Properties dialog > Advanced Lighting panel

Layer manager > Click the icon next to an object's name. > Object Properties dialog > Advanced Lighting panel

This panel of the Object Properties dialog lets you customize how objects behave under advanced lighting (the Light Tracer on page 6154 or radiosity on page 6168).

Interface


Adv. Lighting
Selection Information
Num. Geometric Objects: 1
Num Light Objects: 0
Geometric Object Radiosity Properties
Exclude from Adv. Lighting Calculations
By Layer
Adv. Lighting General Properties
Cast Shadows
Receive Illumination
Num. Regathering Rays Multiplier: 1.0
Radiosity-only Properties
Diffuse (reflective & translucent)
Specular (transparent)
Exclude from Regathering
Object Subdivision Properties
Use Global Subdivision Settings
Subdivide
Use Adaptive Subdivision
Mesh Settings
Max Mesh Size 39.37
Min Mesh Size 3.937
Contrast Threshold 75.0
Initial Mesh Size 78.74
Radiosity Refine Iterations : 0
Iterations Done : 0
Light Object Radiosity Properties
Exclude from Radiosity Processing
Mixed
Store Direct Illumination in Mesh

Selection Information rollout

Num. Geometric Objects The number of geometric objects present in the current selection.

Num. Light Objects The number of lights present in the current selection.

Geometric Object Radiosity Properties rollout

Exclude from Adv. Lighting Calculations When on, the current selection is excluded from advanced lighting (radiosity or light tracing). Objects excluded from advanced lighting will not contribute to indirect illumination.

By Object/By Layer Toggles between object settings and object layer settings. Object settings affect only the object or objects selected. Object layer settings affect all objects on the same layer as the selected object. Most settings on this rollout are available only when this toggle is set to By Object. Default=By Object.

NOTE If multiple objects with different settings are selected, this button will read “Mixed.”

Adv. Lighting General Properties group

Cast Shadows Determines whether objects will cast shadows in the radiosity solution.

NOTE When disabling Cast Shadows, you should also turn off *Diffuse (reflective & translucent)* and *Specular (transparent)* in the Radiosity-only Properties group. If these switches are left turned on, objects will still generate light that can produce artifacts in the solution.

Receive Illumination Determines whether objects will receive indirect illumination.

Num. Regathering Rays Multiplier Lets you adjust the number of rays cast by this object, per pixel. If an object looks “blotchy” after rendering, Increasing this value can improve its appearance. Default=1.0.

TIP Increasing this setting is most useful for objects with large, smooth surfaces. More complex geometry tends not to show advanced lighting artifacts as much as smooth surfaces do.

Radiosity-only Properties group

Diffuse (reflective & translucent) When on, the radiosity solution will process diffuse reflection and translucency on page 8158 of the selected objects.

Specular (transparent) When on, radiosity will process transparency of the selected objects.

Exclude from Regathering When on, objects are excluded from the regathering process when rendering.

For more information on the Radiosity-only Properties group, see Radiosity Control Panel on page 6188.

Object Subdivision Properties group

Use Global Subdivision Settings When on, the object's meshing settings correspond to the global subdivision settings on the Radiosity Control Panel. When off, you can change the meshing settings for each object. Default=on.

- **Subdivide** When on, a radiosity mesh is created for the objects regardless of the global meshing state. The subdivision that is performed is determined by the Use Adaptive Subdivision switch. When off, the settings in the Mesh Settings group are unavailable. Default=on.

- **Use Adaptive Subdivision** Toggles adaptive subdivision. Default=on.

TIP Adaptive meshing is computed for an object only if *Shoot Direct Lights* is turned on in the Radiosity Meshing Parameters rollout on page 6200.

NOTE The Mesh Settings group parameters Contrast Threshold, Min Mesh Size, and Initial Mesh Size are available only when Use Adaptive Subdivision is turned on.

Mesh Settings group

Max Mesh Size The size of the largest faces after adaptive subdivision. Default=36” for imperial units and 100cm for metric units.

When Use Adaptive Subdivision is off, Max Mesh Size sets the size of the radiosity mesh in world units.

Min Mesh Size Faces are not divided smaller than the minimum mesh size. Default=3 inches for Imperial units and 10cm for metric units.

Contrast Threshold Faces that have vertex illuminations that differ by more than the Contrast Threshold setting are subdivided. Default=75.0.

Initial Mesh Size When improving the face shape, faces that are smaller than the Initial Mesh Size are not subdivided. The threshold for deciding whether a face is poorly shaped also gets larger as the face size is closer to the Initial Mesh Size. Default=12 inches for Imperial units and 30cm for metric units.

Radiosity Refine Iterations The number of refine iterations in the radiosity process for the current selection.

Iterations Done The number of refine iterations performed on the current selection.

Light Object Radiosity Properties rollout

These options are available only for light objects.

Exclude from Radiosity Processing When on, the current selection is excluded from the radiosity solution. When lights are excluded from radiosity, their direct contribution is only used for rendering. This option is available only when By Object is selected.

By Object/By Layer Toggles between object settings or object layer settings. Object settings affect only the object or objects selected. Object layer settings affect all objects on the same layer as the selected object.

NOTE If multiple objects are selected and have different settings, this button reads "Mixed."

Store Direct Illumination in Mesh When on, the light's direct illumination is added to the radiosity mesh, even if the global rendering mode is Render Direct Illumination. This is comparable to the Re-Use Direct Illumination option when rendering radiosity, but only for this particular light.

When off, the light's direct illumination is used only when you render the scene. This is comparable to the Render Direct Illumination option.

for more information about the Re-Use Direct Illumination and Render Direct Illumination options, see Rendering Parameters Rollout (Radiosity) on page 6208. In general, re-using direct illumination stored in the radiosity mesh improves render time, but shadows appear coarse and inaccurate unless the mesh is very fine. Rendering direct illumination and shadows (using the radiosity mesh to provide only indirect light) takes more time but gives you a more finished and accurate image.

mental ray Panel (Object Properties Dialog)

Edit menu > Object Properties > Object Properties dialog > mental ray panel

Select object or objects. > Right-click. > Transform (lower-right) quadrant of the quad menu > Properties > Object Properties dialog > mental ray panel

Layer manager > Click the icon next to an object's name. > Object Properties dialog > mental ray panel

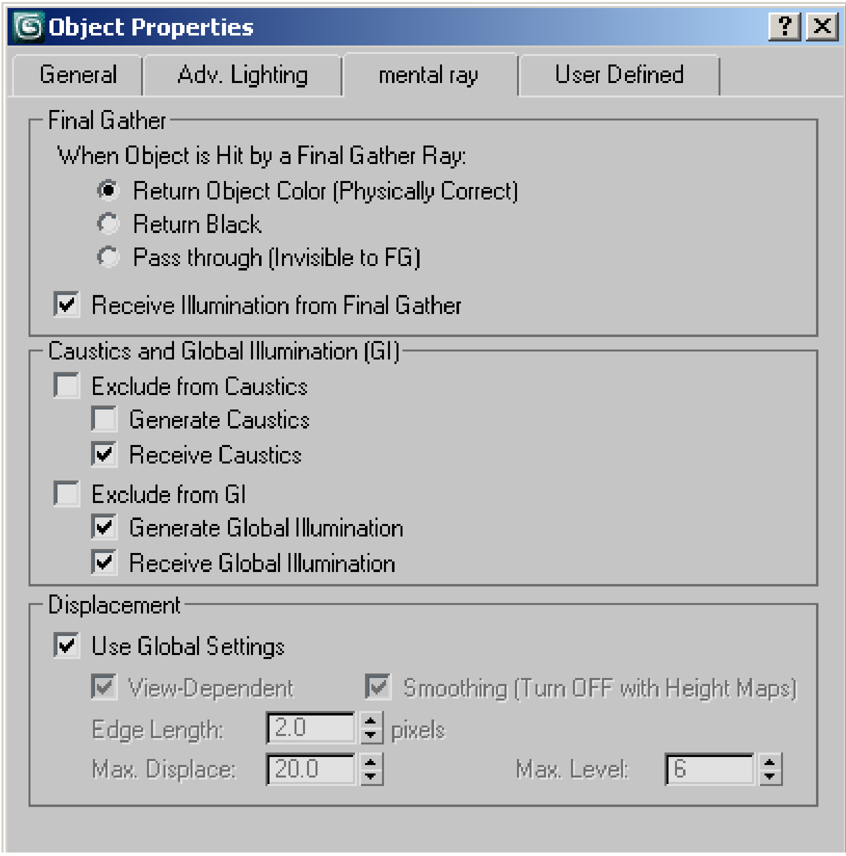
This panel of the Object Properties dialog supports mental ray rendering; specifically, the indirect illumination features caustics on page 6255 and global illumination on page 6261. They control whether objects generate or receive caustics or global illumination.

These settings are ignored where they aren't appropriate. For example, lights can be set to generate caustics, but for a light, the Receive Caustics setting has no effect, as lights aren't renderable. Similarly, these settings have no meaning for cameras.

Also available on this panel are controls for setting displacement parameters on a per-object basis.

TIP Most of these settings, as well as additional rendering-related settings, are also available on the quad menu that opens when you Alt+Ctrl+right-click a selected object.

Interface



The mental ray panel contains object properties for the mental ray renderer.

Final Gather group

These settings let you specify how an object interacts with the final gather process.

When Object is Hit by a Final Gather Ray

- **Return Object Color (Physically Correct)** Returns the object's material color at the point that the final gather ray intersects it, and contributes to final gather illumination. This is the default mode.

- **Return Black** Blocks final gather rays, returning no color, thus not shading the object at all.
- **Pass through (Invisible to FG)** Prevents the final gather process from seeing the object during ray casting. This is useful for ignoring the light contribution from small, complex objects like blades of grass. The final gather rays pass through and strike the underlying object, such as a ground plane that is easier to shade.

Receive Illumination from Final Gather When on, the object is subject to illumination from final gather rays. When off, the object is not illuminated by final gather rays. Default=on.

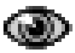
Caustics and Illumination group

These settings determine whether and how the object participates in the simulation of caustics and global illumination (photons). To use these capabilities, the respective check boxes must also be enabled on the Render Setup dialog's Caustics And Global Illumination rollout on page 6306.

 **Exclude from Caustics** When on, the object does not participate in the caustics solution. When off, the additional caustics-related object properties are available. Default=off.

Generate Caustics When on, the object can generate caustics. When off, the object does not generate caustics. Default=off.

Receive Caustics When on, the object can receive caustics. That is, caustic effects are cast onto this object. When off, the object does not receive caustics. Default=on.

 **Exclude from GI** When on, the object does not participate in the global-illumination solution. When off, the additional global-illumination-related object properties are available. Default=off.

Generate Global Illumination When on, the object can generate global illumination. When off, the object does not generate global illumination. Default=off.

Receive Global Illumination When on, the object can receive global illumination. That is, reflected light is cast onto this object. When off, the object does not receive global illumination. Default=on.

Displacement group

These settings let you apply displacement parameters on a per-object basis.

Use Global Settings When on, applies to all objects the Displacement settings on the Render Setup dialog > Renderer panel > Shadows and Displacement rollout on page 6292. Turn off to make settings on a per-object basis. Default=on.

View-Dependent Defines the space for displacement. When View-Dependent is on, the Edge Length setting specifies the length in pixels. When off, Edge Length is specified in world-space units. Default=on.

Smoothing Turn off to have the mental ray renderer correctly render height maps. Height maps can be generated by normal mapping; see Creating and Using Normal Bump Maps on page 6384.

When on, mental ray simply smoothes the geometry using the interpolated normals, making the geometry look better. This result, however, cannot be used for height map displacement because smoothing affects geometry in a way that is incompatible with height mapping.

Edge Length Defines the smallest allowable edge length. The mental ray renderer will stop subdividing an edge once it reaches this size. Default=2.0 pixels.

Max. Displace Controls the maximum offset, in world units, that can be given to a vertex when displacing it. This value can affect the bounding box of an object. Default=20.0.

TIP If displaced geometry appears to be "clipped," try increasing the value of Maximum Displace.

NOTE When using placeholders (see the Translator Options rollout on page 6316), if this value is larger than it needs to be, it can reduce performance. If you experience slow times while displaced objects when Use Placeholder Objects is on, try lowering the Max. Displace value.

Max. Level Controls how many times a triangle can be subdivided. Default=6.

User Defined Panel (Object Properties Dialog)

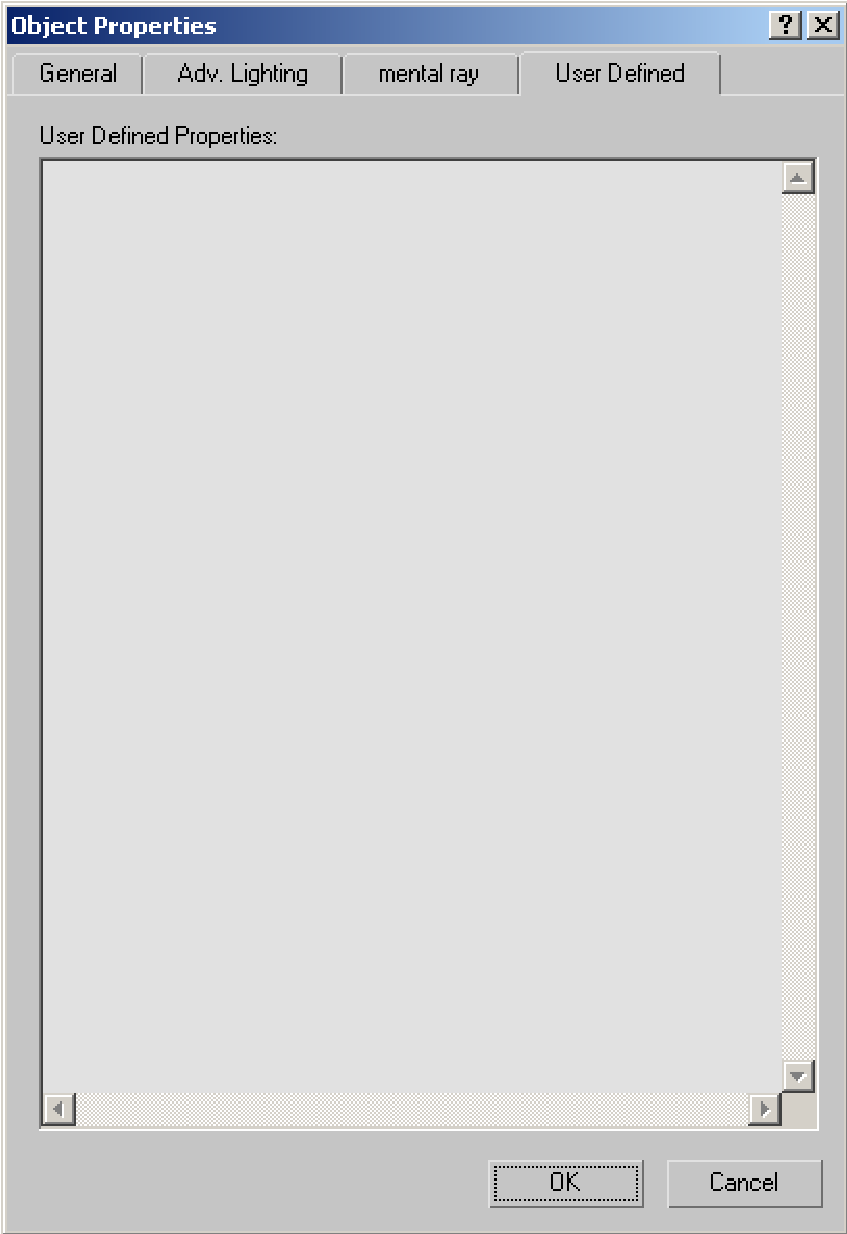
Edit menu > Object Properties > Object Properties dialog > User Defined panel

Select object or objects. > Right-click. > Transform (lower-right) quadrant of the quad menu > Properties > Object Properties dialog > User Defined panel

Layer manager > Click the icon next to an object's name. > Object Properties dialog > User Defined panel

This panel of the Object Properties dialog lets you enter properties or comments that you define yourself.

Interface


Object Properties
General
Adv. Lighting
mental ray
User Defined
User Defined Properties:
OK
Cancel

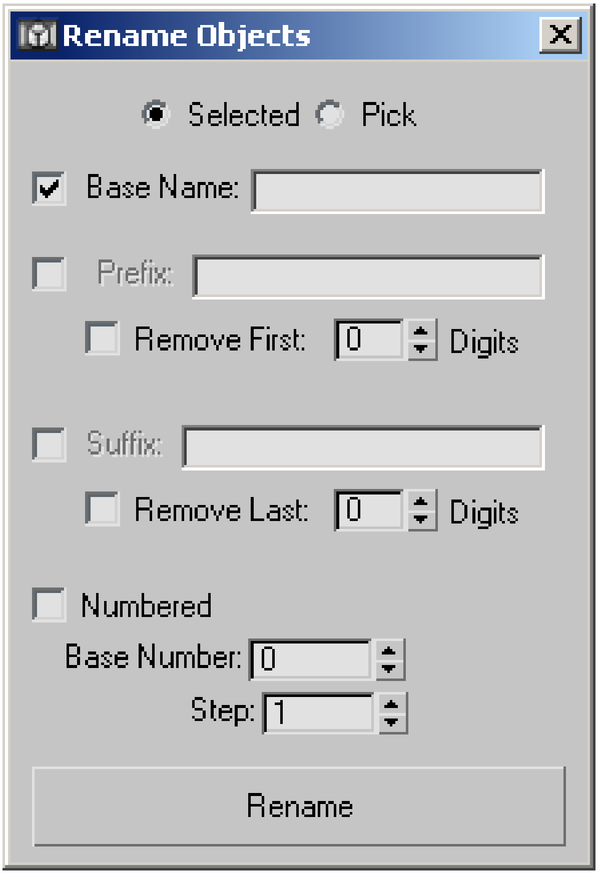
User Defined Properties In this text box, you can enter properties for the object, or comments about it, that you define yourself. 3ds Max doesn't use these properties, but it saves them with the scene, and they reappear whenever you view the Object Properties dialog for the object.

Rename Objects Tool

Tools menu > Rename Objects

The Rename Objects tool helps you rename several objects at once.

Interface



Selected When chosen, renaming affects currently selected objects.

Pick Click to open the Pick Objects To Rename dialog, which lets you choose the objects to rename. This dialog has the same controls as Select From Scene on page 228.

Base Name Enter a base name for all objects. The toggle enables or disables this name.

Prefix When on, lets you enter a string that will be a prefix to the name of all renamed objects.

Remove First N Digits When on, the first N characters in the base name are removed from object names. The spinner sets the value of N.

Suffix When on, lets you enter a string that will be a suffix to the name of all renamed objects.

Remove Last N Digits When on, the last N characters in the base name are removed from object names. The spinner sets the value of N.

Numbered When on, lets you number object names incrementally.

- **Base Number** The base number appended to the name of the first renamed object.
- **Step** The step by which the base number is incremented in succeeding renamed objects.

Rename Click to rename the affected objects and have your changes take effect.

Custom Attributes

Animation menu > Parameter Editor

Keyboard > Alt+1

Use the Parameter Editor to assign custom attributes to objects, modifiers, materials, and animation tracks. A custom attribute is an additional, abstract parameter; abstract in the sense that it does not directly extend the functionality of the object by default. It affects an object only after wire parameters on page 3322, reaction controllers on page 3222, or expression controllers on page 3154 are set up to connect the custom attribute to another parameter in the scene. You can also use custom attributes to store job-specific notes and data.

Custom attributes behave like other object parameters in several ways:

- They are saved and loaded in the scene file along with the object.
- They can be animated and keyframed.
- They are displayed in Track View along with the base parameters.

Each custom attribute parameter can be one of a number of different data types, including integers, floating numbers, Booleans, arrays, nodes, colors, and texture maps. Parameters added to an object or modifier appear on a Custom Attributes rollout on the Modify panel. For each custom attribute parameter you create, you can specify the name, layout, value range, default value, and UI type: spinner or slider for floats and integers, check box for Booleans, etc.

As you customize an attribute, the result is displayed on the Testing Attribute rollout at the bottom of the dialog.

See also:

- Parameter Collector on page 347
- Attribute Holder Modifier on page 1206

Custom Attributes Special Features

The Custom Attributes feature offers an array of workflow-enhancing functionality, including:

- the ability to add custom attributes to specific animation tracks.
- the ability to edit existing custom attributes.
- 13 available data types.
- a variety of available UI options, such as ComboBox and ListBox for the Array data type.
- the ability to position UI elements precisely with X and Y Offset controls.
- the ability to preserve custom attributes on page 7653 when collapsing the stack.
- A special Attribute Holder modifier on page 1206 that lets you collect attributes from different entities and access them in one place on the Modify panel.

Procedures

To add a parameter to an object:

1 Select the object.

2 Choose Animation menu > Parameter Editor.

The Parameter Editor opens.

3 Change settings as desired.

4 Click Add.

The parameter is added to the level specified in the Add To Type list. If an object has no custom attributes, Parameter Editor first adds a Custom Attributes entry to the current Add To Type level, and then adds the parameter to the Custom Attributes entry. If an object has more than one Custom Attributes entry as a result of collapsing its stack, the parameter is added to the first Custom Attributes entry.

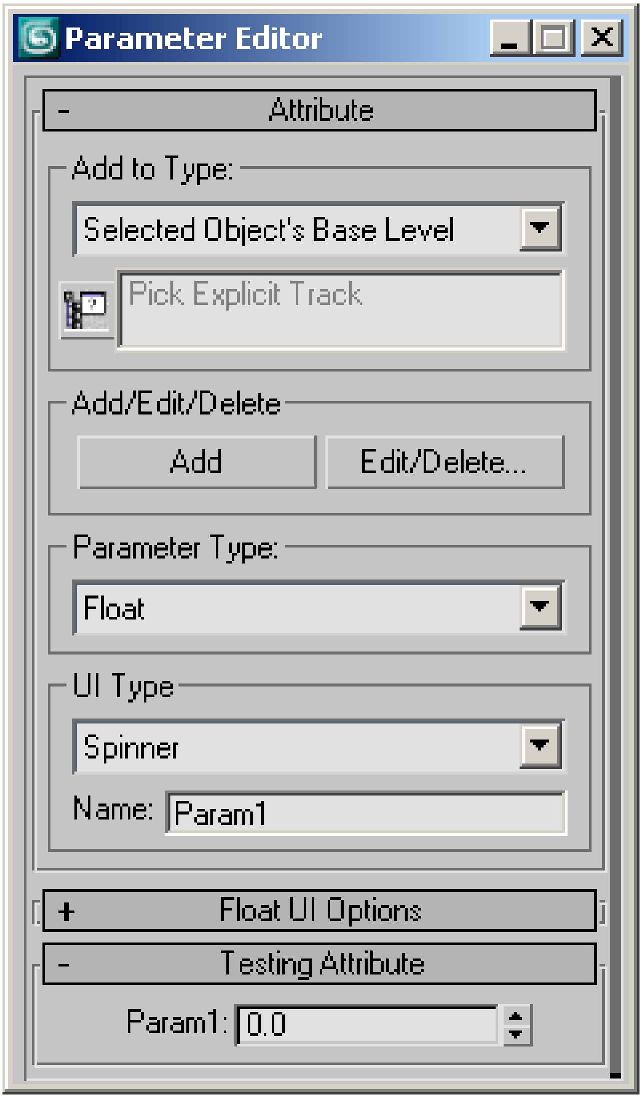
If a custom attribute parameter is assigned to an object or modifier, you can see and edit its value on the Modify panel after adding it by activating the entity to which the attribute is assigned. If the custom attribute is assigned to a material, it's available for that material in the Material Editor, on the Custom Attributes rollout. To access a parameter that's assigned to an animation track, open Track View, highlight the track's Custom Attributes entry, and then right-click and choose View Attribute Dialog.

To edit a parameter or custom attribute:

1 Select the object.

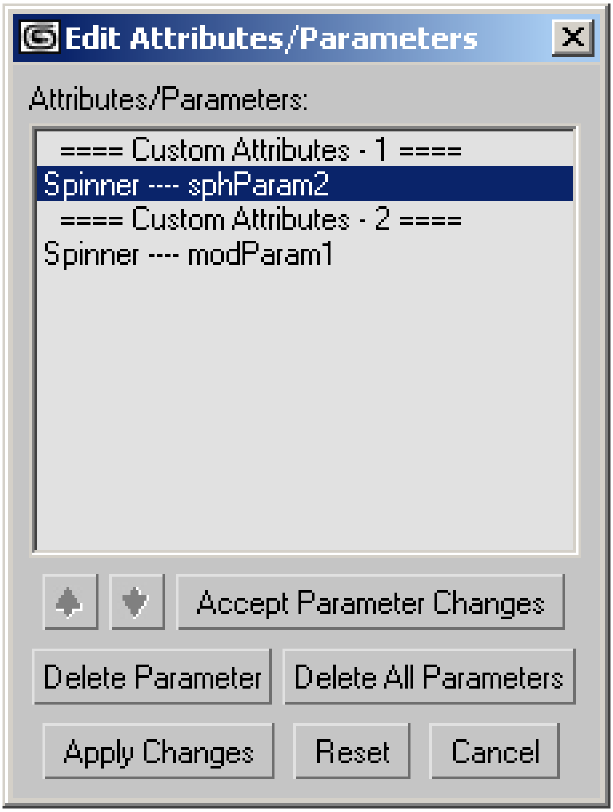
2 Choose Animation menu > Parameter Editor.

The Parameter Editor opens.



3 From the Add To Type drop-down list, choose the type of parameter to edit, and then click Edit/Delete.

The Edit Attributes/Parameters dialog opens.



NOTE If you chose Add To Type > Picked Track, the Track View Pick dialog might open first to prompt you to choose the animation track whose attribute to edit.

4 In the Edit Attributes/Parameters dialog, highlight the parameter to edit.

Its settings appear in the Parameter Editor.

5 Change the settings in the Parameter editor, and then click Accept Parameter changes.

6 With multiple parameters or custom attributes, to change the ordering, click the entity to move, and then use the up and down arrow buttons to move the entity in the list. Moving a Custom Attributes entry also moves its parameters.

NOTE You cannot rename a Custom Attributes entry.

7 When finished editing, click Apply Changes, and then exit the dialog by clicking the Close or Cancel button.

To delete a custom attribute or parameter:

1 Select the object.

2 Choose Animation menu > Parameter Editor.

 The Parameter Editor opens.

3 From the Add To Type drop-down list, choose the type of parameter to delete, and then click Edit/Delete.

 The Edit Attributes/Parameters dialog opens.

 NOTE If you chose Add To Type > Picked Track, the Track View Pick dialog might open first to prompt you to choose the animation track from which to delete the attribute.

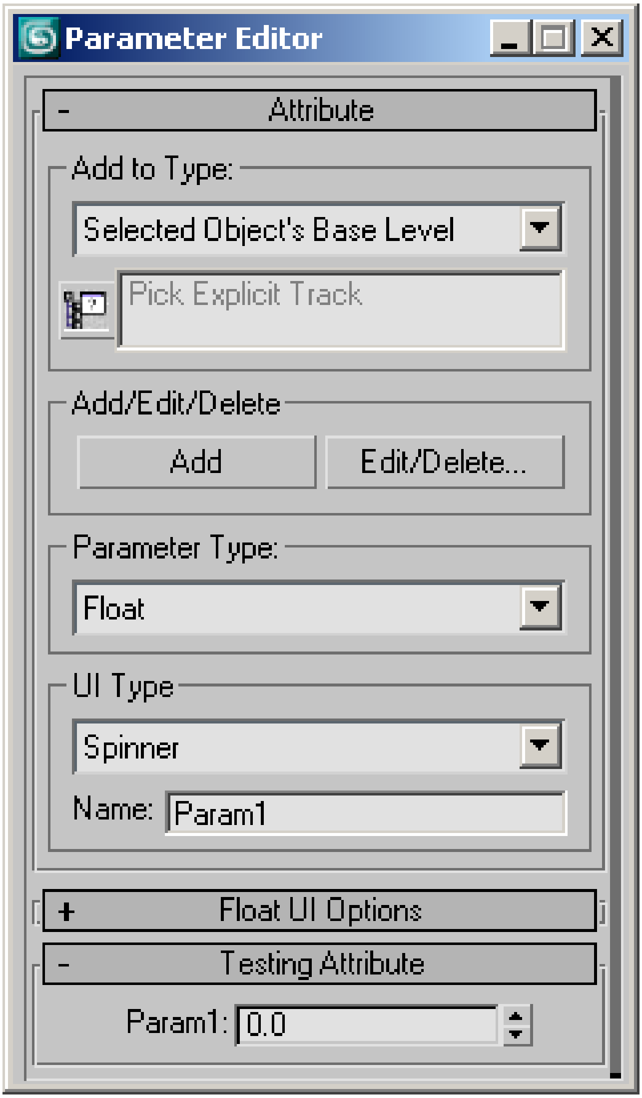
4 In the Edit Attributes/Parameters dialog, highlight the parameter to delete, and then click Delete Parameter. Alternatively, to delete all parameters under the same Custom Attributes heading as the highlighted parameter, click Delete All Parameters.

 To delete one or more custom attributes instead, click a Custom Attributes heading, and then click Delete Attribute or Delete All Attributes. Multiple attributes can result from collapsing an object's stack with Preserve Custom Attributes on. For example, one set of custom attributes might be applied to an object and a second set of attributes assigned to one of its modifiers. Collapsing such an object results in two sets of custom attributes.

5 Click Apply Changes, and then close the dialog by clicking its Close box or the Cancel button.

Interface

The Parameter Editor takes the form of a dialog with several rollouts: The first rollout sets general options for the attribute; the central rollout sets options for the current parameter type; and the third lets you preview the attribute user interface (UI).



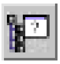
Attribute rollout

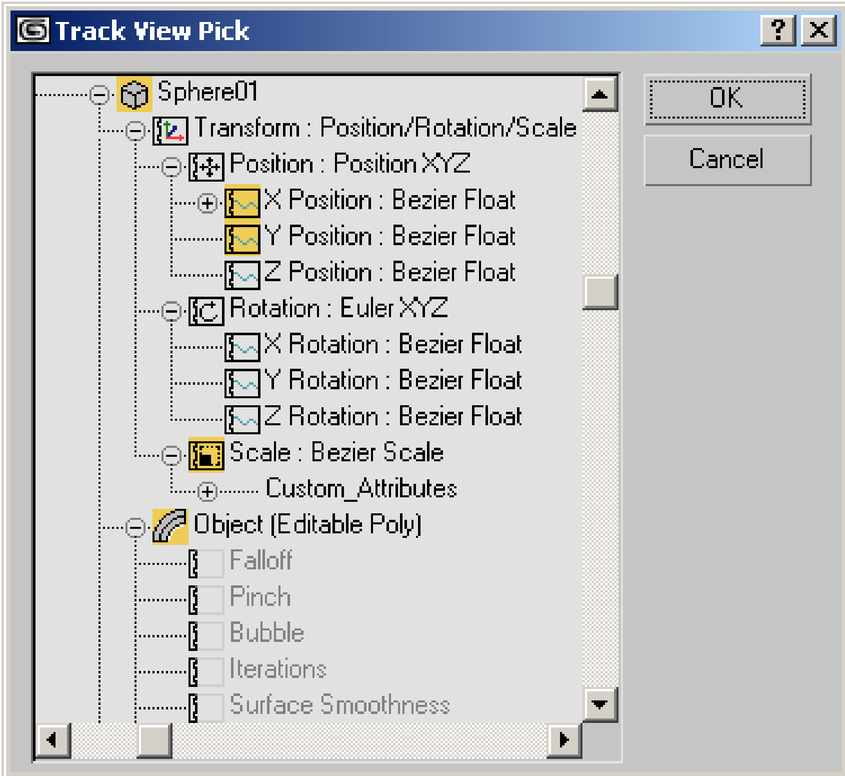
Add to Type group

Add to Type list Choose whether the custom attribute is assigned to the selected object, its active modifier (as highlighted in the modifier stack), its material, or a picked track. Also use this drop-down list to choose the attribute type to delete or edit.

If the text “Pick Explicit Track” appears in the box below the drop-down list before you choose Picked Track , the Track View Pick dialog appears showing the Track View hierarchy. Expand the hierarchy as necessary, click the track to add the attribute to, and then click OK.

NOTE If you choose Selected Object's Current Modifier and multiple modifiers are highlighted in the stack when you click Add, Parameter Editor applies the custom attribute to the first modifier you highlighted and removes the highlighting from the other modifiers.

 **[pick explicit track]** Click this button to open a Track View hierarchy window from which to choose an animation track. Navigate the hierarchy to find the desired track, highlight the track, and then click OK. The controller information for the track then appears in the text box to the right of the button, and the Parameter Editor will then use this track for adding or editing custom attributes and parameters.



Add/Edit/Delete group

Add Applies the custom attribute parameter to the current object, modifier, material, or track, depending on the current choice in the Add To Type list.

If a custom attribute parameter is assigned to an object or modifier, you can see and edit its value on the Modify panel after adding it by activating the entity to which the attribute is assigned. If the custom attribute is assigned to a material, it's available for that material in the Material Editor. To access a parameter that's assigned to an animation track, open Track View, highlight the track's Custom Attributes entry, and then right-click and choose View Attribute Dialog.

Edit/Delete Opens the Edit Attributes/Parameters dialog.

This dialog displays a list of all of the custom attribute parameters assigned to the currently object at the current level. Dialog behavior is described in these two procedures: To edit a parameter or custom attribute: on page 331 and To delete a custom attribute or parameter: on page 334.

Parameter Type group

Parameter Type Use the drop-down list to choose the data type for the current parameter. The following list includes links to the sections describing the UI settings for each parameter's data type:

- Angle: Spinner on page 339 or Slider on page 339
- Array on page 342
- Boolean: CheckBox on page 341 or CheckButton on page 341
- Color on page 344
- Float: Spinner on page 339 or Slider on page 339
- fRGBA on page 344
- Integer: Spinner on page 339 or Slider on page 339
- Material on page 345
- Node on page 344
- Percent: Spinner on page 339 or Slider on page 339
- String on page 346
- TextureMap on page 345
- WorldUnits: Spinner on page 339 or Slider on page 339

UI Type group

UI Type Selects the type of UI element that controls the parameter.

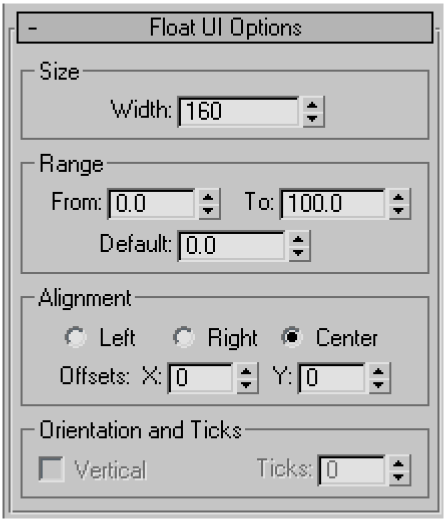
The UI types available depend on which parameter type you specify. For example, float and integer values are controlled by spinners or sliders, and Boolean values by check boxes or check buttons. Array values are always controlled by drop-down lists, node values by pick buttons, color values by color pickers, and texture map values by map buttons.

Full descriptions of each UI Options rollout follow, and the list of parameter types, above, includes links to the respective UI Options rollout descriptions.

Name The name of the parameter. Parameter Editor gives the parameter the default name Param#, with # being a number. Change the name by editing this field.

Angle/Float/Integer/Percent/WorldUnits UI Options rollout: Spinner

This is a numeric value that the user can set with a standard 3ds Max spinner.



Size group

Width Sets the width of the spinner.

Range group

From Sets the minimum value of the spinner.

To Sets the maximum value of the spinner.

Default Sets the default value of the spinner.

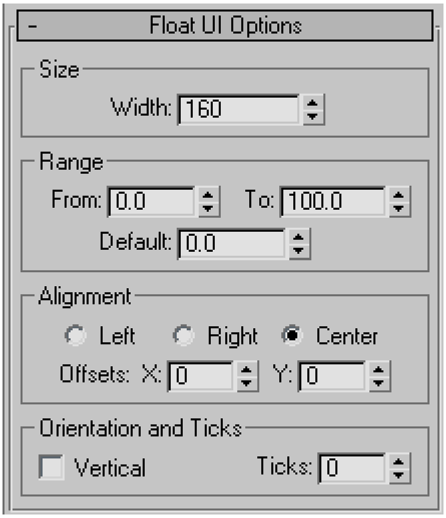
Alignment group

Left/Right/Center Sets the alignment of the spinner in the rollout.

Offsets X/Y Fine-tune the position of the spinner on the horizontal and vertical axes.

Angle/Float/Integer/Percent/WorldUnits UI Options rollout: Slider

This is a numeric value that the user can set with a standard 3ds Max slider.



Size group

Width Sets the width of the slider.

Range group

From Sets the minimum value of the slider.

To Sets the maximum value of the slider.

Default Sets the default value of the slider.

Alignment group

Left/Right/Center Sets the alignment of the slider in the rollout.

Offsets X/Y Fine-tune the position of the slider on the horizontal and vertical axes.

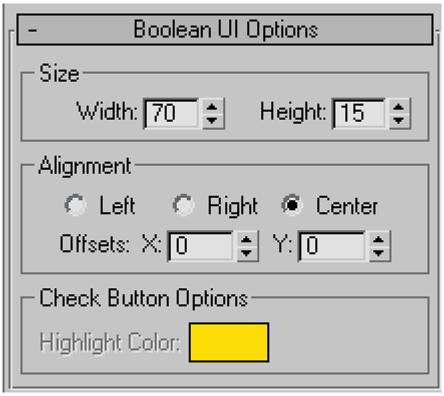
Orientation And Ticks group

Vertical When on, the slider will be displayed vertically. When off, the slider is displayed horizontally.

Ticks Sets the number of ticks along the slider. The ticks are distributed evenly along the length of the slider.

Boolean UI Options rollout: Check Box

This is a standard 3ds Max check box that the user can turn on and off by clicking it with the mouse.



Size group

Width Sets the width of the check box.

Height Sets the height of the check box.

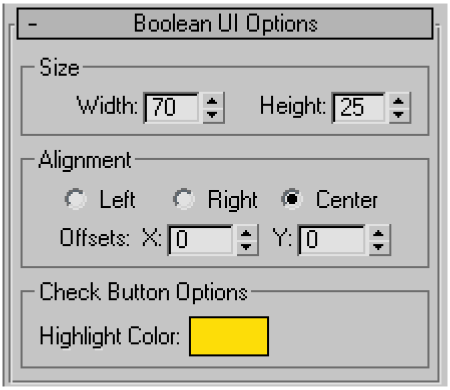
Alignment group

Left/Right/Center Sets the alignment of the check box in the rollout.

Offsets X/Y Fine-tune the position of the check box on the horizontal and vertical axes.

Boolean UI Options rollout: Check Button

This is a standard 3ds Max check button that the user can turn on and off by clicking it with the mouse.



Size group

Width Sets the width of the check button.

Height Sets the height of the check button.

Alignment group

Left/Right/Center Sets the alignment of the check button in the rollout.

Offsets X/Y Fine-tune the position of the check button on the horizontal and vertical axes.

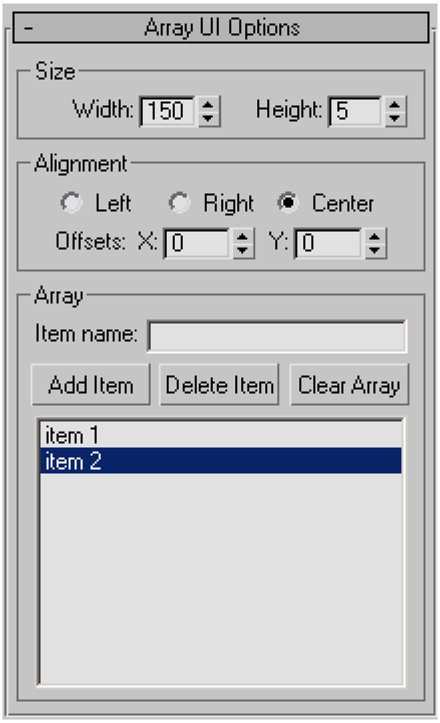
Check Button Options group

Highlight Color Sets the color of the button when it is pressed.

Array UI Options rollout: Drop-Down List/ComboBox/ListBox

This lets the user of the custom attribute choose a named option from a list. The options for the three Array UI types are the same; they differ in how they appear on the Custom Attributes rollout. The types are:

- Drop-Down List: Only the current choice is visible by default. The user clicks the field to open the list and then clicks to choose a different item.
- ComboBox: Displays an editable field above a list box. The user clicks to choose from the list, or edits the field.
- ListBox: Displays a list. The user clicks the desired item; the highlighting indicates the current choice.



Size group

Width Sets the width of the list.

Height Sets the height of the list.

Alignment group

Left/Right/Center Sets the alignment of the drop-down list in the rollout.

Offsets X/Y Fine-tune the position of the array list on the horizontal and vertical axes.

Array group

Item name Lets you enter a name into the list.

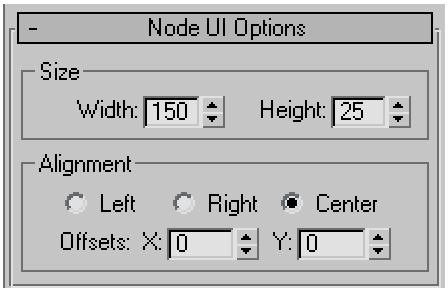
Click Add Item to add the name to the array list. To remove an item, highlight its name and click Delete Item. Click Clear Array to remove all items from the list.

[array list] Displays the contents of the list.

The item at the top of the array list is the default selection.

Node UI Options rollout: Pick Button

A node is any object in the 3ds Max scene. The Node UI element creates a button that, when clicked, lets the user pick a scene node other than the one to which the attribute is attached. After selecting the node, its name appears on the button.



Size group

Width Sets the width of the pick button.

Height Sets the height of the pick button.

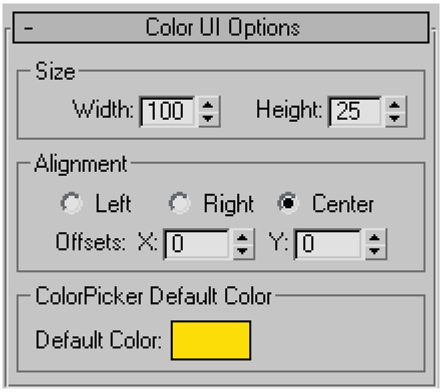
Alignment group

Left/Right/Center Sets the alignment of the pick button in the rollout.

Offsets X/Y Fine-tune the position of the pick button on the horizontal and vertical axes.

Color UI Options rollout: Color Picker

This creates a color swatch that displays the current color and lets the user click it to choose a new color with the Color Selector dialog.



Size group

Width Sets the width of the color picker.

Height Sets the height of the color picker.

Alignment group

Left/Right/Center Sets the alignment of color picker in the rollout.

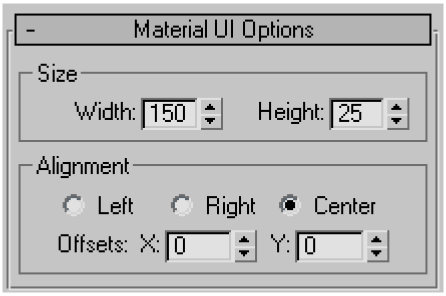
Offsets X/Y Fine-tune the position of the color picker on the horizontal and vertical axes.

ColorPicker Default Color group

Default Color Sets the default color of the color picker.

Material/TextureMap Options rollout: MaterialButton/MapButton

The options for the Material UI type (MaterialButton) and TextureMap UI type (MapButton) are the same. The difference is that, when the user clicks the resulting button to open the Material/Map Browser, the former displays only materials and the latter displays only maps.



Size group

Width Sets the width of the material/map button.

Height Sets the height of the material/map button.

Alignment group

Left/Right/Center Sets the alignment of the material/map button in the rollout.

Offsets X/Y Fine-tune the position of the material/map button on the horizontal and vertical axes.

String Options rollout: EditText

The String parameter type creates a text box that the user can edit with the keyboard, with optional default text.

Size group

Width Sets the width of the material/map button.

Height Sets the height of the material/map button.

Alignment group

Left/Right/Center Sets the alignment of the text box in the rollout.

Offsets X/Y Fine-tune the position of the text box on the horizontal and vertical axes.

Label group

Label above text box When off, the label (parameter name) appears to the left of the text box. When on, the label appears above the text box.

EditText Default Text group

Default Text Enter any default text that should appear in the box before the user edits it.

Testing Attribute rollout

This rollout displays the UI layout for the custom attribute that you are working on. It updates continuously so that you can see how different settings in the various rollouts affect the UI display of the attribute.

The UI element is operational in this rollout in the sense that it can be moved, clicked, toggled, and so on.

Parameter Collector

Animation menu > Parameter Collector

Keyboard > Alt+2

Parameter Collector lets you sort and present animatable parameters so that you can access and key selected parameter sets with a click or two. It takes the form of a resizable dialog that regenerates dynamically as parameters change. The dialog supports drag-and-drop rollout reordering. Collections are saved with their scenes and can be merged into other scenes.

One of Parameter Collector's most powerful features is the ability to change all parameters in a collection simultaneously, in an absolute or relative mode. For example, if you're animating a character's hand, you can use Parameter Collector to easily make all the fingers curl up together to form a fist.

NOTE Parameter Collector does not support parameters of externally referenced objects on page 6936 or objects in externally referenced scenes on page 6959.

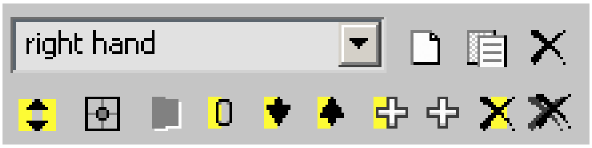
See also:

- Custom Attributes on page 329
- Attribute Holder Modifier on page 1206

Menu Bar

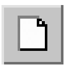
The menu bar provides a range of functions for using Parameter Collector. See Parameter Collector Menu Bar on page 356 for details. Also, you can open the Spinner Right-Click menu on page 3090 by right-clicking a numeric field in Parameter Collector.

Toolbar



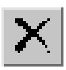
The Parameter Collector toolbar provides button access to the most commonly used functions.

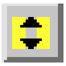
[collection name] If empty, enter a name for the current collection, or choose a different collection from the drop-down list. If a name appears and you edit it, pressing Enter duplicates the current collection with the new name.

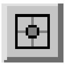
 **New Collection** Creates a new, empty collection, clearing the current collection name and the rollout area. You can restore any existing collection by choosing it from the drop-down list.

 **Duplicate Collection** Creates a new, unnamed collection containing the same data as the current collection. Enter a name for the duplicate selection in the editable field.

You can also duplicate a collection and name it at the same time by editing the name of an existing collection and pressing Enter.

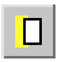
 **Delete Collection** Removes the current collection from memory.

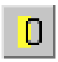
 **Multiple Edits** Enables multiple editing, in which changing the value of any selected parameter simultaneously changes all selected parameters of the same type by the same amount. This applies to both Absolute and Relative modes (see following).

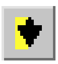
 **Absolute/Relative** Works the same way as the Absolute/Offset mode toggle on the Coordinate Display on page 7541. When Absolute is chosen, modifying a value changes it to the exact amount you specify. When Relative is chosen, the displayed value is 0, and modifying the parameter adds the specified change to the original value. The actual value appears only in Absolute mode.

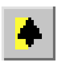
This applies to numeric values only; any changes to other values, such as color, are always absolute.

NOTE With multiple parameters selected, and Multiple Edits on, changing the value of a selected parameter changes the other selected parameter values *by* the same amount, not *to* the same amount. This happens in both Absolute and Relative modes.

 **Key Selected** Sets keys on page 8020 for selected parameters only at the current frame. Available only when Auto Key on page 7549 is on.

 **Reset Selected** Sets all selected numeric parameters to 0. Has no effect on other parameter types.

 **Move Parameters Down** Moves each selected parameter down one position within its rollout, if possible.

 **Move Parameters Up** Moves each selected parameter up one position within its rollout, if possible.

 **Add to Selected Rollout** Lets you add new parameters to the selected rollout. Click this button to open the Track View Pick dialog, and then choose the parameters from the dialog.

NOTE You can add several parameters at once by highlighting them in the dialog before clicking OK.

 **Add to New Rollout** Lets you add new parameters to a new rollout. Click this button to open the Track View Pick dialog, and then choose the parameter from the dialog. Parameter Collector creates a new rollout to hold the parameters.

NOTE You can add several parameters at once by highlighting them in the dialog before clicking OK.

 **Delete Selected** Deletes all selected parameters.

 **Delete All** Deletes all parameters and rollouts.

Rollouts

Rollouts work the same way in Parameter Collector as they do on the command panel on page 72. You can expand and collapse a rollout by clicking its title bar, and move it to another location by dragging the title bar. You can resize the dialog to be able to see all rollouts at once.

Only one rollout can be selected at a time. You select a rollout by clicking the horizontal bar beneath the title; when selected, this bar is orange-yellow in color, and angle brackets surround the rollout title (for example, "> Hand Parameters <").

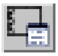
The interface for each parameter on a rollout is as follows:

[Select Parameter] A small check button on the left side of the rollout. Click it to toggle the parameter's selection status. When selected, the button appears pressed in and is colored yellow-orange.

[parameter name] By default, the parameter has the same name as is shown in Track View, but you can change it with the Edit menu > Edit Notes command. You can see the default name for a parameter as well as the object it controls, if any, by hovering the mouse over the parameter name; the information appears on a tooltip.

[parameter value] Shows the current value of the parameter. The parameter type determines how this appears: numeric field/spinner, color swatch, etc. You can edit the value the same way as on the command panel or a dialog.

If a key exists for the value at the current frame, the spinner or swatch appears with red brackets at the corners.

 **[Properties]** Opens a Key Info dialog on page 3127 for the parameter. Available only if the parameter has an animation controller.

Use the Key Info dialog to edit an animation key's value, time, and interpolation methods.

Procedures

Example: To use Parameter Collector:

This exercise demonstrates some basic Parameter Collector functions.

In general, start with a scene containing one or more objects whose parameters you'll collect. Ideally, they should be animated, but it's not absolutely necessary.

1. For this example, reset 3ds Max and then add a sphere.
2. Open Parameter Collector from the Animation menu, or press Alt+2.
3.  On the Parameter Collector toolbar, click the Add To New Rollout button.

   The Track View Pick dialog opens. This lets you specify parameters to collect.
4. On the Track View Pick dialog, expand Objects > Sphere01 > Transform: Position/Rotation/Scale > Position: Position XYZ.
5. Click X Position: Bezier Float.

   The parameter highlights.
6. Click OK to close the dialog.

   A new rollout named Parameters 1 appears, containing the X Position parameter.

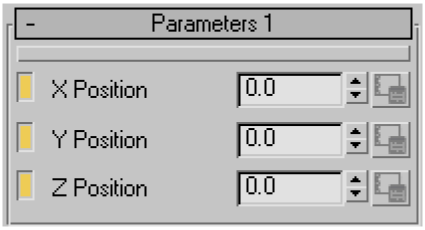
   NOTE A parameter in Parameter Collector can contain only a single value (e.g., float, integer, color), so the software doesn't let you add parameters such as Position: Position XYZ, which contains three distinct values.
7.  Click Add To Selected Rollout and then use the same method to add the Y Position and Z Position parameters: Highlight both parameters on the Track View Pick dialog and then click Add To Selected Rollout to add both at once to the Parameters 1 rollout.

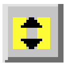
8 Drag each spinner in turn to move the sphere on the respective axis.

As you change each parameter value, the sphere moves in real time in the viewports.

9 Set all three parameters to **0.0**.

10 Click the check button next to each parameter to select all three.



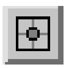
11  On the toolbar, turn on Multiple Edits.

12 Drag one of the spinners.

All three change by the same amount, so that the sphere moves diagonally in the scene.

13 Use the keyboard to change the Y Position value to **30.0**.

Again, the other two change.

14  Click the Absolute/Relative button to turn on Relative mode.

All the values display 0.0, as with Offset mode on the status bar coordinate display.

15 Use the spinner to set Y Position to **0.65**.

The three change in unison, and then reset back to 0.0 when you release the mouse button. This has added the value you set to each of the three positions, as you'll see in the next step.

16 Click the Absolute/Relative button to return to Absolute mode.

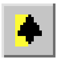
The values are all set to 30.65, reflecting the relative change that you made.

Next, you'll try a few Edit commands.

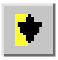
17 Click the Y Position check button to deselect the parameter.

18 From the Edit menu, choose Select Invert.

The Y Position parameter is now selected, and the other two are deselected.

19  Click the Move Parameters Up button.

The Y Position parameter now sits above the X Position parameter.

20  Click the Move Parameters Down button.

The Y Position returns to its position below the X Position parameter.

21 Choose Edit > Edit Notes.

The Notes dialog opens. Here you can change the parameter name, set a URL or file location with further information about the parameter, and enter comments.

22 In the box below Parameter Name, type **Sphere Y Loc.**, and then press Enter.

The new name replaces the old one on the rollout. You can see the original name by hovering the mouse cursor over the parameter name; it appears on a tooltip.

To conclude this exercise, you'll use Parameter Collector to set and edit animation keyframes.

23 On the Collection menu, turn on Show Keys In Track Bar if necessary.

24 

On the 3ds Max status bar, turn on Auto Key.

25 Change the Sphere Y Loc. (the old Y Position) parameter value to **20.0**.

Because you're at frame 0, no key is set. This is the same way Auto Key works normally.

26 Go to frame 20 and then set Sphere Y Loc. to **30.0**.

This sets animation keys at frames 0 and 20.

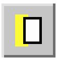
27 Right-click the key at frame 20.

The menu shows that a key exists at frame 20 only for Y Position. Normally, 3ds Max would create keys for all three axes, even if you moved the sphere only on one axis.

Parameter Collector can set keys for unselected objects as well.

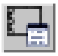
28 Click in an empty area of the active viewport to deselect the sphere, and then go to frame 30 and change the Sphere Y Loc. value to **40.0**.

This sets another key for Y Position at frame 30.

29  In Parameter Collector, select the X Position and Z Position parameters, and then click Key Selected.

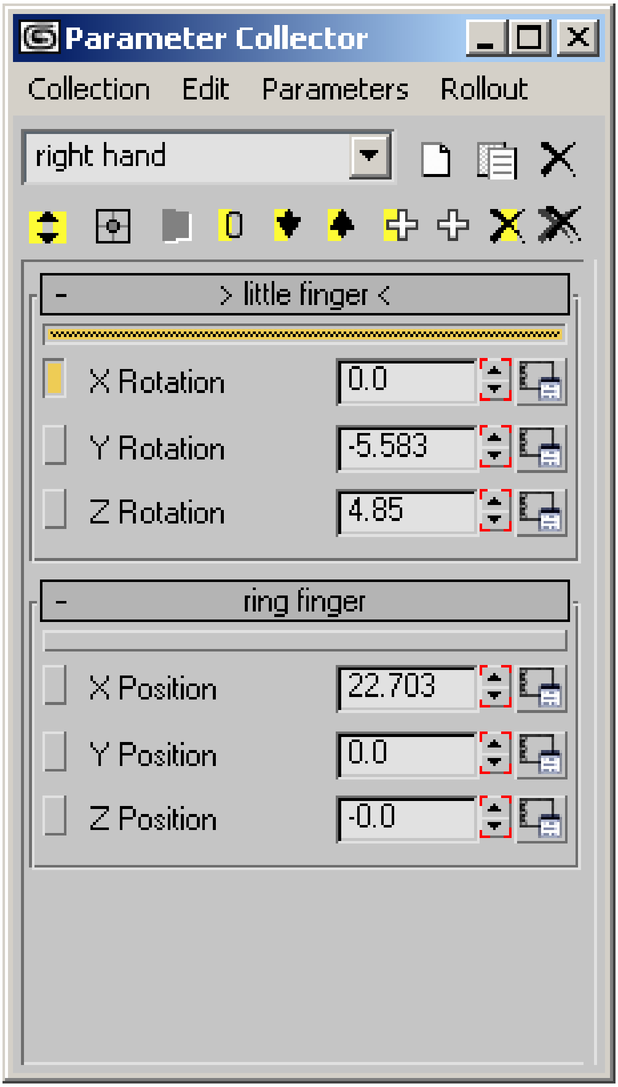
This button is available only when Auto Key is on.

30 Check the track bar key again. Now there are keys for all three parameters, as demonstrated by the red brackets on the spinners in Parameter Collector.

31  Click the Properties button to the right of the Sphere Y Loc. parameter.

This opens a Key Info dialog on page 3127 for the parameter, with the ability to edit the key time and value as well as interpolation with other keys. The dialog is also available from the track bar right-click menu, but it's much easier to access the data for a specific key from Parameter Collector.

Interface



Parameter Collector takes the form of a dialog with a menu bar, a toolbar, and rollouts that you create and modify using the dialog tools. You can resize the dialog horizontally and vertically; expanding it lets you see all rollouts simultaneously.

Parameter Collector Menu Bar

Animation menu > Parameter Collector > Parameter Collector menu bar

Keyboard > Alt+2 > Parameter Collector menu bar

The Parameter Collector dialog menu bar provides access to a number of important commands. Some of these commands are replicated on the dialog toolbar; others, such as the Select tools, are available only from the menus. Also, you can open the Spinner Right-Click menu on page 3090 by right-clicking a numeric field in Parameter Collector.

Interface

Collection menu

The first three items in this menu are unavailable until you enter a name for the current collection in the editable field (drop-down list) just below the menu bar.

New Collection Creates a new, empty collection, clearing the current collection name and the rollout area. You can restore any existing collection by choosing it from the drop-down list.

Duplicate Collection Creates a new, unnamed collection containing the same data as the current collection. Enter a name for the duplicate selection in the editable field.

Delete Collection Removes the current collection from memory.

Show Keys in Track Bar Displays in the track bar on page 7531 animation keys for all objects with parameters in the current collection, whether or not the objects are selected in the scene.

Isolate Keys in Track Bar The track bar displays only keys for parameters in the Parameter Collector.

Show Selected Keys in Track Bar Displays in the track bar on page 7531 animation keys for all objects with selected parameters in the current collection, whether or not the objects are selected in the scene.

Isolate Selected Keys in Track Bar The track bar displays only keys for selected parameters in the Parameter Collector.

Put to Object Stores the current collection as part of an object in the scene. Opens the Put To Object dialog; highlight an object in the list, and then click Pick.

Although parameter collections are stored with the scene in which they're created, you can use this function to transfer a collection to a different scene. After putting the collection to an object, save the scene. Open or create another scene, merge the object from the saved scene to the new one, and then use Get from Object.

You can also back up, organize and streamline parameter collections by putting and getting different collections to and from various objects in your scene. Just remember that if you add, reorder, or remove parameters or rollouts to a collection that has been put to an object, you must then put it to the object again so the changes are saved to the collection.

Link to Object Stores the current collection *using a live link* as part of an object in the scene. Any change to the collection instantly updates the version of the collection stored in the object. Opens the Link To Object dialog; highlight an object in the list, and then click Pick.

Link to Object has basically the same function as Put To Object (see previous entry), except that it guarantees an up-to-date stored version of the collection, especially when merging the object into another scene that is a common production workflow.

NOTE Only one “linked-to” object can be active in a scene, but you can use Put To Object on any number of objects at a time.

Get from Object Retrieves a collection that you stored with Put To Object or Link To Object.

Remove from Object Deletes a collection that you stored with Put To Object or Link To Object.

Edit menu

Parameter Collector lets you select parameters in any combination, but you can select no more than one rollout at a time. To select or deselect a parameter, click the small button on its left side. To select or deselect a rollout, click the wide horizontal button just below the rollout title. Selecting a rollout deselects any other selected rollout.

Select All Selects all parameters and rollouts.

Select All Rollout Selects all parameters on the current rollout. Unavailable if no rollout is selected.

Select None Deselects all parameters.

Select Invert Inverts the current selection of parameters.

Delete Selected Deletes all selected parameters.

Delete All Deletes all parameters and rollouts.

Multiple Edits Enables multiple editing, in which changing any parameter simultaneously changes all selected parameters of the same or similar type.

NOTE The changed parameter need not be selected.

Absolute/Relative This works the same as the Absolute/Offset mode toggle on the Coordinate Display on page 7541. When Absolute is chosen, modifying a value changes it to the exact amount you specify. When Relative is chosen, the displayed value shows 0, and modifying the parameter adds the specified change to the original value. This applies to numeric values only; changes to any other value types such as color are always absolute.

Edit Notes Opens a single Notes dialog on page 359 for all selected parameters.

You can open the Notes dialog for a single parameter by right-clicking its Select Parameter button.

Parameters menu

Add to Selected Lets you add new parameters to the selected rollout.

Add to New Rollout Lets you add new parameters to a new rollout.

Move Up Moves selected parameters up one position within their rollout, if possible.

Move Down Moves selected parameters down one position within their rollout, if possible.

Move Up By Rollout Moves selected parameters to the rollout above, if possible. If the same parameter already exists in the rollout above, the selected parameter is simply deleted.

Move Down By Rollout Moves selected parameters to the next rollout, if possible. If the same parameter already exists in the next rollout, the selected parameter is simply deleted.

Key All Sets keys on page 8020 for all parameters at the current frame. Available only when Auto Key on page 7549 is on.

Key Selected Sets keys on page 8020 for selected parameters only at the current frame. Available only when Auto Key on page 7549 is on.

Reset All Sets all numeric parameters to 0. Has no effect on other parameter types.

Reset Selected Sets all selected numeric parameters to 0. Has no effect on other parameter types.

Rollout menu

NOTE While there are no menu commands for moving rollouts, you can do so simply by dragging the rollout title bar to a new location.

New Rollout Creates a new, empty rollout.

New Rollout Selected Parameters Creates a new rollout and populates it with copies of any selected parameters.

Rename Rollout Opens a small dialog that lets you rename the selected rollout.

Delete Rollout Deletes the selected rollout.

Delete Rollout Move Up Deletes the selected rollout and moves its parameters to the rollout above.

Delete Rollout Move Down Deletes the selected rollout and moves its parameters to the rollout below.

Notes Dialog (Parameter Collector)

Parameter Collector > Select one or more parameters. > Parameter Collector menu bar > Edit menu > Notes

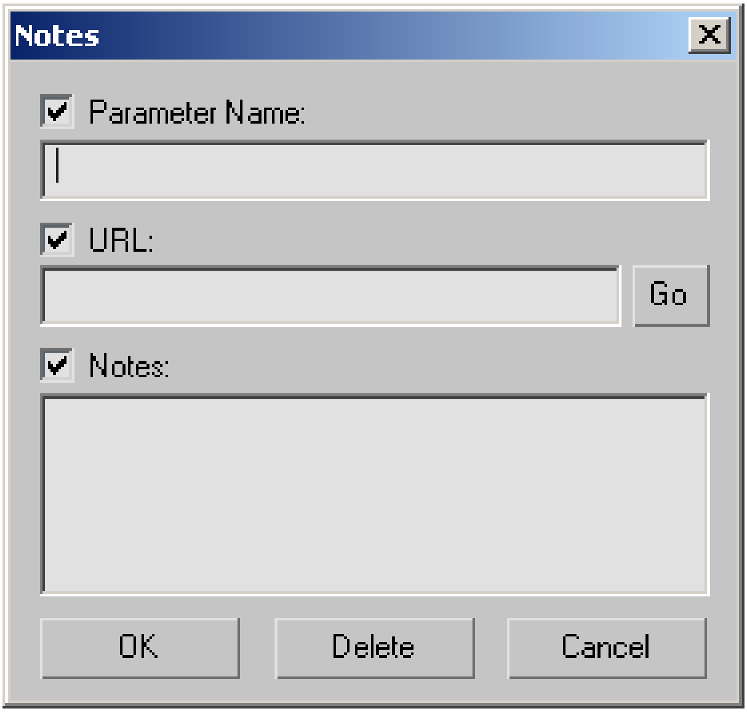
Parameter Collector > Right-click a Parameter Select button.

The Notes dialog lets you enter a name, URL, and comments for one or more selected parameters in Parameter Collector.

Choosing Notes from the Edit menu with multiple parameters selected opens a single dialog common to all selected parameters. Right-clicking a Parameter Select button opens a dialog for that parameter only.

When you open Notes from the Edit menu with multiple parameters selected, if the text contents for a box in all selected parameters are the same (or null), its check box is on, indicating that changes to the text will apply to all selected parameters. If a text box has different contents for different selected parameters, the check box is off, and the corresponding text box is empty and unavailable, preventing any changes. If you turn on a check box, you can edit the text, and changes will be applied to all selected parameters.

Interface



The Notes dialog interface comprises three text boxes, each with its respective check box, and a button. By default, the text boxes are empty; you can enter any text into each box, although each has a specific purpose, as described below.

Parameter Name Lets you change the parameter name shown in Parameter Collector.

By default, the parameter name displayed in Parameter Collector is the same as the name that appeared in the Track View Pick dialog when you added it to Parameter Collector. If you enter a different name in the Notes dialog, Parameter Collector then displays that name. The changed name is used only in Parameter Collector; elsewhere, such as the Modify panel, it remains the same as before.

You can see the original parameter name, as well as the object to which it's attached, by hovering the mouse over the parameter name in Parameter Collector; the information appears in a tooltip.

URL Lets you enter a URL, such as www.discreet.com.

This could be a link to a Web or intranet page, or even a network location or file pertaining to the selected parameter. To access the link, click the Go button.

Go If the URL text box contains a valid URL, clicking Go opens the URL in a separate browser window.

Notes Contains any comments on the parameters. This field is strictly for informational purposes.

Expression Techniques

In 3ds Max, you can use mathematical expressions (rather than constant numbers) to express parameter values. For example, you could use the expression 24*6 to represent the number 144.

You can use mathematical expressions to control the following object properties:

- Object parameters, such as length, width, and height
- Transform and modifier values, such as an object's position coordinates

Parameter wiring on page 3322, the expression controller on page 3154, and the numerical expression evaluator on page 73 all use expressions, which are described in this topic.

An expression is a mathematical function that returns a value. You can use expressions to control the following scene elements:

Scene element	Calculatable property
Creation parameters	Any numeric creation parameter
Transforms	Position [X, Y, Z] X Rotation Y Rotation Z Rotation Scale [X%, Y%, Z%]
Modifiers	Any numeric modifier parameter (including creation parameters)

Scene element	Calculatable property
Materials	Colors [R, G, B] Any numeric material parameter

NOTE Expressions only work with the individual XYZ components of Euler rotation. You can't assign an expression to TCB rotation or other kinds of rotation controllers.

The links below are to the sections that follow in this topic.

Expression Return Types on page 362

Operators on page 362

Variables on page 365

Functions on page 367

See also:

- Trigonometric Functions on page 370
- Vectors on page 373
- Expression Controller Techniques on page 3161

Expression Return Types

The type of value returned by an expression depends on the kind of controller:

- Float expressions return a floating-point scalar value (For example, 5.617). Scalars are used in the animation controllers of numeric parameters.
  If the parameter has an integer value, the expression rounds the float value to the nearest integer.
- Position, Scale, and Point3 expressions return a three-component vector. For example, [5, 18, 24]. The vector can represent an object's X,Y,Z location, percent scaling in X, Y, and Z, or a color (RGB values) in a material.

Operators

In the following tables, p and q are any scalar value or expression, V and W are any vector value or expression. (The character "x" is used as the vector cross-product operator.)

Scalar Operators

These are the arithmetic operators for scalar values:

Operator	Use	Meaning
+	p+q	Addition
-	p-q	Subtraction
-	-p	Additive inverse
*	p*q	Multiplication
/	p/q	Division
^	p^q	power (p to the power of q)
**	p**q	^ and ** are the same operation

You can also use logical (Boolean) operators with scalar values. These operators all return 1 if true, 0 otherwise:

Operator	Use	Meaning
=	p=q	equal to
<	p<q	less than
>	p>q	Greater than
<=	p<=q	less than or equal to
>=	p>=q	Greater than or equal to
|	p|q	Logical OR, returns 1 if either p nonzero; otherwise, returns 0

Operator	Use	Meaning
&	p&q	Logical AND, returns 1 if p and q are nonzero; otherwise, returns 0

TIP Logical operators are useful with the "if" function.

Vector Operators

For vectors that have a variable name, you can use a special component operator (.) to refer to the three scalar components of the vector:

Use	Meaning
V.x	first component (X)
V.y	second component (Y)
V.z	third component (Z)

These are the operators for vector arithmetic:

Operator	Use	Meaning
+	V+W	Addition
-	V-W	subtraction
*	p*V	scalar multiplication
*	V*p	scalar multiplication
*	V*W	dot product
X	VxW	cross product
/	V/p	scalar division

Operator Precedence

Expressions have eight levels of precedence. The higher the operator is on the list, the earlier it is evaluated.

<table>
<tr><th>Operator</th><th>Level of Precedence</th></tr>
<tr><td>- +</td><td>as unary operators, as in -8, +25</td></tr>
<tr><td>.</td><td>the component operator, as in V.x</td></tr>
<tr><td>** ^</td><td></td></tr>
<tr><td>X</td><td>cross product</td></tr>
<tr><td>* /</td><td></td></tr>
<tr><td>+ -</td><td></td></tr>
<tr><td>= < > <=
>=</td><td></td></tr>
<tr><td>| &</td><td></td></tr>
</table>

Parentheses are a special case. They are a grouping or subexpression operator that is provided so you can override the precedence order of the other operators.

Variables

In expressions you write for expression controllers on page 3154, variables are represented by symbolic names. You create them to contain constant or variable values in your expressions. Several predefined variables are also provided. Some of these have a constant value, others can vary.

In expressions used for parameter wiring on page 3322 and the numerical expression evaluator on page 73, you can use predefined variables with constant values.

Predefined Variables with Constant Values

These are the predefined variables that have a constant value (variable names are case-sensitive):

Variable Name	Constant Value	Use
pi	3.14159	Ratio of a circle's circumference to its diameter.
e	2.71828	Base of natural logarithms.
TPS	4800	Ticks per second. The tick is the basic time unit of 3ds Max animation.

Predefined Variables with Variable Values

These are the predefined variables that have a variable, time-based value (variable names are case-sensitive).

Variable Name	Meaning
F	Frame number. For each frame, F equals the current frame number, counting from zero. The range of frames can vary depending on the number of frames in the active time segment.
NT	Normalized time. By definition, normalized time (NT) ranges from 0 to 1 over the active time segment, regardless of how many frames are in the segment. If you base an expression on NT, its effect happens exactly once over the range. You can also multiply NT by a factor for the expression's effect to occur a certain number of times (for example, 2*NT causes the expression's effect to occur twice). Expressions based on NT

Variable Name	Meaning
	speed up or slow down if you change the length of the time segment.
S	Seconds (elapsed time in seconds). Elapsed time is measured from the first frame to the current frame. The range of seconds can vary depending on the total time of the active time segment.
T	Ticks (elapsed time in ticks). There are 4800 ticks per second. Elapsed time is measured from the first frame to the current frame. The range of ticks can vary depending on the total time of the active time segment.

Rules for Variable Names

- Variable names can contain as many alphanumeric characters as you like. Their length is not limited.
- Variable names cannot contain spaces.
- The variable name must begin with a letter. Numbers are valid within a variable name (as in "Pos1" or "M23").
- Variable names are case-sensitive. For example, "pos", "Pos", and "POS" designate three different variables.
- You can't create a variable with a name that duplicates another name, including the variable names that are predefined.

Functions

Following is a list of the functions provided for expressions. In this list, p, q, and r represent scalar values or scalar expressions. V and W represent vector values or vector expressions.

To use a function in an expression, enter the name of the function and appropriate arguments to it.

Trigonometric Functions

The sine, cosine, and tangent functions take an angle in degrees and return a floating-point value. The arc functions take a floating-point value and return a value in degrees.

Function	Meaning
sin(p)	sine
cos(p)	cosine
tan(p)	tangent
asin(p)	arc sine
acos(p)	arc cosine
atan(p)	arc tangent

Hyperbolic Functions

Hyperbolic functions take a floating-point value and return a floating-point value.

Function	Meaning
sinh(p)	hyperbolic sine
cosh(p)	hyperbolic cosine
tanh(p)	hyperbolic tangent

Conversion Between Radians and Degrees

Function	Meaning
radToDeg(p)	takes p in radians and returns the same angle in degrees

Function	Meaning
degToRad(p)	takes p in degrees and returns the same angle in radians

Rounding Functions

Function	Meaning
ceil(p)	smallest integer greater than or equal to p
floor(p)	largest integer less than or equal to p

Standard Calculations

Function	Meaning
ln(p)	natural (base e) logarithm
log(p)	common (base 10) logarithm
exp(p)	exponential function exp(p)=e^p
pow(p,q)	p to the power of q (p^q)
sqrt(p)	square root
abs(p)	absolute value
min(p,q)	minimum returns p or q, depending on which is smaller
max(p,q)	maximum returns p or q, depending on which is greater
mod(p,q)	remainder of p divided by q

Conditional Functions

Function	Meaning
if(p,q,r)	works like the common spreadsheet "if" (If p is nonzero then "if" returns q, otherwise "if" returns r.)
vif(c,V1,V2)	"Vector If" (Value is V1 if c is true, else V2.)

Vector Handling Functions

Function	Meaning
length(V)	length of V
comp(V,i)	i'th component (I=0,1,2): comp([5,6,7],1)=6
unit(V)	returns a unit vector in the same direction as V

NOTE The comp function is an alternative to the notation V.x, V.y, V.z.

Special Animation Function

Function	Meaning
noise(p,q,r)	3D noise: returns a randomly generated position

The arbitrary values p, q and r, are used as a random-generation seed. You can reuse these values to ensure that noise() returns the same value.

Trigonometric Functions

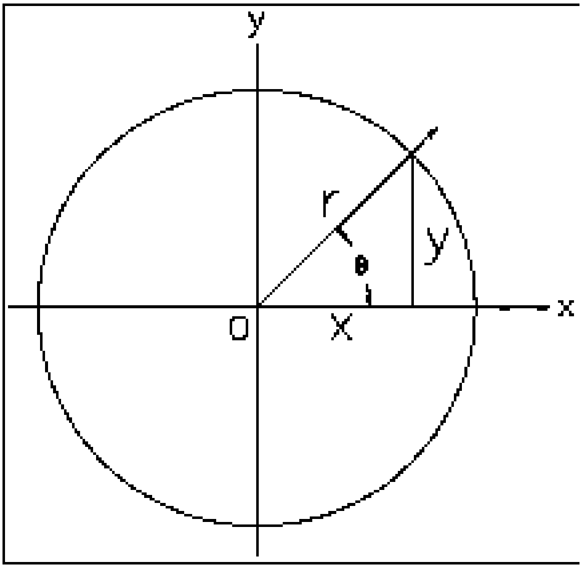
This topic is a quick review for readers who need a reminder about this area of mathematics. If you're familiar with trigonometry, you can skip this topic.

If you find this topic difficult to follow, you might consult a more basic reference on mathematics.

Trigonometric functions are principally used to model or describe:

- The relation between angles in a triangle (hence the name).
- Rotations about a circle, including locations given in polar coordinates.
- Cyclical or periodic values, such as sound waves.

The three basic trigonometric functions are derived from an angle rotating about a unit circle.



Trigonometric functions based on the unit circle

$$\sin\theta = \frac{y}{r} \quad \cos\theta = \frac{x}{r} \quad \tan\theta = \frac{y}{x}$$

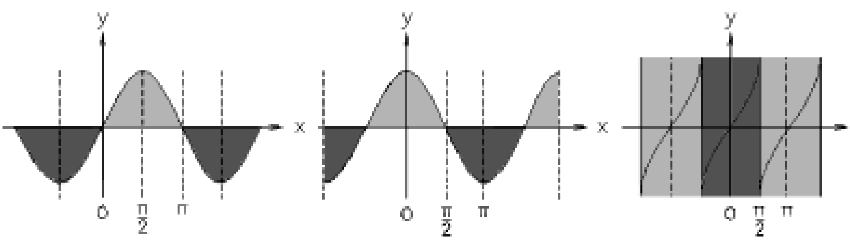
The tangent function is undefined for x=0. Another way to define the target is:

$$\tan\theta = \frac{\sin\theta}{\cos\theta}$$

Because *XYR* defines a right-angled triangle, the relation between the sine and cosine is:

$$(\cos\theta)^2 + (\sin\theta)^2 = 1$$

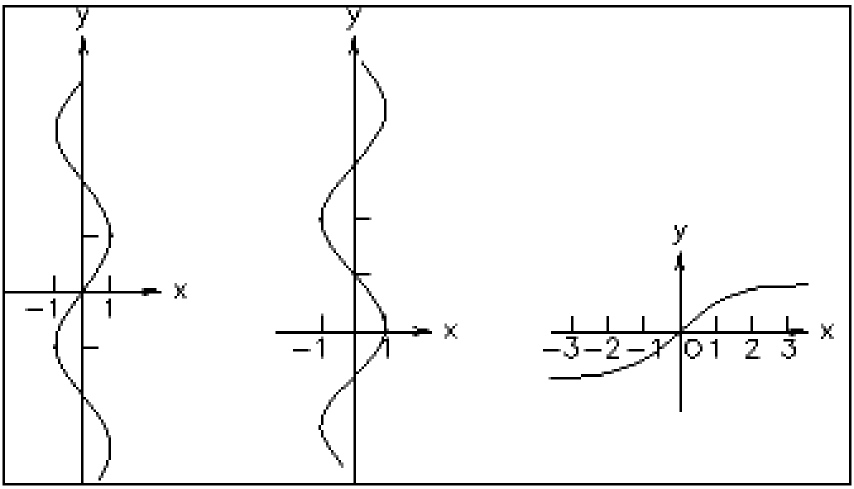
The graphs of the basic trigonometric functions illustrate their cyclical nature.



Graphs of basic trigonometric functions

The sine and cosine functions yield the same values, but the phase differs along the X axis by /2: in other words, 90 degrees.

The inverse functions for the trigonometric functions are the arc functions; the inverse only applies to values of x restricted by – /2 X /2. The graphs for these functions appear like the basic trigonometric function graphs, but turned on their sides.



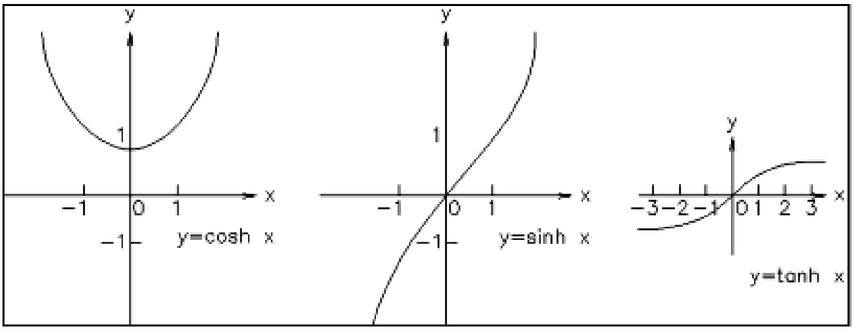
Graphs of basic arc functions

The hyperbolic functions are based on the exponential constant *e* instead of on circular measurement. However, they behave similarly to the trigonometric functions and are named for them. The basic hyperbolic functions are:

$$\sinh x = \frac{e^x - e^{-x}}{2}$$

$$\cosh x = \frac{e^x + e^{-x}}{2}$$

$$\tanh x = \frac{e^x - e^{-x}}{e^x + e^{-x}} = \frac{\sinh x}{\cosh x}$$



Graphs of basic hyperbolic functions

Vectors

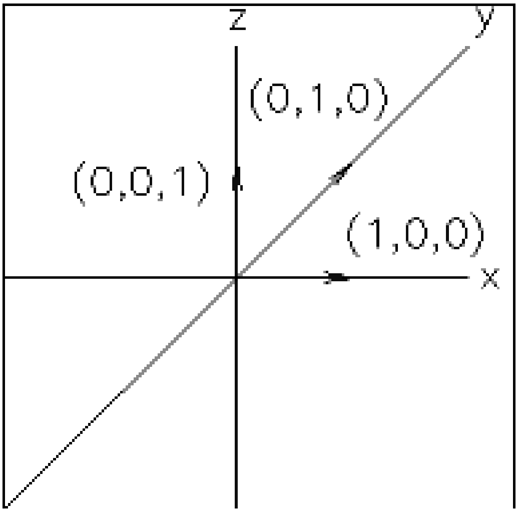
This topic is a quick review for readers who need a reminder about vector arithmetic. If you're familiar with vectors and vector calculations, you can skip this topic. If this topic is difficult to follow, you might consult a more basic reference on mathematics.

A vector expresses a length and a direction in a particular space. The vector is expressed as a point; for example, [5, 5, 7]. The length is the distance from the origin to that point, and the direction is similarly from the origin to (and through) the point.

In 3ds Max, vectors have three values and describe positions in three-dimensional space. They can also represent percent scaling in X, Y, and Z; and (more abstractly) describe locations in RGB color space.

Unit Vectors and Basic Vectors

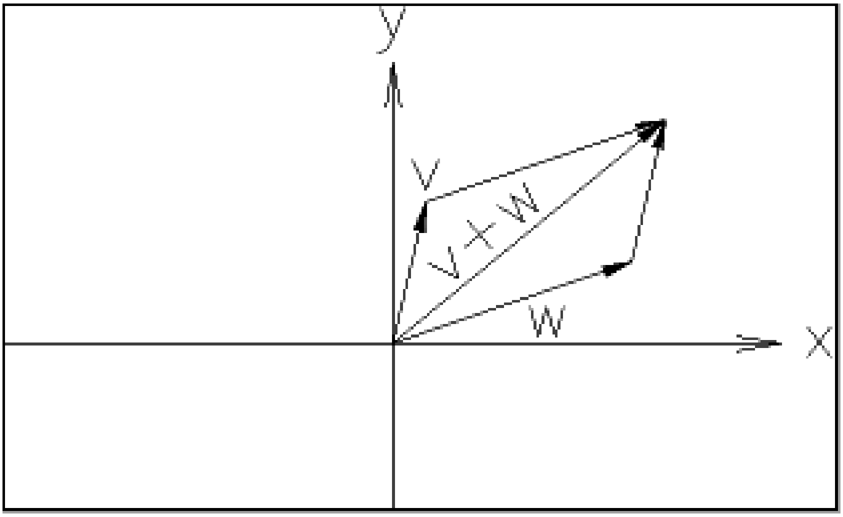
A unit vector has a length of one. Unit vectors are often used to express direction only. The three basic vectors are unit vectors that describe the three axes (X, Y, and Z) of 3D space.



Basic vectors and the XYZ axes

Adding and Subtracting Vectors

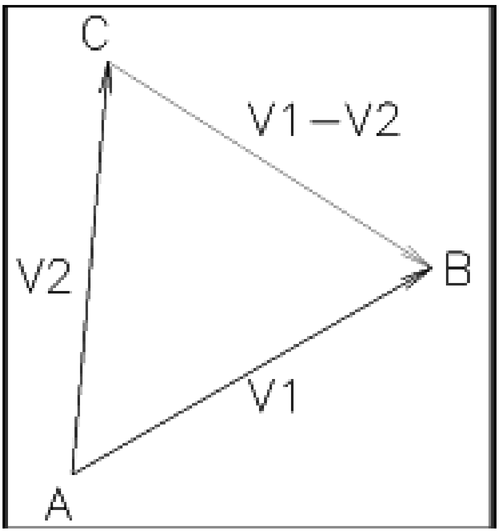
Adding two vectors creates a new vector that combines the length and direction of the original two. Vector addition is commutative: V+W=W+V.



Adding two vectors

$$V + W = [v_1 + w_1, v_2 + w_2, v_3 + w_3]$$

Subtracting two vectors gives the vector between the two points.



Subtracting two vectors

$$V - W = [v_1 - w_1, v_2 - w_2, v_3 - w_3]$$

Scalar Multiplication and Division

Multiplying a vector by a scalar changes the vector's length, as does dividing the vector by a scalar.

$$yV = Vy = [yv_1, yv_2, yv_3]$$

$$V/y = [v_1/y, v_2/y, v_3/y]$$

Vector Length and Direction

The length of a vector is obtained from the Pythagorean theorem.

$$|V| = \sqrt{v_1^2 + v_2^2 + v_3^2}$$

In 3ds Max expressions, the length() function returns this value.

The direction of the vector is the vector divided by its length; this gives you a unit vector with the same direction.

$$\frac{V}{|V|}$$

The distance between two points is the length of the vector between them.

$$|VW| = |W - V|$$

Subtracting vectors to obtain a distance

Creating Geometry

8

The solid 3D objects in the scene, and the objects used to create them, are known as *geometry*. Usually, geometry comprises the subject of your scene and the objects that you render.

This section describes the types of geometry you can create using the Create panel on page 7631.

See also:

- Surface Modeling on page 2007
- Space Warps and Particle Systems on page 2685

Basics of Creating and Modifying Objects

This section provides an introduction to techniques for creating and modeling objects.

The Create panel on page 7631 contains controls for creating new objects, the first step in building a scene. Despite the variety of object types, the creation process is consistent for most objects.

The Modify panel on page 7633 provides controls to complete the modeling process. You can rework any object, from its creation parameters to its internal geometry. Both object-space and world-space modifiers let you apply a wide range of effects to objects in your scene. The modifier stack allows editing of the modifier sequence.

In 3ds Max, you model basic parametric on page 8080 objects into more complex ones by:

- Changing parameters on page 8080
- Applying modifiers
- Directly manipulating sub-object geometry

These topics will help you start creating and modifying objects:

Using the Create Panel on page 379

Creating an Object on page 383

Using the Modify Panel on page 1086

Using the Modifier Stack on page 1090

Editing the Stack on page 1093

Modifying at the Sub-Object Level on page 1098

Using the Stack at the Sub-Object Level on page 1102

Modifying Multiple Objects on page 1104

How Instanced Modifiers Work on page 1110

Transforms, Modifiers, and Object Data Flow on page 1074

Varying the Parameters

Unlike physical objects, with a fixed shape and size, you can change the creation parameters of objects and shapes to dramatically alter topology. Here are some examples of changes you can make:

- Turn a cone into a four-sided pyramid by reducing the number of sides and turning the Smooth option off.
- Slice any circular object as if it were a pie.
- Animate almost all creation parameters, and interactively change their settings during animation playback.

- Render splines directly at any assigned width.
- Break, detach, and divide wall segments.
- Change the number of risers without affecting the overall rise of the stairs.

Collapsing Primitives to Base Geometry

You can collapse a geometric primitive or shape to one of a variety of base geometric types once you no longer need access to its creation parameters. For example, you can convert any standard primitive to an editable mesh on page 2075, editable poly on page 2123, editable patch on page 2019, or NURBS on page 2261 object, and you can convert a spline shape to an editable mesh, editable spline on page 659, or NURBS object. The easiest way to collapse an object is to select it, right-click it, and choose a "Convert to" option from the quad menu > Transform quadrant. This lets you use explicit editing methods with the object, such as transforming vertices. You can also use the Modify panel to collapse a primitive.

Mapping Coordinates

Most Geometry objects have an option for generating mapping coordinates. Objects need these mapping coordinates if you plan to apply a mapped material to them. Mapped materials include a wide range of rendered effects, from 2D bitmaps to reflections and refractions. See Mapping Coordinates on page 5279 and Using Maps to Enhance a Material on page 5274. If mapping coordinates have already been applied to an object, the check box for this feature is turned on.

Using the Create Panel

The Create panel provides the controls for creating objects and adjusting their parameters.

To access the Create panel:

1.  Click the Create tab in the command panels on page 7630.

   By default, this panel is open when you start the program. If the command panel isn't visible, choose it from the Customize Display right-click menu on page 7687.

2 Click an object type to display its Parameters rollout.

The Creation Process

The actual creation of objects is accomplished with a single click of the mouse, a drag, or some combination, depending on the object type. This is the general sequence:

- Choose an object type.
- Click or drag in a viewport to create an object of approximate size and location.
- Adjust the object's parameters and position, either immediately or later.

See Creating an Object on page 383.

Create Panel Interface

Controls in the Create panel vary depending on the kind of object you are creating. However, certain controls are always present, and others are shared by nearly all object types.

Category Buttons at the top of the panel access the seven main categories of objects. Geometry is the default category.

Subcategory A list lets you select subcategories. For example, subcategories under Geometry include Standard Primitives, Extended Primitives, Compound Objects, Particle Systems, Patch Grids, NURBS Surfaces, and Dynamics Objects.

Object Type A rollout contains labeled buttons for creating objects in a particular subcategory, plus the AutoGrid on page 2597 check box.

Name and Color The Name shows the automatically assigned name of the object. You can edit this name or replace it with another. (Different objects can have the same name, though this is not recommended.) Clicking the square color swatch brings up an Object Color dialog on page 387 to change the color of the object as it appears in viewports (the wireframe color).

Creation Method This rollout provides a choice of how you use the mouse to create an object. For example, you can use either the center (radius) or edge (diameter) to define the size of a Circle shape.

A default creation method is always selected when you access the tool. If you want to use an alternate method, choose the option before you create the object. The creation method has no effect on a finished object; the options are for your convenience during creation.

Keyboard Entry This rollout lets you enter creation parameters from the keyboard for geometric primitive and shape objects.

Parameters This rollout shows creation parameters: the defining values for an object. Some parameters can be preset, while others are only for adjustment after an object has been created.

Other rollouts Additional rollouts can appear on the Create panel, depending on what kind of object you create.

Identifying the Basic Building Blocks

On the Create panel, the categories for Geometry and Shapes supply the "building blocks" to combine and modify into more sophisticated objects. These parametric on page 8080 objects are ready to use. By adjusting values and turning some buttons on or off, you can create dozens of "new" building blocks from the ones listed here.

You can choose these types from the sub-categories list on the Create panel.

Geometry Types

Standard Primitives Relatively simple 3D objects such as Box, Sphere, and Cylinder, as well as Torus, Plane, Cone, GeoSphere, Tube, Teapot, and Pyramid.

Extended Primitives More complex 3D objects such as Capsule, OilTank, Spindle, Hedra, Torus Knot, and Prism.

Compound Objects Compound objects include Scatter, Connect, ShapeMerge, Booleans, Morph, BlobMesh, Terrain, and Loft. Booleans combine the geometry of two objects using union, intersection, and difference operations. Morphs are animated objects that change one geometric shape into other shapes over time. ShapeMerge lets you embed a spline shape into a geometric mesh. Loft on page 786 uses shapes as cross sections along a path to produce a 3D object.

Particle Systems Animated objects that simulate spray, snow, blizzard, and similar collections of small objects.

Patch Grids Simple 2D surfaces ready for modeling or repairing existing meshes.

NURBS Surfaces Analytically generated surfaces especially suited for modeling surfaces with complicated curves.

AEC Extended Elements useful for AEC design, including Terrain, Foliage (plants and trees), Railing, for creating custom railings, and Wall, for the production of Wall objects.

Stairs Four types of stairs: Spiral, L-Type, Straight, and U-Type.

Doors Parametric door styles include Pivot, BiFold, and Sliding.

Windows Parametric window styles include Awning, Fixed, Projected, Casement, Pivoted, and Sliding.

NOTE Default materials are automatically applied to Foliage, as well as to the following object types: Railing, Stairs, Doors, and Windows.

Dynamics Objects Objects designed for use in dynamics simulations.

Shape Types

Splines Common 2D shapes such as a Line, Rectangle, Circle, Ellipse, Arc, Donut, NGon, and Star. Text shapes support TrueType fonts. Section creates a spline from the cross-section of an object. Helix is a 3D shape.

NURBS Curves A Point Curve and CV Curve provide the starting points for complex surfaces. See Introduction to NURBS Modeling on page 2237.

Extended Splines More complex 2D shapes including Walled Rectangle, Channel Spline, Angle Spline, Tee Spline, and Wide Flange Spline. Extended splines can be used in architectural and similar applications.

Varying the Parameters

Unlike physical building blocks, with fixed shape and size, you can change the parameters of objects and shapes to dramatically alter topology. Here are some examples of changes you can make:

- Turn a cone into a four-sided pyramid by reducing the number of sides and turning the Smooth option off.
- Slice any circular object as if it were a pie.
- Animate almost all creation parameters, and interactively change their settings during animation playback.
- Render splines directly at any assigned width.
- Break, detach, and divide wall segments.
- Change the number of risers without affecting the overall rise of the stairs.

Collapsing Primitives to Base Geometry

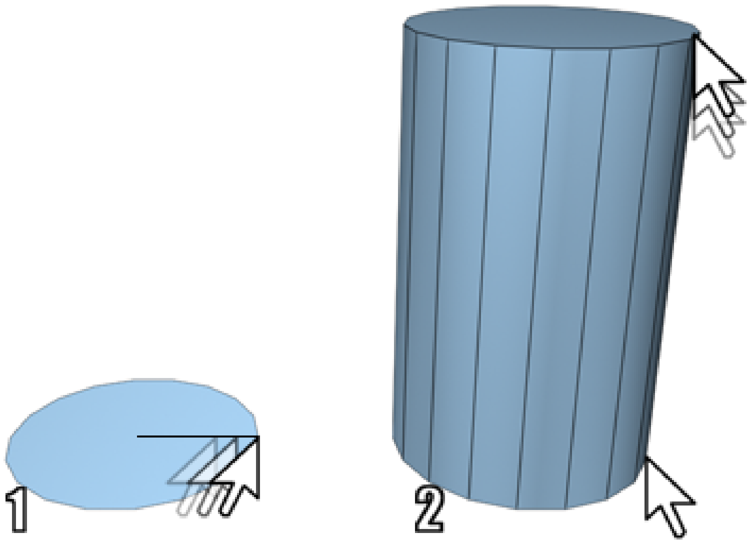
You can collapse a building-block object to one of a variety of base geometric types once you no longer need access to its creation parameters. For example, you can convert any standard primitive to an editable mesh on page 2075, editable poly on page 2123, editable patch on page 2019, or NURBS on page 2261 object, and you can convert a spline shape to an editable mesh, editable spline on page 659, or NURBS object. The easiest way to collapse an object is to select it, right-click it, and choose a "Convert to" option from the quad menu > Transform quadrant. This lets you use explicit editing methods with the object, such as transforming vertices. You can also use the Modify panel to collapse a primitive.

Mapping Coordinates

Most Geometry objects have an option for generating mapping coordinates. Objects need these mapping coordinates if you plan to apply a mapped material to them. Mapped materials include a wide range of rendered effects, from 2D bitmaps to reflections and refractions. See Mapping Coordinates on page 5279 and Using Maps to Enhance a Material on page 5274. If mapping coordinates have already been applied to an object, the check box for this feature is turned on.

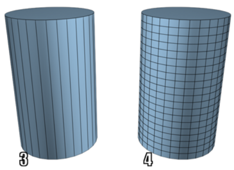
Creating an Object

With some variations, the steps shown in the following images apply to creating any type of object on the Create panel. For specific examples, see the Procedures section in any object's topic.



1. Radius defined

2. Height defined



3. Sides increased

4. Height Segments increased

To choose an object category:

1.  Click the Create tab to view the Create panel.
2. Click one of the buttons at the top of the Create panel. For example, Geometry.
3. Choose the subcategory Standard Primitives from the list.

A number of buttons appear on the Object Type rollout.

To choose an object type:

- Click the button for the type of object you want to create.
  The button highlights, showing that it is active. Four rollouts appear: Name and Color, Creation Method, Keyboard Entry, and Parameters.

To choose a creation method (optional):

You can accept the default method and skip this step.

- Choose a method in the Creation Method rollout.

To preset the creation parameters (optional):

You can adjust *all* creation parameters after you create an object. Skip this step if you prefer.

- In the Parameters rollout, you can set parameters before you create an object. However, the values of parameters you set by dragging the mouse (for example, the Radius and Height of a cylinder) have no effect until after you create the object.

To create the object:

1. Put the cursor at a point in any viewport where you want to place the object, and hold the mouse button down (do not release the button).
2. Drag the mouse to define the first parameter of the object; for example, the circular base of a cylinder.
3. Release the mouse button. The first parameter is set with this release.

   In some cases, such as Sphere, Teapot, and Plane, this completes the object. You can skip the remaining steps.
4. Move up or down without touching the mouse button. This sets the next parameter; for example, the height of a cylinder.

   If you want to cancel: Until you complete the next step, you can cancel the creation process with a right-click.
5. Click when the second parameter has the value you want, and so on.

   The number of times you press or release the mouse button depends on how many spatial dimensions are required to define the object. (For some kinds of objects, such as Line and Bones, the number is open-ended.)

When the object is complete, it is in a selected state and ready for adjustments.

To name the object (optional):

- Highlight the default object name in the Name and Color rollout, and then enter a name. This option is available only when a single object is selected.
  Naming objects is a good practice for organizing your scenes. To name a set of selected objects, see Named Selection Sets on page 206.

To change the object's display color (optional):

- The color swatch next to the object name field displays the selected object's color and lets you select a new one. The color is the one used to display the object in viewports. Click the color swatch to display the Object Color dialog on page 387.
  You can also change object colors with Layers on page 7441.

To adjust the object's parameters:

- You can change the creation parameters immediately after you complete an object, while it's still selected. Or, you can select the object later and adjust its creation parameters on the Modify panel.

While making adjustments, you can use viewport navigation controls like Zoom, Pan, and Orbit to change your view of the selected object. You can also adjust the time slider.

To end the creation process:

While the object type button remains active, you can continue creating objects of the same type until you do one of the following:

1. Select an object other than the one you created most recently.
2. Transform an object.
3. Change to another command panel.
4. Use commands other than viewport navigation or the time slider.

After you end the creation process, changing parameters on the Create panel will have no effect on the object; you must go to the Modify panel to adjust the object's parameters. See Using the Modify Panel on page 1086.

Assigning Colors to Objects

3ds Max is a truecolor on page 8159 program. When you pick a color in the program, you are specifying 24 bits of color data, which provide a range of over 16 million colors.

Object wireframe colors are used primarily as an organizational tool. Object naming strategies, named selection sets, and object wireframe color strategies provide a rich set of tools for organizing even the most complex scenes.

You can use two dialogs to specify colors:

- The Object Color dialog on page 387 contains two preset palettes of colors that you use to set an object's wireframe color. This is also the surface color you see in a rendered viewport. The two color palettes are Default palette and AutoCAD ACI palette.
- The Color Selector on page 391 is a generic dialog that you use to define any color in the 24-bit color range. For the purpose of defining colors to assign to objects, it is available only through the Default palette.

The Layers functionality lets you organize your scene and can also be used for assigning object colors. For more information, see Layer Manager on page 7441.

Object Color Dialog

Click the color swatch by the object's name in any command panel.

The Object Color dialog contains two preset palettes of colors that you use to set an object's wireframe color. This is also the surface color you see in a shaded viewport.

Using Random Color Assignment

By default, 3ds Max assigns colors randomly as objects are created. The colors are chosen from the current palette in the Object Color dialog. If you turn on Customize > Preferences > General panel on page 7744 > Default to By Layer for New Nodes, new objects are assigned the color set by the layer.

For individual objects, you can click the By Layer/By Object button on the Object Color dialog to change the method used to set the object color.

Defining Custom Colors

When using the 3ds Max palette, the Object Color dialog contains a palette of 16 custom color swatches. You can define any color for each of the 16 color swatches by selecting a swatch from the Custom Colors group, then clicking Add Custom Colors.

Switching Between Palettes

You can alternate between two versions of the Object Color dialog at any time by clicking the appropriate Basic Colors toggle:

- **3ds Max palette:** Contains a fixed palette of 64 colors, plus a custom palette of 16 user-defined custom colors.
  Use this version when you want to work with a smaller palette of colors or when you want to define custom object wireframe colors.
- **AutoCAD-compatible version:** Contains a fixed palette of 256 colors matching the colors in the AutoCAD Color Index (ACI).
  Use this version when you want to assign object colors that match the AutoCAD Color Index. Using ACI colors is useful if you plan to export objects to AutoCAD and want to organize them by object color, or when you want a wide selection of colors to choose from.

Procedures

To set object color:

This is the general procedure for selecting object color.

1. Select one or more objects.
2. On any command panel, click the color swatch to the right of the Object Name field to display the Object Color dialog.
3. On the Object Color dialog, click the By Layer / By Object toggle to set it to By Object.
4. Click a color swatch from the palette, and then click OK to apply the color to the selection.

To create objects of the same color:

- Choose the color you want to use and turn off Assign Random Colors. Newly created objects appear in this color until you change the setting.

To define a custom color:

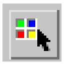
1. With the 3ds Max palette option active, click one of the 16 custom color swatches.
2. Click Add Custom Colors to display the Color Selector on page 391.
3. Define a custom color and click Add Color.

   The custom color is stored in the selected color swatch of the Object Color dialog and is set as the current color.

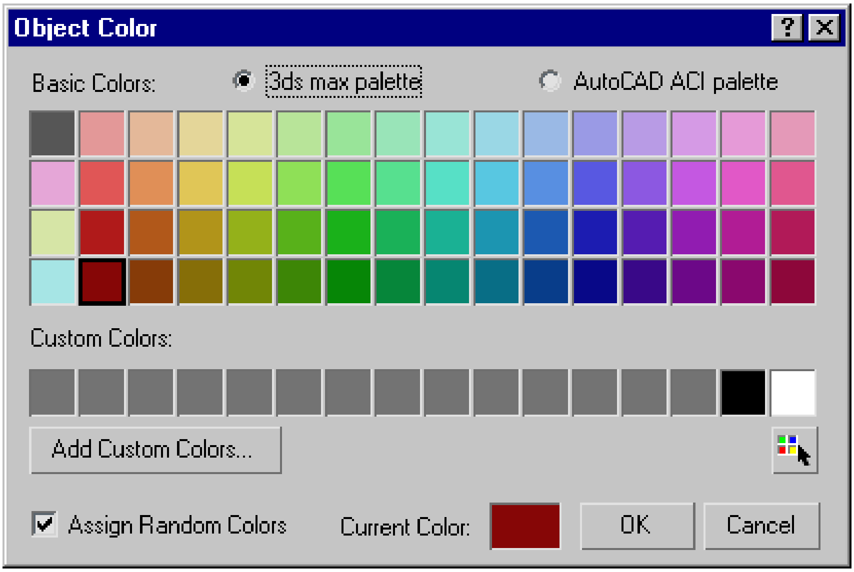
To copy a custom color from an object in your scene to one of your custom color swatches:

- Drag the Active Color swatch up to one of the custom color swatches. The Active Color swatch is in the Object Color dialog, to the left of the OK button.

To select objects by color:

-  Click Select By Color. This displays the Select Objects dialog on page 228. All objects that have the same color as the current object are highlighted in the list. Click Select.

Interface



Palette Choose one of these:

- **3ds Max palette** When chosen, the dialog displays Basic Colors and Custom Colors groups, and you have the option to add custom colors.
- **AutoCAD ACI palette** When chosen, the AutoCAD ACI palette is shown. When you click a color, its ACI# is displayed at the bottom of the dialog.

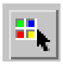
Basic Colors A set of 64 default colors, available only when 3ds Max Palette is active.

Custom Colors Displays 16 custom colors when 3ds Max Palette is active. To choose a custom color, click its swatch. To define or change a custom color, click its swatch and then click Add Custom Colors.

Add Custom Colors Available only when 3ds Max Palette is active. Clicking this option displays the Color Selector on page 391, which allows you to modify the currently selected custom color. If you click Add Custom Colors with a basic color chosen, the dialog switches to the first custom color before opening the Color Selector.

By Layer/By Object Sets the object's color by layer or by object. If color is set by object, choosing a new color on the Object Color dialog changes the object's wireframe color in viewports.

ACI# Displays the ACI number for the selected color. Available only when AutoCAD ACI palette is active.

 **Select by Color** Opens the Select Objects dialog on page 228 listing all objects that use the current color as their wireframe color.

NOTE This button is available only if at least one object in the scene has the Current Color as its wireframe color.

Assign Random Colors When on, 3ds Max will assign a random color to each object created. When off, 3ds Max will assign the same color to every object created until the color swatch is changed. This setting affects wireframe colors only when By Object is turned on as the color method.

Active/Current Color Displays the active color (if no object is selected) or current color. When you click the color swatch, the Color Selector dialog on page 391 opens, where you can mix a custom color.

Color Selector Dialog

Any command panel > Name and Color fields > Click color swatch. > Object Color dialog > Add Custom Colors button or Current Color swatch.

Material Editor > Click any color swatch.

Select or add a light object. > Modify panel > Intensity/Color/(Distribution/Attenuation) rollout > Click color or Filter Color swatch.

Rendering menu > Environment > Environment and Effects dialog > Click color swatch for Background, Tint, and Ambient components of Global Lighting, and various components of atmospheric effects such as Fire, Fog, and so on..

The Color Selector dialog lets you specify a custom color parameter in 3ds Max. You can work simultaneously with three different color models to help you zero in on the exact color you want.

You can use the Color Selector to specify many color parameters, such as light colors, material colors, background colors, and custom object colors. (Another way to choose an object's viewport color is to use the predefined colors in the Object Color dialog on page 387.)

In most contexts, the Color Selector is modeless on page 8045; that is, it remains on the screen until you dismiss it, and you can use other controls or work in a viewport while the dialog is still visible. In other contexts, the Color Selector is modal, and you must close the dialog before proceeding.

The dialog is divided into three different color selection models. You can use the controls for any model to define a color. The three color models are:

- Hue/Blackness/Whiteness (HBW)
  The most prominently displayed and intuitive color model is the HBW model. This model represents a natural, pigment-based way of mixing color by starting with a pure color (hue) and then making it darker by adding black, or lighter by adding white.

  The main feature of the HBW model is a large square box displaying the color spectrum. Across the top of this box you have the spectrum of pure colors, or hue. Down the side of the box you see increasing levels of blackness, making the color dark as you approach the bottom.

  To the right of the color spectrum box is the Whiteness box, which controls the amount of white in the color. Use higher positions to decrease the whiteness, or lower positions to increase the whiteness.

- Red/Blue/Green (RGB)
  The RGB model adjusts the mix of Red, Green, and Blue to define a color. This model represents the way colored light can be mixed. This is additive color mixing, as opposed to the subtractive color mixing for paint and other pigments. You can adjust values using the color sliders, the numeric fields to their right (via the keyboard), or the spinners to the right of the numeric fields.

- Hue/Saturation/Value (HSV)
  The HSV color model adjusts Hue, Saturation, and Value. Hue sets the color; Saturation (labeled "Sat") sets the color's purity; and Value sets the color's brightness, or intensity. You can adjust values using the color sliders, the numeric fields to their right (via the keyboard), or the spinners to the right of the numeric fields.

As you adjust the controls of one color model, the controls of the other two models change to match. The color defined by the color model is displayed in the right half of the Color Output box. The original color, before you began making changes, is displayed in the left half.

Procedures

To display the Color Selector:

1. Click the color swatch of a color parameter such as the color of a light or of a material component.

   NOTE The object color displayed next to an object's name on command panels uses the Object Color dialog on page 387. On the Object Color dialog, clicking the Active (or Current) Color swatch or the Add Custom Colors button displays a Color Selector.

2. Make a color selection and click OK or Cancel, or the Close button (X).

   If using the Add Color version of the Color Selector, be sure to click Add Color first.

3. To revert to the original color, click Reset.

To choose the hue of a color, do one of the following:

1. Click anywhere in the Hue rainbow (the large, multicolored square).
2. Drag the Hue slider at the top of the rainbow.
3. Drag the Red, Green, and Blue sliders.
4. Drag the Hue slider.
5. Use the Red, Green, Blue, or Hue spinners.

To make a color lighter, do one of the following:

1. Drag the vertical Whiteness slider (at the right of the Hue rainbow) downward.
2. Drag the vertical Blackness slider (at the left of the Hue rainbow) upward.
3. Drag the Saturation (Sat.) slider to the left.
4. Use the Saturation spinner to decrease saturation.
5. Drag the Value slider to the right.
6. Use the Value spinner to increase the value.

To make a color darker, do one of the following:

1 Drag the vertical Whiteness slider (at the right of the Hue rainbow) upward.

2 Drag the vertical Blackness slider (at the left of the Hue rainbow) downward.

3 Drag the Saturation (Sat.) slider to the right.

4 Use the Saturation spinner to increase saturation.

5 Drag the Value slider to the left.

6 Use the Value spinner to decrease the value.

To return to the original color:

- Click Reset.
  The new color is replaced by the original color, and all parameter values are reset.

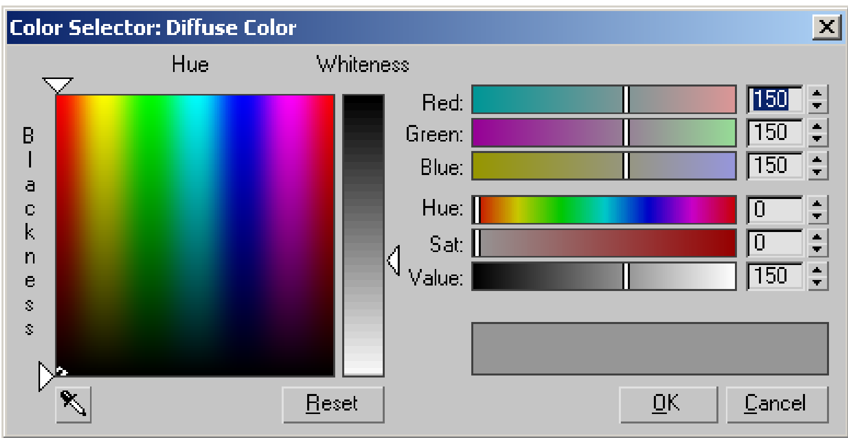
To dismiss the Color Selector, do one of the following:

1 Click Close.

2 Click OK or Cancel.

3 Click the dialog's Close (X) button.

Interface



Hue Define a pure color by dragging the hue pointer across the top of the box.

Blackness Drag the blackness pointer down the side to darken the pure color by adding black. You can also click or drag inside the box to change hue and blackness at the same time.

Whiteness The vertical bar to the right controls the amount of whiteness. The color set by the hue and blackness pointers is displayed at the top of the bar and pure white at the bottom. Drag the whiteness pointer down to lighten the color by adding white.

Red, Green, and Blue When a red, green, or blue slider is all the way to the left, its numeric field contains 0; none of the color controlled by that slider is used. If the slider is all the way to the right, the field reads 255; the maximum amount of that color is being used.

The spinners to the right of each slider are another way of setting the red, blue, or green component.

The colors in the sliders change to show an approximation of what the color result will be if you move the slider to that location, without adjusting any other color parameter.

Hue Sets the pure color. Locating the slider all the way to the left gives you pure red. As you drag the slider to the right you move through the spectrum of Red, Yellow, Green, Cyan, Blue, Magenta, and back to Red again. Hue is more accurately represented as a color wheel rather than a linear slider. That is why the Hue slider is red at both ends. Think of the hue range from 0 to 255 as being points on a circle where the numbers 0 and 255 are right next to each other.

Saturation ("Sat") Sets the purity or strength of the color. A weak color, with a saturation near 0, is dull and gray. A strong color, with a saturation near 255 is very bright and pure.

Value Sets the lightness or darkness of a color. Low values darken the color toward black. High values lighten the color toward white. A value in the middle, at a setting of 127, gives you the color defined only by hue and saturation.

Color Output This pair of color swatches, below the Value slider, lets you compare the new color, shown on the right, to the original color, shown on the left.

 **Sample Screen Color** Lets you pick a new color from anywhere on the screen. After clicking this button, the mouse cursor changes to the eyedropper icon shown on the button. While this cursor appears, use any of these methods:

- Click anywhere on the screen to replace the current color with the color of the pixel under the lower-right corner of the cursor.
- Drag to continually update the current color with the color of the pixel under the lower-right corner of the cursor. This makes it easier to make sure you get the right color if the desired color area is small (say, a one-pixel-thick line).
- At any time, press and hold Shift to *average* the current color with colors the cursor moves over.
  Instead of replacing the current color with the new sampled color, smoothed sampling gradually mixes the sampled color with the current color, giving a smoothed color transition during sampling. This is useful for sampling noisy areas, where the variations in colors are accumulated to provide a representative general color.

  Unlike the other methods, releasing the left mouse button only does not exit the sampler mode; you can move the mouse elsewhere (without sampling) and then start dragging again to continue smoothed sampling in other areas. Releasing Shift only returns to regular sampling. Releasing both Shift and the left mouse button exits the sampler mode, returning the mouse cursor and behavior to normal.

Sampling can occur under any conditions anywhere within any windows that belong to the current instance of 3ds Max. To sample anywhere outside of 3ds Max (for example, the desktop), drag the mouse from within one of these 3ds Max windows.

The color sampler tool compensates for any gamma applied to the color selector using the Customize > Preferences > Gamma And LUT on page 7758 > Affect Color Selectors option. This means that the color-corrected, displayed sampled visual color in the color selector always matches the on-screen visual color of the sampled location. If the gamma of the color selector does not match the gamma of the sampled location, the true color values (RGB/HSV) of the sampled color will differ from the true color values of the sampled location. This behavior applies to both regular gamma and Autodesk LUT gamma correction modes.

Reset Click to restore color settings to the original color.

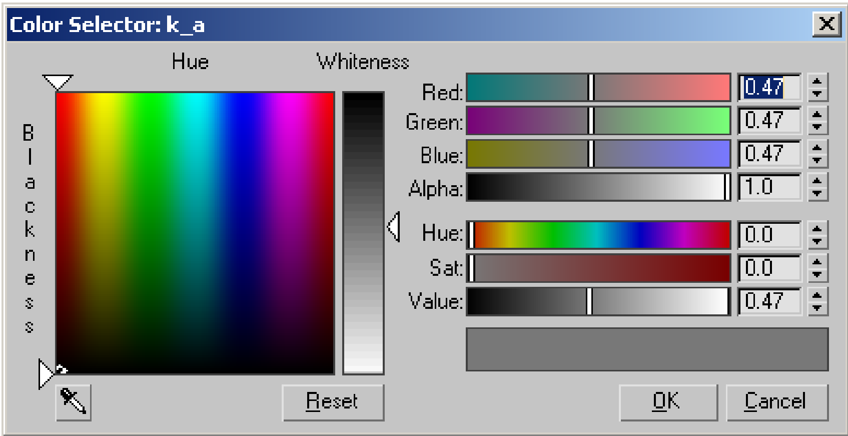
OK Accepts any changes and closes the dialog.

NOTE If you opened the dialog by clicking Add Custom Colors on the Object Color dialog, the button label reads "Add Color."

Cancel Restores the original color and closes the dialog.

Color Selector for mental ray Materials and Shaders

When you click a color swatch in the interface for a mental ray material on page 5543 or mental ray shader on page 5974, or a DirectX material on page 5758, you see a variant of the Color Selector.



This dialog differs from the standard Color Selector in two ways:

- The RGB and HSV values appear as normalized values between 0.0 and 1.0, rather than as 8-bit integers (0–255).
- An additional Alpha slider and spinner let you explicitly set the alpha value for this color. This value is also normalized, where 0.0 represents fully transparent, and 1.0 represents fully opaque.

This version of the Color Selector also appears when you use the DirectX 9 Shader material on page 5758 and the mental ray renderer's Sampling Quality rollout on page 6272.

Color Clipboard Utility

Tools menu > Color Clipboard

Utilities panel > Utilities rollout > More button > Utilities dialog > Color Clipboard button

The Color Clipboard utility stores color swatches for copying from one map or material to another.

For example, if in the Material Editor, you want to copy a color from a swatch in one level of a material to a swatch in another level (or from another material), there would be no way to do it with drag and drop. This is because you can't have two materials/maps visible at the same time. However, you can drag the color from one material to the color clipboard, switch to the other material, and then drag the color from the clipboard to the swatch in the new material.

You can save and load color clipboard files. The saved file, which is given a *.ccb* (color clip board) extension, is an ASCII file that contains a palette description. The first 12 lines of the file consist of three RGB numbers, so you can easily edit or create your own clipboard files. This file format is also used by the VertexPaint modifier on page 1959.

Procedures

To copy a color from a swatch to the color clipboard:

1 On the Utilities panel, click Color Clipboard.

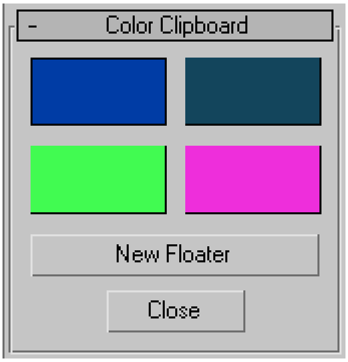
2 Open the Material Editor.

3 Select a color from any swatch in a material.

4 Drag the color to a swatch in the color clipboard.

5 A dialog appears asking if you want to copy or swap the material. Choose copy to replace the swatch in the color clipboard with the swatch from the material you selected. Choose swap to swap colors on the Color Clipboard swatch and material swatch.

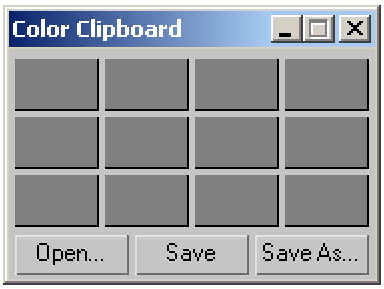
Interface



Color swatches Click a color swatch to edit its value with the Color Selector.

NOTE The Color Selector invoked by this utility uses decimal numbers in the range 0.0 to 1.0, instead of integers in the range 0 to 255 as with other color-selection dialogs in 3ds Max.

New Floater Displays a floating clipboard with 12 slots, plus buttons for opening and saving color clipboard files. You can open up as many of these floaters as you want and you can minimize them. If you exit the Utilities panel or select the Close button to exit the Color Clipboard utility, any visible floaters remain open. When you close a floater, any changed values are lost.



Close Exits the Clipboard utility.

Adjusting Normals and Smoothing

In general, you adjust normals and smoothing to prepare objects for rendering.

A normal on page 8059 is a unit vector that defines which way a face or vertex is pointing. The direction in which the normal points represents the front, or outer surface of the face or vertex, which is the side of the surface that is normally displayed and rendered.

You can manually flip or unify *face* normals to fix surface errors caused by modeling operations or by importing meshes from other programs.

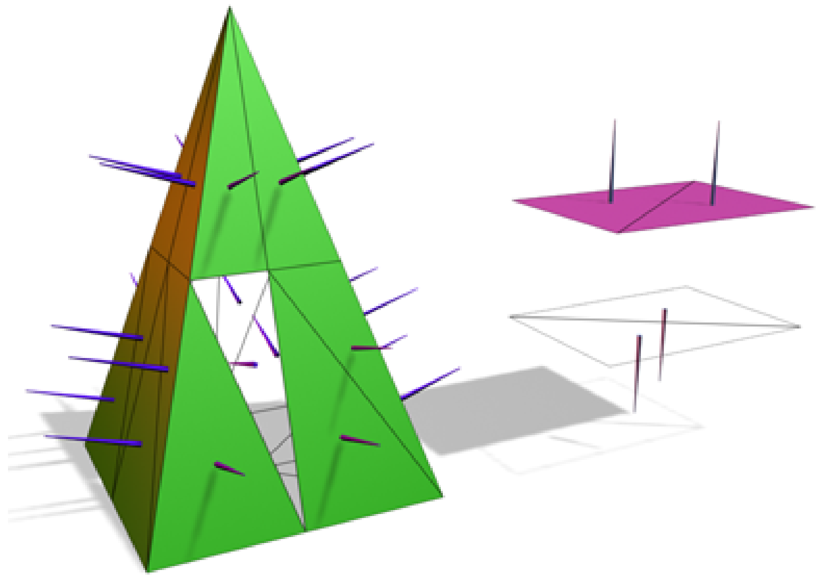
Smoothing groups define whether a surface is rendered with sharp edges or smooth surfaces. Smoothing groups are numbers assigned to the faces of an object. Each face can carry any number of smoothing groups up to the maximum of 32. If two faces share an edge and share the same smoothing group, they render as a smooth surface. If they don't share the same smoothing group, the edge between them renders as a corner. You can change and animate smoothing group assignments manually. Changing smoothing groups does not alter geometry in any way; it simply changes the way faces and edges are shaded.

See also:

- Viewing and Changing Normals on page 400
- Viewing and Changing Smoothing on page 403

Viewing and Changing Normals

When you create an object, normals on page 8059 are generated automatically. Usually objects render correctly using these default normals. Sometimes, however, you need to adjust the normals.



Left: The normals shown as spikes indicate the orientation of faces on the pyramid.

Right: Flipping normals can make faces invisible (or visible) in shaded viewports and renderings.

Undesired normals can appear in these objects:

- Meshes imported from other applications.
- Geometry generated by complex operations such as Boolean objects, lathe objects, or lofts.

Normals are used to define which side of a face or vertex is considered the "out" side. The out side of a face or vertex is the side that gets rendered unless you are using two-sided materials, or turn on the Force 2-Sided option in the Render Setup dialog > Common panel > Common Parameters rollout on page 6121.

Do one of the following to view or change *face normals*:

- Apply a Normal modifier on page 1581. If a Face sub-object selection is active, Normal applies to the selected faces. If no faces are selected, Normal applies to the entire object.

- Apply an Edit Mesh modifier on page 1353, enable Face, Polygon or Element sub-object mode, and then use the features on the Surface Properties rollout to change the directions in which normals point.
- Convert the object to an editable mesh on page 2075, enable Face, Polygon or Element sub-object mode, and use the features on the Surface Properties rollout

Viewing Normals

The easiest way to view normals is to look at an object in a shaded viewport. In this case, you are not viewing the normal arrows themselves, but rather their effects on the shaded surface. If the object looks as if it is inside-out, or has holes, then some of the normals might be pointing in the wrong direction.

You can display the normal vectors for selected faces or vertices by enabling Show Normals on the Selection rollout of an editable mesh object or the Edit Mesh modifier.

Unifying Normals

Use Unify Normals to make normals point in a consistent direction. If an object has normals that are inconsistent (some point outward and others inward) the object will appear to have holes in its surface.

Unify Normals is found on the Surface Properties rollout and on the Normal modifier.

If you are animating the creation of a complex object such as a nested Boolean or a loft, and you think the operation might result in inconsistent faces, apply a Normal modifier on page 1581 to the result, and turn on Unify Normals.

Flipping Normals

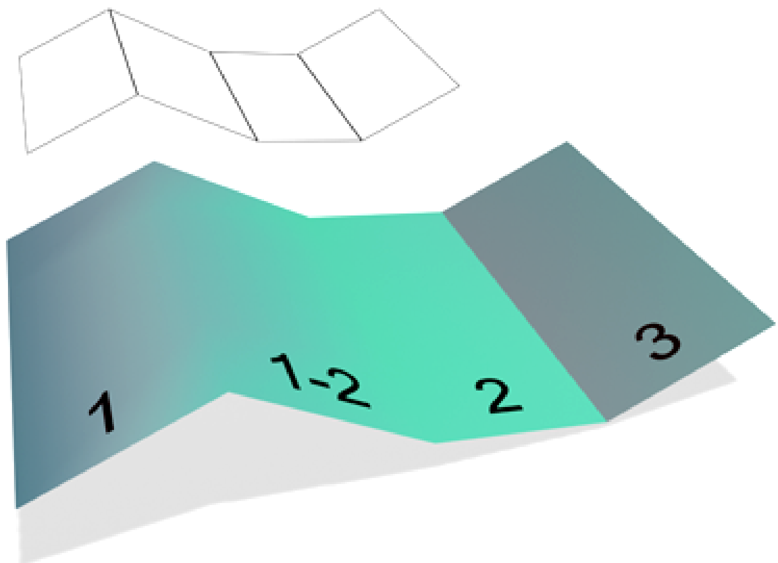
Use Flip Normals to reverse the direction of all selected faces. Flipping the normals of an object turns it inside-out.

Flip Normals is found on the Surface Properties rollout and on the Normal modifier.

The Lathe modifier on page 1501 sometimes creates an object with normals pointing inward. Use the Flip Normals check box on the Lathe modifier's Parameters rollout to adjust the normals. You can also use the Normal modifier with both Unify and Flip turned on to fix inside-out lathed objects.

Viewing and Changing Smoothing

Smoothing blends the shading at the edges between faces to produce the appearance of a smooth, curved surface. You can control how smoothing is applied to a surface so your objects can have both smooth surfaces and sharp, faceted edges where appropriate.



The face labeled "1-2" shares smoothing groups with adjacent faces, so the edges between them are smoothed over in renderings.

The face labeled "3" does not share a smoothing group, so its edge is visible in renderings.

Smoothing does not affect geometry. It affects only the way geometry is colored when rendered.

Smoothing is controlled by smoothing groups, which are numbered groups ranging from 1 to 32. Each face is assigned to one or more smoothing groups. When a scene is rendered, the renderer checks each adjacent pair of faces to see if they share a smoothing group, and renders the object as follows:

- If faces have no smoothing groups in common, the faces are rendered with a sharp edge between them.

- If faces have at least one smoothing group in common, the edge between the faces is "smoothed", meaning it is shaded in such a way that the area where the faces meet appears smooth.

Because each face has three edges, only three smoothing groups can be in effect for any face. Extra smoothing groups assigned to a face are ignored.

Do one of the following to view or change smoothing group assignments:

- Turn on the Smooth check box on the Parameters rollout of a parametric object to set default smoothing for the object.
- Turn on the Auto Smooth check box on the Rendering rollout of a spline shape to turn on smoothing.
- Apply a Smooth modifier on page 1736. If a Face sub-object selection is active, Smooth applies to the selected faces. If no faces are selected, Smooth applies to the entire object.
- Apply an Edit Mesh modifier on page 1353, enable Face (or Polygon or Element) sub-object mode, then use the features on the Surface Properties rollout.
- Convert the object to an editable mesh on page 2075, enable Face (or Polygon or Element) sub-object mode, then use the features on the Surface Properties rollout.

Viewing Smoothing Groups

The easiest way to view smoothing is to look at an object in a shaded viewport. In this case, you are not viewing the smoothing groups themselves but rather their effects on the shaded surface.

You can see the smoothing group numbers for selected faces of an editable mesh object or the Edit Mesh modifier by looking at the Smoothing Group buttons on the Surface Properties rollout, or of an editable poly object on the Polygon Properties rollout.

Smoothing Group buttons appear as follows:

- Group numbers not used by any face in the selection, appear normal.
- Group numbers used by all faces in the selection, appear selected.
- Group numbers used by some, but not all, faces in the selection, appear blank.

Automatically Smoothing an Object

Click Auto Smooth to assign smoothing automatically. You set a Threshold angle to determine whether to smooth adjacent faces.

- If the angle between face normals is less than or equal to the threshold, the faces are assigned to a common smoothing group.
- If the angle between face normals is greater than the threshold, the faces are assigned to separate groups.

Auto Smooth is found on the Surface Properties rollout and on the Smooth modifier.

Manually Applying Smoothing Groups

You manually assign smoothing groups to a selection of faces by clicking Smoothing Group buttons on the Surface Properties rollout or the Smooth modifier. The smoothing group of each button you click is assigned to the selection.

Selecting Faces by Smoothing Group

You can also select faces according to the assigned smoothing groups. Click Select By SG on the Surface Properties rollout (editable mesh) or Polygon Properties rollout (editable poly) and then click the smoothing group of the faces to select.

This is a convenient way to examine smoothing groups on an object someone else created.

Geometric Primitives

Geometric primitives are basic shapes that 3ds Max provides as parametric objects on page 8080. Primitives are divided into two categories:

Standard Primitives on page 408

Extended Primitives on page 443

See also:

- Basics of Creating and Modifying Objects on page 377

■ Creating an Object on page 383

■ Creating Primitives from the Keyboard on page 406

Creating Primitives from the Keyboard

Create panel > Geometry > Standard or Extended Primitives > Keyboard Entry rollout

You can create most geometric primitives from your keyboard using the Keyboard Entry rollout. In a single operation, you define both the initial size of an object and its three-dimensional position. The software automatically assigns the object's name and color. See Object Name and Wireframe Color on page 7631.

This method is generally the same for all primitives; differences occur in the type and number of parameters. The Hedra primitive, a complex and highly visual family of objects, is unsuited to this method and has no keyboard entry.

Procedures

To open the Keyboard Entry rollout:

1. On the Create panel for Standard or Extended Primitives, click any of the primitive Object Type rollout buttons, except Hedra or RingWave.
2. Click the Keyboard Entry rollout to open it. This rollout is closed by default.

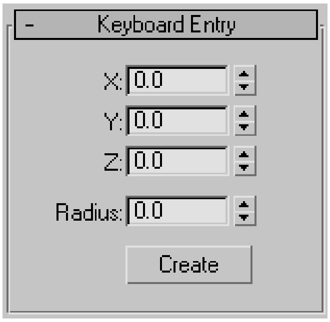
NOTE The buttons on the Creation Method rollout have no effect on keyboard entry.

To create a primitive from the keyboard:

1. On the Keyboard Entry rollout, select a numeric field with the mouse and then enter a number.
2. Press Tab to move to the next field. You do not have to press Enter after entering a value. Press Shift+Tab to reverse direction.
3. When you have all fields set, press Tab to move the focus to the Create button. Press Enter.
4. The object appears in the active viewport.

Once created, a new primitive is unaffected by the numeric fields in the Keyboard Entry rollout. You can adjust parameter values on the Parameters rollout, either immediately after creation or on the Modify panel.

Interface



The Keyboard Entry rollout contains a common set of position fields, labeled X, Y, and Z. The numbers you enter are offsets along the axes of the active construction plane; either the home grid or a grid object. Plus and minus values correspond to positive and negative directions for these axes. Defaults=0,0,0; the center of the active grid.

The location set by X,Y is equivalent to the first mouse-down position in the standard method of creating objects.

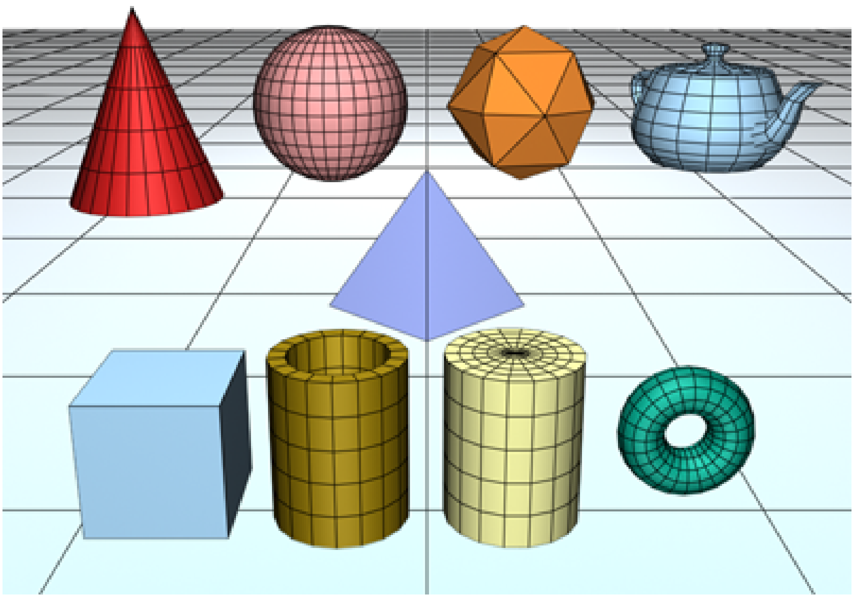
Each standard primitive has the following parameters on its Keyboard Entry rollout.

Primitive	Parameters	XYZ point
Box	Length, Width, Height	Center of base
Cone	Radius 1, Radius 2, Height	Center of base
Sphere	Radius	Center
GeoSphere	Radius	Center
Cylinder	Radius, Height	Center of base
Tube	Radius 1, Radius 2, Height	Center of base

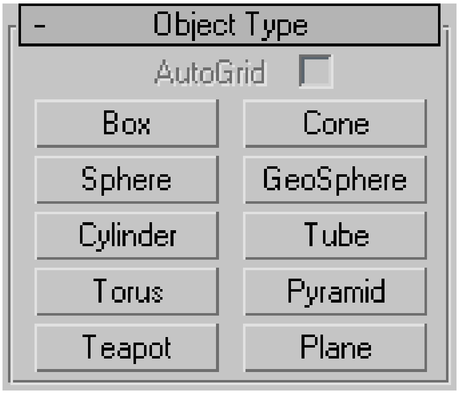
Primitive	Parameters	XYZ point
Torus	Radius 1, Radius 2	Center
Pyramid	Width, Depth, Height	Center of base
Teapot	Radius	Center of base
Plane	Length, Width	Center

Standard Primitives

Geometric primitives are familiar as objects in the real world such as beach balls, pipes, boxes, doughnuts, and ice cream cones. In 3ds Max, you can model many such objects using a single primitive. You can also combine primitives into more complex objects, and further refine them with modifiers.



A collection of standard primitive objects



3ds Max includes a set of 10 basic primitives. You can easily create the primitives with the mouse in the viewport, and most can be generated from the keyboard as well.

These primitives are listed in the Object Type rollout and on the Create menu:

Box on page 410

Cone on page 413

Sphere on page 416

GeoSphere on page 421

Cylinder on page 424

Tube on page 427

Torus on page 430

Pyramid on page 434

Teapot on page 437

Plane on page 440

Also available from the Object Type rollout is the AutoGrid option on page 2597.

You can convert standard primitive objects to editable mesh objects on page 2075, editable poly objects on page 2123, and NURBS surfaces. on page 2304 You can also convert primitives to patch objects; see the path annotation at Editable Patch on page 2019 (the information at the start of the topic that tells you how to create this type of object).

All primitives have name and color controls, and allow you to enter initial values from the keyboard. See these topics:

Object Name and Wireframe Color on page 7631

Creating Primitives from the Keyboard on page 406

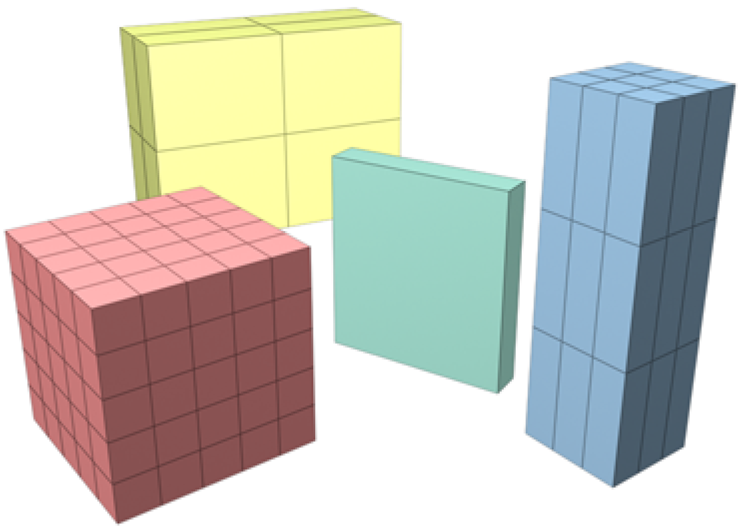
The remaining rollouts are covered in the topic for each primitive.

Box Primitive

Create panel > Geometry button > Standard Primitives > Object Type rollout > Box button

Create menu > Standard Primitives > Box

Box produces the simplest of the primitives. Cube is the only variation of Box. However, you can vary the scale and proportion to make many different kinds of rectangular objects, from large, flat panels and slabs to tall columns and small blocks.



Examples of boxes

Procedures

To create a box:

1. On the Object Type rollout, click Box.
2. In any viewport, drag to define a rectangular base, then release to set length and width.
3. Move the mouse up or down to define the height.
4. Click to set the finished height and create the box.

To create a box with a square base:

- Hold down Ctrl as you drag the base of the box. This keeps length and width the same. Holding the Ctrl key has no effect on height.

To create a cube:

1. On the Creation Method rollout, choose Cube.
2. In any viewport, drag to define the size of the cube.
3. As you drag, a cube emerges with the pivot point at the center of its base.
4. Release to set the dimensions of all sides.

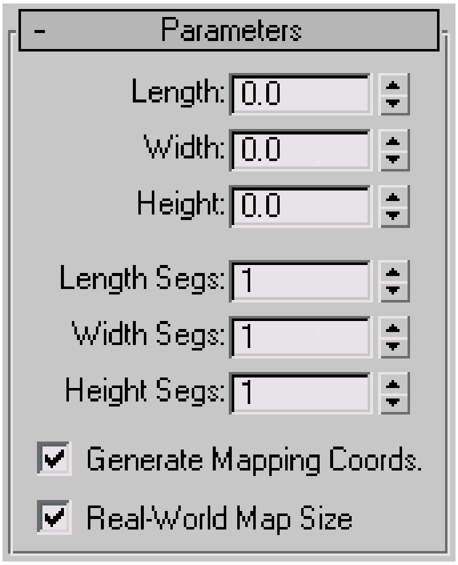
Interface

Creation Method rollout

Cube Forces length, width, and height to be equal. Creating a cube is a one-step operation. Starting at the center of the cube, drag in a viewport to set all three dimensions simultaneously. You can change a cube's individual dimensions in the Parameters rollout.

Box Creates a standard box primitive from one corner to the diagonally opposite corner, with different settings for length, width, and height.

Parameters rollout



The defaults produce a box with one segment on each side.

Length, **Width**, **Height** Sets the length, width, and height of the Box object. These fields also act as readouts while you drag the sides of the box. Default=0,0,0.

Length, **Width**, **Height Segments** Sets the number of divisions along each axis of the object. Can be set before or after creation. By default, each side of the box is a single segment. When you reset these values, the new values become the default during a session. Default=1,1,1.

TIP Increase the Segments settings to give objects extra resolution for being affected by modifiers. For example, if you're going to bend on page 1208 a box on the Z axis, you might want to set its Height Segments parameter to 4 or more.

Generate Mapping Coords Generates coordinates for applying mapped materials to the box. Default=on.

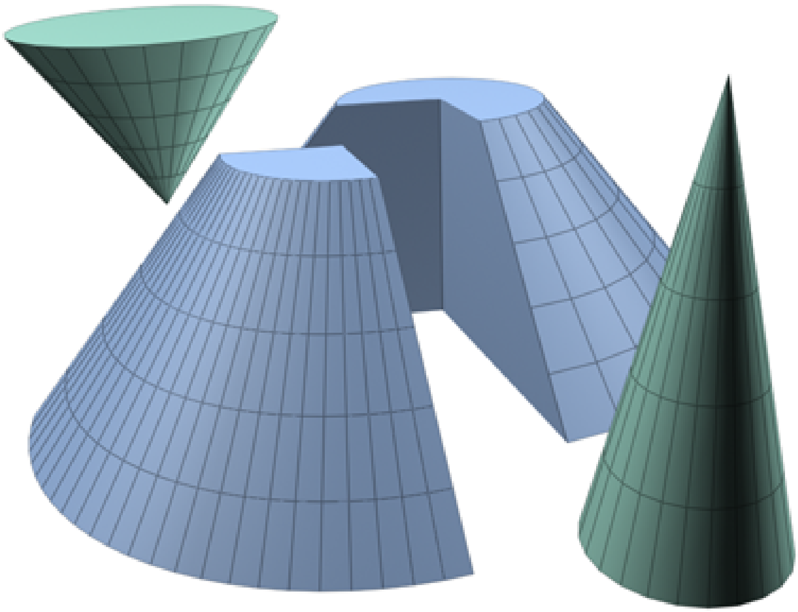
Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Cone Primitive

Create panel > Geometry button > Standard Primitives > Object Type rollout > Cone button

Create menu > Standard Primitives > Cone

The Cone button on the Creation command panel lets you produce round cones, either upright or inverted.



Examples of cones

Procedures

To create a cone:

1. On the Create menu choose Standard Primitives > Cone.
2. In any viewport, drag to define a radius for the base of the cone, then release to set it.
3. Move to up or down to define a height, either positive or negative, then click to set it.

4 Move to define a radius for the other end of the cone. Decrease this radius to 0 for a pointed cone.

5 Click to set the second radius and create the cone.

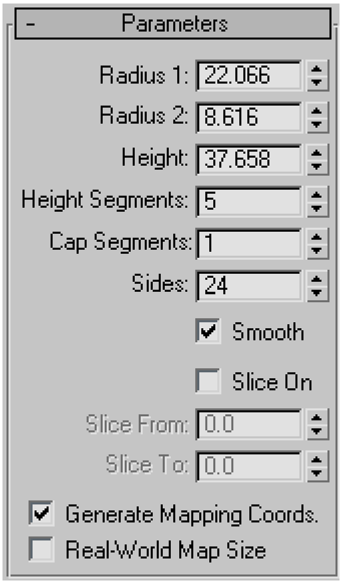
Interface

Creation Method rollout

Edge Draws a cone from edge to edge. You can change the center location by moving the mouse.

Center Draws a cone from the center out.

Parameters rollout



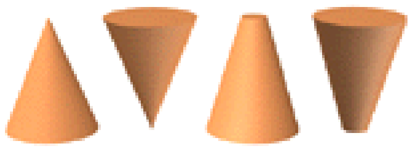
The defaults produce a smooth, round cone of 24 sides with five height segments, one cap segment, and the pivot point at the center of the base. For improved rendering of smoothly shaded cones, particularly those with pointed tips, increase the number of height segments.

Radius 1, Radius 2 Set the first and second radii for the cone. The minimum value for both is 0.0. If you enter a negative value, the software converts it to

0.0. You can combine these settings to create pointed and flat-topped cones, upright or inverted. The following combinations assume a positive height:

Radius Combinations	Effect
Radius 2 is 0	Creates a pointed cone
Radius 1 is 0	Creates an inverted pointed cone
Radius 1 is larger than Radius 2	Creates a flat-topped cone
Radius 2 is larger than Radius 1	Creates an inverted flat-topped cone

If Radius 1 and 2 are the same, a cylinder is created. If the two radius settings are close in size, the effect is similar to applying a Taper modifier to a cylinder.



Effect of Radius settings

Height Sets dimension along the central axis. Negative values create the cone below the construction plane.

Height Segments Sets the number of divisions along the cone's major axis.

Cap Segments Sets the number of concentric divisions around the center of the cone's top and bottom.

Sides Sets the number of sides around the cone. Higher numbers shade and render as true circles with Smooth selected. Lower numbers create regular polygonal objects with Smooth off.

Smooth Blends the faces of the cone, creating a smooth appearance in rendered views.

Slice On Enables the Slice function. Default=off.

When you create a slice and then turn off Slice On, the complete cone reappears. You can use this check box to switch between the two topologies.

Slice From, Slice To Sets the number of degrees around the local Z axis from a zero point at the local X axis.

For both settings, positive values move the end of the slice counterclockwise; negative values move it clockwise. Either setting can be made first. When the ends meet, the whole cone reappears.

Generate Mapping Coords Generates coordinates for applying mapped materials to the cone. Default=on.

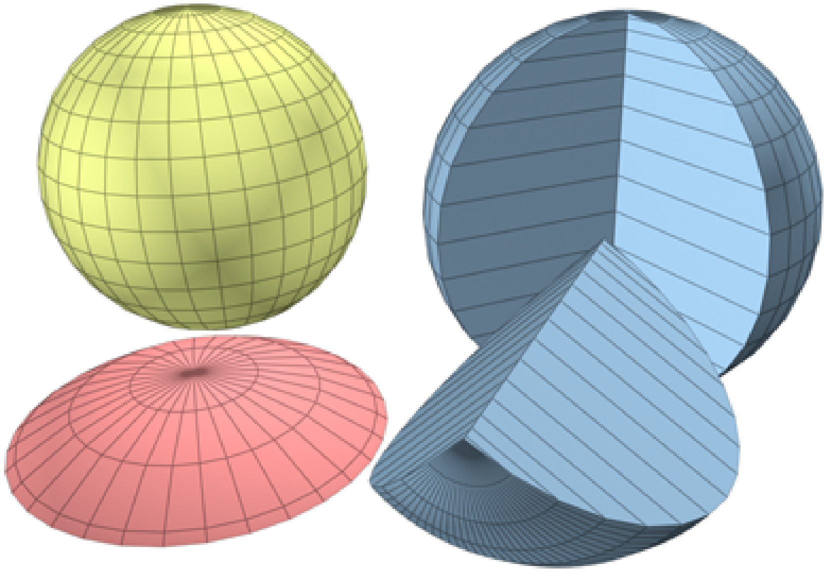
Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Sphere Primitive

Create panel > Geometry button > Standard Primitives > Object Type rollout > Sphere button

Create menu > Standard Primitives > Sphere

Sphere produces a full sphere, or a hemisphere or other portion of a sphere. You can also "slice" a sphere about its vertical axis.



Examples of spheres

Procedures

To create a sphere:

1. On the Create menu choose Standard Primitives > Sphere.
2. In any viewport, drag to define a radius.

   As you drag, a sphere emerges with its center at the pivot point.
3. Release the mouse to set the radius and create the sphere.

To create a hemisphere:

You can reverse the order of the following steps, if you like.

1. Create a sphere of desired radius.
2. Type **0.5** in the Hemisphere field.

   The sphere is reduced to exactly the upper half, a hemisphere. If you use the spinner, the sphere changes in size.

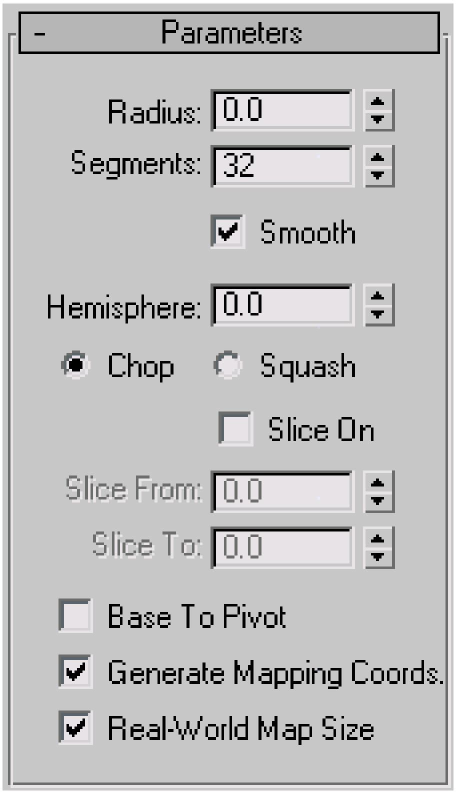
Interface

Creation Method rollout

Edge Draws a sphere from edge to edge. You can change the center location by moving the mouse.

Center Draws a sphere from the center out.

Parameters rollout



The defaults produce a smooth sphere of 32 segments with the pivot point at its center.

Radius Specifies the radius of the sphere.

Segments Sets the number of polygonal divisions for the sphere.

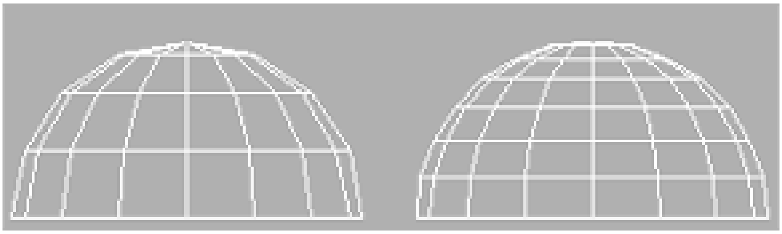
Smooth Blends the faces of the sphere, creating a smooth appearance in rendered views.

Hemisphere Increasing values progressively will "cut off" the sphere, starting at the base, to create a partial sphere. Values range from 0.0 to 1.0. The default is 0.0, producing a full sphere. A setting of 0.5 produces a hemisphere, and 1.0 reduces the sphere to nothing. Default=0.0.

Chop and Squash toggle creation options for Hemisphere.

Chop Reduces the number of vertices and faces in the sphere by "chopping" them out as the hemisphere is cut off. Default=on.

Squash Maintains the number of vertices and faces in the original sphere, "squashing" the geometry into a smaller and smaller volume toward the top of the sphere.



Effects of Chop and Squash during hemisphere creation

Slice On Uses the From and To angles to create a partial sphere. The effect is similar to lathing a semicircular shape fewer than 360 degrees.

Slice From Sets the start angle.

Slice To Sets the stop angle.

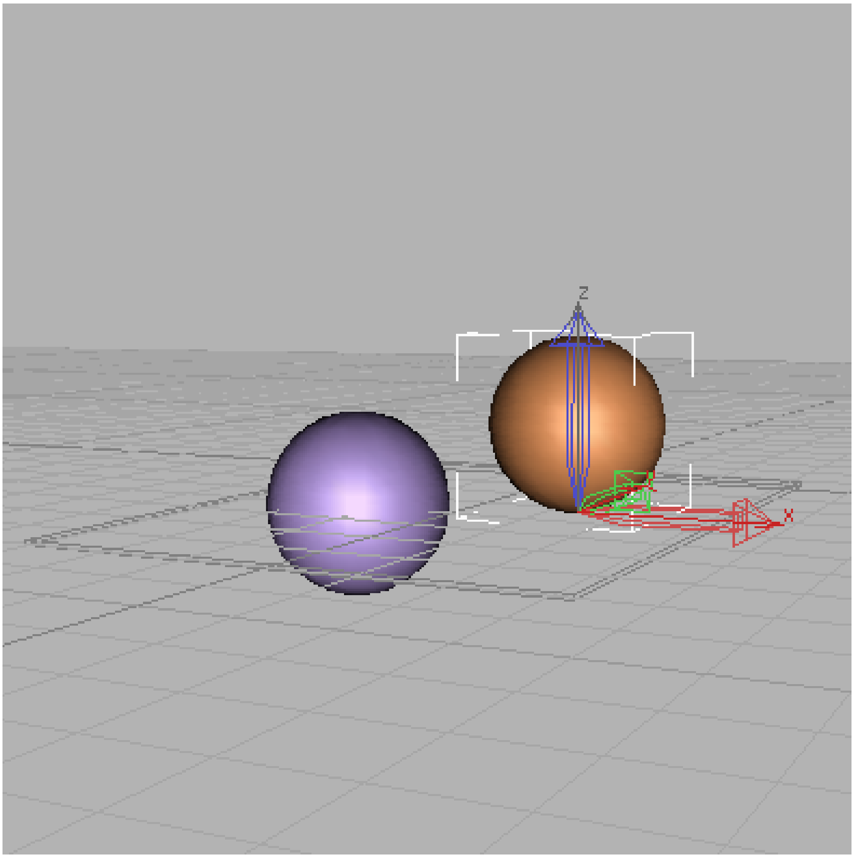
For both settings, positive values move the end of the slice counterclockwise; negative values move it clockwise. Either setting can be made first. When the ends meet, the whole sphere reappears.

Smoothing groups are assigned to sliced spheres as follows: The surface of the sphere is always assigned group 1; the bottom, when Smooth is on, gets group 2. Facing the pie-slice surfaces, the cut on the left gets group 3, and the cut on the right gets group 4.

Material IDs are assigned to sliced spheres as follows: The bottom is 1 (when Hemisphere is greater than 0.0), the surface is 2, and the slice surfaces are 3 and 4.

Base To Pivot Moves a sphere upward along its local Z axis so the pivot point is at its base. When off, the pivot point is on the construction plane at the center of the sphere. Default=off.

Turning on Base To Pivot lets you place spheres so they rest on the construction plane, like pool balls on a table. It also lets you animate a hemisphere so it appears to grow out of the construction plane or sink into it.



Effect of using Base To Pivot setting

Generate Mapping Coords Generates coordinates for applying mapped materials to the sphere. Default=on.

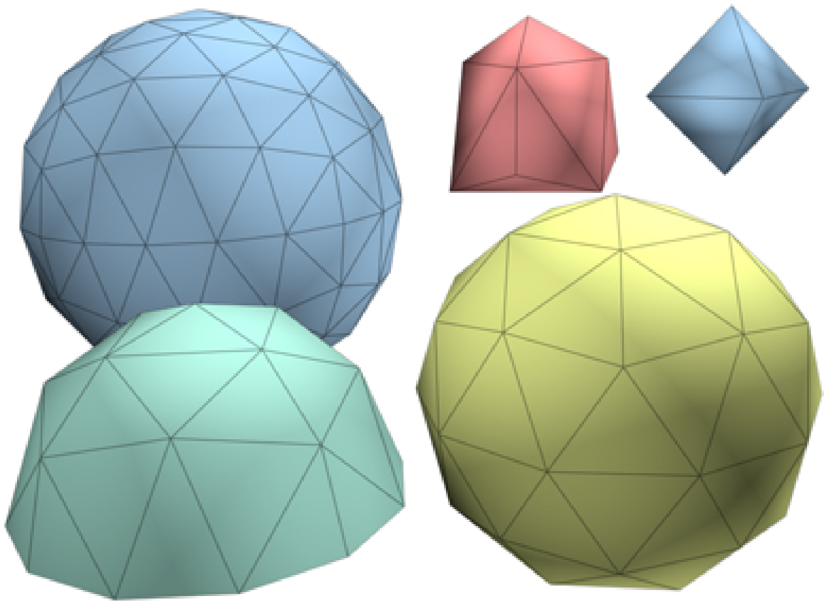
Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

GeoSphere Primitive

Create panel > Geometry button > Standard Primitives > Object Type rollout > GeoSphere button

Create menu > Standard Primitives > GeoSphere

Use GeoSphere to make spheres and hemispheres based on three classes of regular polyhedrons.



Examples of geospheres

Geospheres produce a more regular surface than standard spheres. They also render with a slightly smoother profile than a standard sphere given the same number of faces. Unlike a standard sphere, a geosphere has no poles, which can be an advantage when you apply certain modifiers such as Free-Form Deformation (FFD) modifiers on page 1460.

Procedures

To create a geosphere:

1. On the Create menu choose Standard Primitives > Geosphere.
2. In any viewport, drag to set the center and radius of the geosphere.
3. Set parameters such as Geodesic Base Type and Segments.

To create a geo-hemisphere:

1. Create a geosphere.
2. In the Parameters rollout, turn on the Hemisphere check box. The geosphere is converted to a hemisphere.

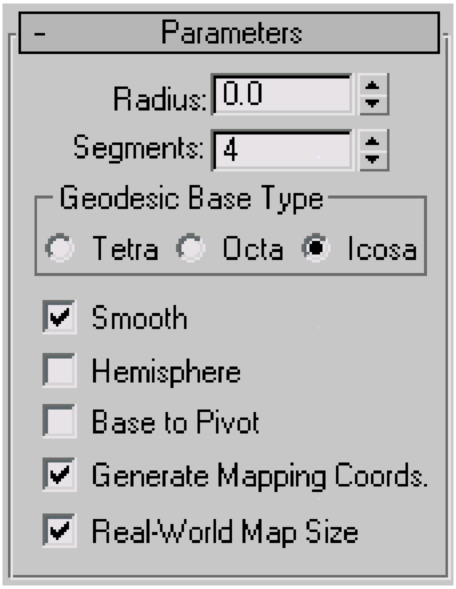
Interface

Creation Method rollout

Diameter Draws a geosphere from edge to edge. You can change the center location by moving the mouse.

Center Draws a geosphere from the center out.

Parameters rollout



Radius Sets the size of the geosphere.

Segments Sets the total number of faces in the geosphere. The number of faces in a geosphere is equal to the sides of the base polyhedron times the segments squared.

Lower segment values work best. Using the maximum segment value of 200 can generate up to 800,000 faces, impairing performance.

Geodesic Base Type group

Lets you choose one of three types of regular polyhedrons for the geosphere's basic geometry.

- **Tetra** Based on a four-sided tetrahedron. The triangular facets can vary in shape and size. The sphere can be divided into four equal segments.
- **Octa** Based on an eight-sided octahedron. The triangular facets can vary in shape and size. The sphere can be divided into eight equal segments.
- **Icosa** Based on a 20-sided icosahedron. The facets are all equally sized equilateral triangles. The sphere can be divided into any number of equal segments, based on multiples and divisions of 20 faces.

Smooth Applies smoothing groups to the surface of the sphere.

Hemisphere Creates a half-sphere.

Base To Pivot Sets the pivot point location. When on, the pivot is at the bottom of the sphere. When off, the pivot is at the center of the sphere. This option has no effect when Hemisphere is on.

Generate Mapping Coords Generates coordinates for applying mapped materials to the geosphere. Default=on.

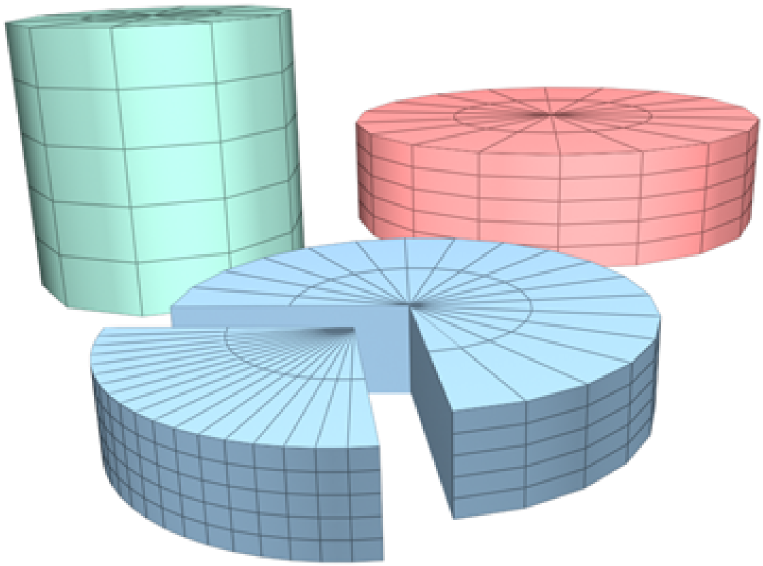
Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Cylinder Primitive

Create panel > Geometry button > Standard Primitives > Object Type rollout > Cylinder button

Create menu > Standard Primitives > Cylinder

Cylinder produces a cylinder, which you can "slice" around its major axis.



Examples of cylinders

Procedures

To create a cylinder:

1. On the Create panel, choose Standard Primitives > Cylinder.
2. In any viewport, drag to define the radius of the base, then release to set the radius.
3. Move up or down to define a height, either positive or negative.
4. Click to set the height and create the cylinder.

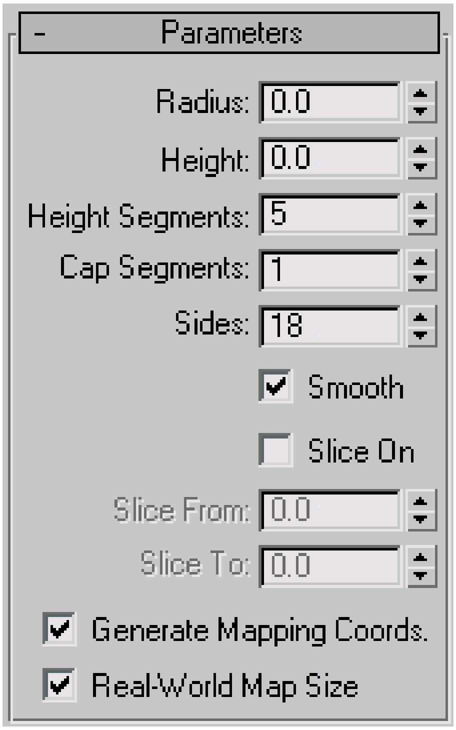
Interface

Creation Method rollout

Edge Draws a cylinder from edge to edge. You can change the center location by moving the mouse.

Center Draws a cylinder from the center out.

Parameters rollout



The defaults produce a smooth cylinder of 18 sides with the pivot point at the center of the base. There are five height segments and one cap segment. If you don't plan to modify the cylinder's shape, such as with a Bend modifier, set Height Segments to 1 to reduce scene complexity. If you plan to modify the ends of the cylinder, consider increasing the Cap Segments setting.

Radius Sets the radius of the cylinder.

Height Sets the dimension along the central axis. Negative values create the cylinder below the construction plane.

Height Segments Sets the number of divisions along the cylinder's major axis.

Cap Segments Sets the number of concentric divisions around the center of the cylinder's top and bottom.

Sides Sets the number of sides around the cylinder. With Smooth on, higher numbers shade and render as true circles. With Smooth off, lower numbers create regular polygonal objects.

Smooth The faces of the cylinder are blended together, creating a smooth appearance in rendered views.

Slice On Enables the Slice function. Default=off.

When you create a slice and then turn off Slice On, the complete cylinder reappears. You can use this check box to switch between the two topologies.

Slice From, Slice To Sets the number of degrees around the local Z axis from a zero point at the local X axis.

For both settings, positive values move the end of the slice counterclockwise; negative values move it clockwise. Either setting can be made first. When the ends meet, the whole cylinder reappears.

Generate Mapping Coords Generates coordinates for applying mapped materials to the cylinder. Default=on.

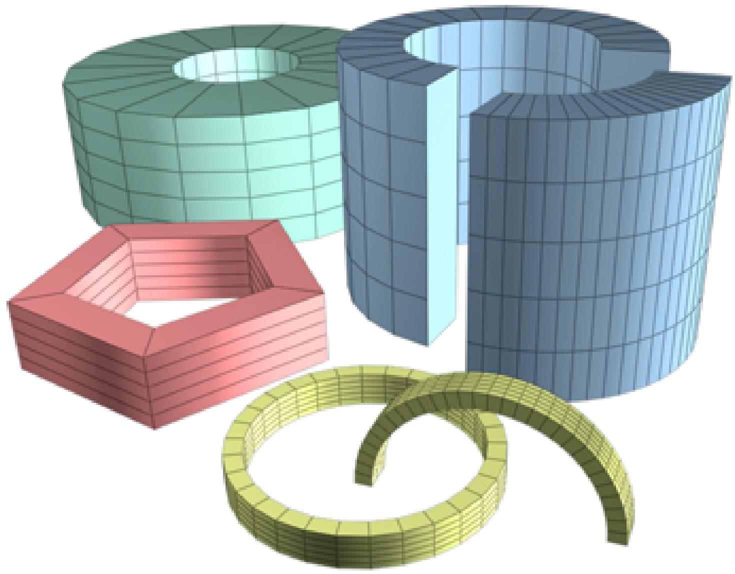
Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Tube Primitive

Create panel > Geometry button > Standard Primitives > Object Type rollout > Tube button

Create menu > Standard Primitives > Tube

Tube produces both round and prismatic tubes. The tube is similar to the cylinder with a hole in it.



Examples of tubes

Procedures

To create a tube:

1 On the Create menu choose Standard Primitives > Tube.

2 In any viewport, drag to define the first radius, which can be either the inner or outer radius of the tube. Release to set the first radius.

3 Move to define the second radius, then click to set it.

4 Move up or down to define a height, either positive or negative.

5 Click to set the height and create the tube.

To create a prismatic tube:

1 Set the number of sides for the kind of prism you want.

2 Turn Smooth off.

3 Create a tube.

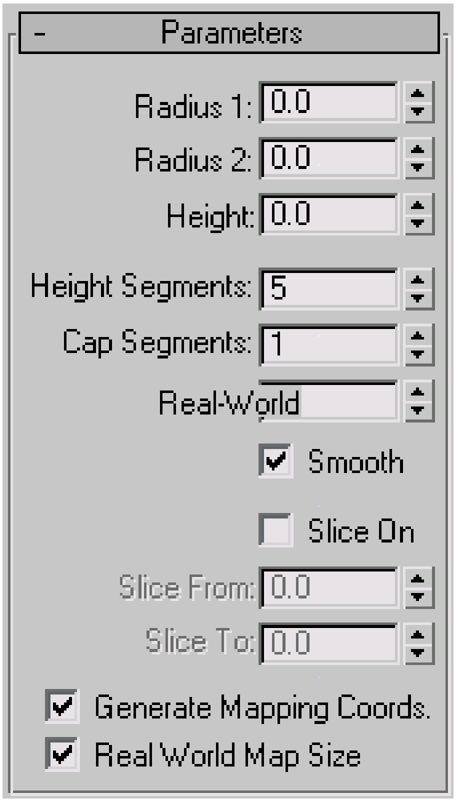
Interface

Creation Method rollout

Edge Draws a tube from edge to edge. You can change the center location by moving the mouse.

Center Draws a tube from the center out.

Parameters rollout



The defaults produce a smooth, round tube of 18 sides with the pivot point at the center of the base. There are five height segments and one cap segment. If you don't plan to modify the cylinder's shape, such as with a Bend modifier, set Height Segments to 1 to reduce scene complexity. If you plan to modify the ends of the cylinder, consider increasing the Cap Segments setting.

Radius 1, Radius 2 The larger setting specifies the outside radius of the tube, while the smaller specifies the inside radius.

Height Sets the dimension along the central axis. Negative values create the tube below the construction plane.

Height Segments Sets the number of divisions along the tube's major axis.

Cap Segments Sets the number of concentric divisions around the center of the tube's top and bottom.

Sides Sets the number of sides around the tube. Higher numbers shade and render as true circles with Smooth on. Lower numbers create regular polygonal objects with Smooth off.

Smooth When on (the default), faces of the tube are blended together, creating a smooth appearance in rendered views.

Slice On Enables the Slice feature, which removes part of the tube's circumference. Default=off.

When you create a slice and then turn off Slice On, the complete tube reappears. You can therefore use this check box to switch between the two topologies.

Slice From, Slice To Sets the number of degrees around the local Z axis from a zero point at the local X axis.

For both settings, positive values move the end of the slice counterclockwise; negative values move it clockwise. Either setting can be made first. When the ends meet, the whole tube reappears.

Generate Mapping Coords Generates coordinates for applying mapped materials to the tube. Default=on.

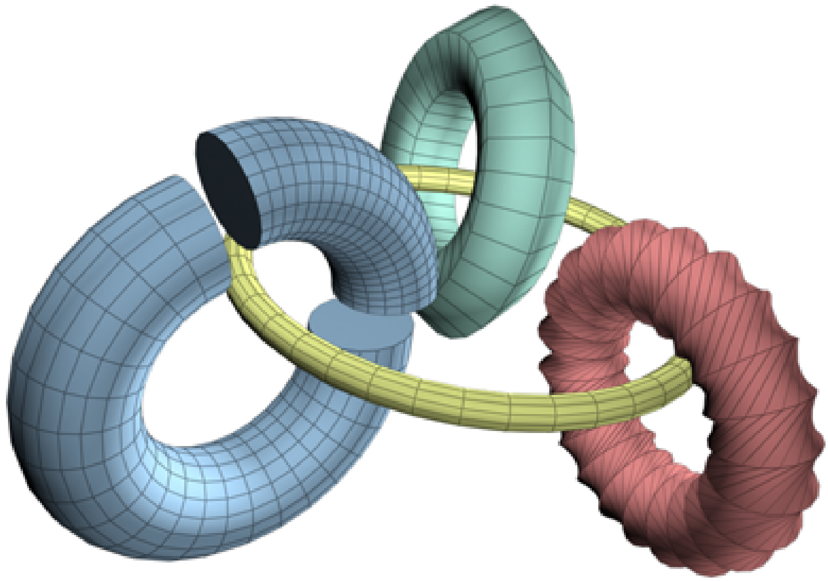
Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Torus Primitive

Create panel > Geometry button > Standard Primitives > Object Type rollout > Torus button

Create menu > Standard Primitives > Torus

Torus produces a torus, or a ring with a circular cross section, sometimes referred to as a doughnut. You can combine three smoothing options with rotation and twist settings to create complex variations.



Examples of tori

Procedures

To create a torus:

1. From the Create menu, choose Standard Primitives > Torus.
2. In any viewport, drag to define a torus.
3. As you drag, a torus emerges with its center at the pivot point.
4. Release to set the radius of the torus ring.
5. Move to define the radius of the cross-sectional circle, then click to create the torus.

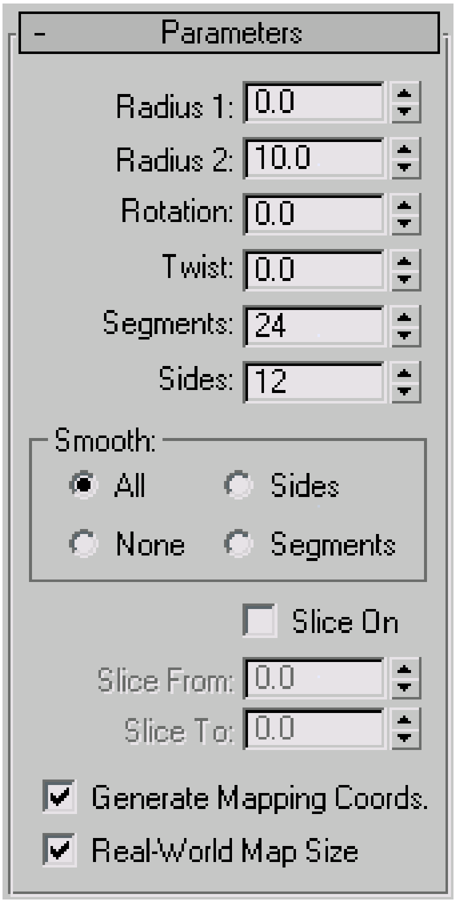
Interface

Creation Method rollout

Edge Draws a torus from edge to edge. You can change the center location by moving the mouse.

Center Draws a torus from the center out.

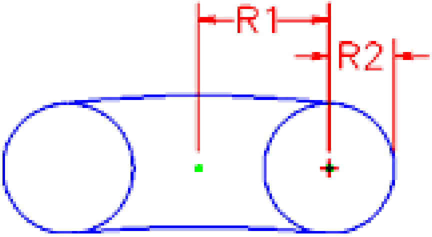
Parameters rollout



The defaults produce a smooth torus with 12 sides and 24 segments. The pivot point is at the center of the torus on the plane, cutting through the center of the torus. Higher settings for sides and segments produce a more dense geometry that might be required for some modeling or rendering situations.

Radius 1 Sets the distance from the center of the torus to the center of the cross-sectional circle. This is the radius of the torus ring.

Radius 2 Sets the radius of the cross-sectional circle. This value is replaced each time you create a torus. Default = 10.



Radius 1 and Radius 2

Rotation Sets the degree of rotation. Vertices are uniformly rotated about the circle running through the center of the torus ring. Positive and negative values for this setting "roll" the vertices in either direction over the surface of the torus.

Rotation and Twist

Twist Sets the degree of twist. Cross sections are progressively rotated about the circle running through the center of the torus. Beginning with twist, each successive cross section is rotated until the last one has the number of degrees specified.

Twisting a closed (unsliced) torus creates a constriction in the first segment. You can avoid this by either twisting in increments of 360 degrees, or by turning Slice on and setting both Slice From and Slice To to 0 to maintain a complete torus.

Segments Sets the number of radial divisions around the torus. By reducing this number, you can create polygonal rings instead of circular ones.

Sides Sets the number of sides on the cross-sectional circle of the torus. By reducing this number, you can create prism-like cross sections instead of circular ones.

Smooth group

Choose one of four levels of smoothing:

- **All** (default) Produces complete smoothing on all surfaces of the torus.
- **Sides** Smoothes the edges between adjacent segments, producing smooth bands running around the torus.
- **None** Turns off smoothing entirely, producing prism-like facets on the torus.
- **Segments** Smoothes each segment individually, producing ring-like segments along the torus.

Slice On Creates a portion of a sliced torus rather than the entire 360 degrees.

Slice From When Slice On is on, specifies the angle where the torus slice begins.

Slice To When Slice On is on, specifies the angle where the torus slice ends.

Generate Mapping Coords Generates coordinates for applying mapped materials to the torus. Default=on.

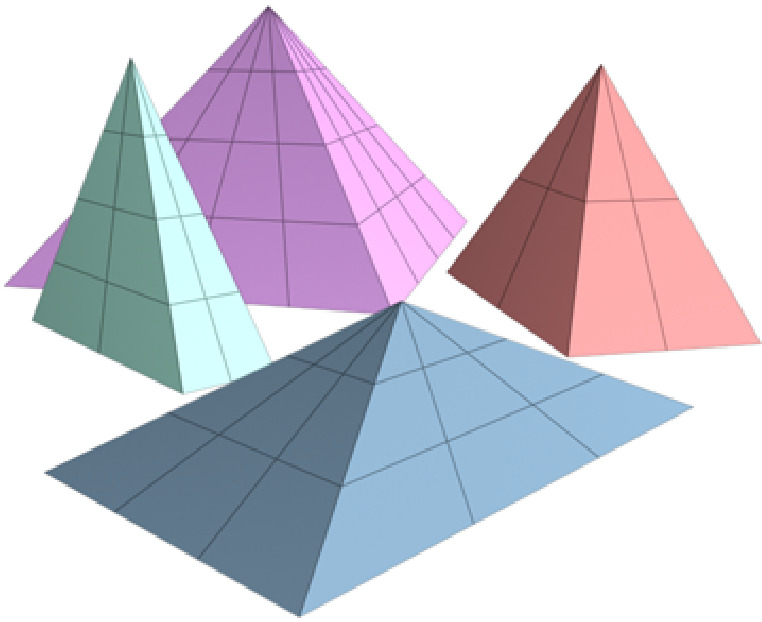
Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Pyramid Primitive

Create panel > Geometry button > Standard Primitives > Object Type rollout > Pyramid button

Create menu > Standard Primitives > Pyramid

The Pyramid primitive has a square or rectangular base and triangular sides.



Examples of pyramids

Procedures

To create a Pyramid:

1. On the Create menu choose Standard Primitives > Pyramid.
2. Choose a creation method, either Base/Apex or Center.

   NOTE Hold the Ctrl key while using either creation method to constrain the base to a square.

3. In any viewport, drag to define the base of the pyramid. If you're using Base/Apex, define the opposite corners of the base, moving the mouse horizontally or vertically to define the width and depth of the base. If you're using Center, drag from the center of the base.
4. Click, and then move the mouse to define the Height.
5. Click to complete the pyramid.

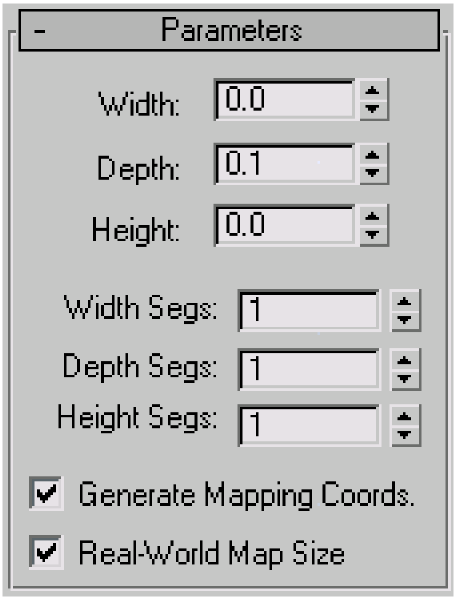
Interface

Creation Method rollout

Base/Apex Creates the pyramid base from one corner to the diagonally opposite corner.

Center Creates the pyramid base from the center out.

Parameters rollout



Width, Depth and Height Sets the dimension of the corresponding side of the pyramid.

Width, Depth and Height Segs Sets the number of segments to the corresponding sides of the pyramid.

Generate Mapping Coords Generates coordinates for applying mapped materials to the pyramid. Default=on.

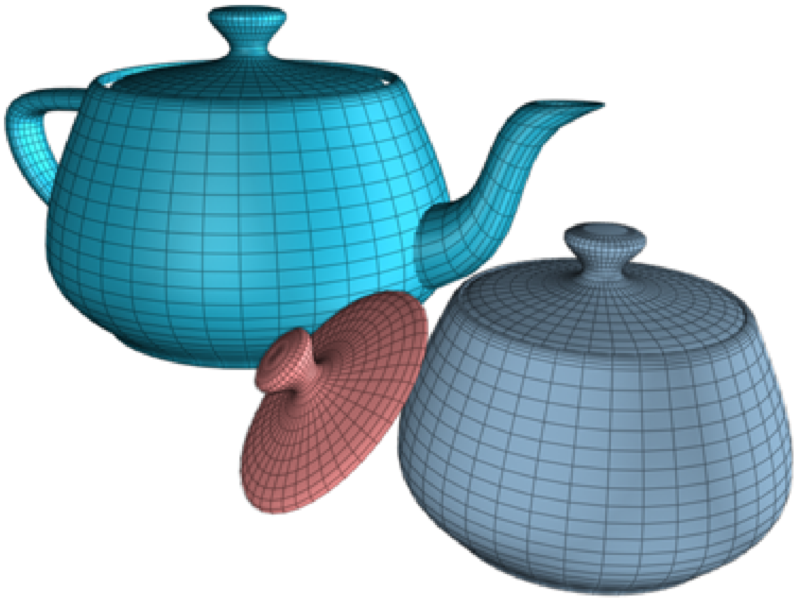
Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Teapot Primitive

Create panel > Geometry button > Standard Primitives > Object Type rollout > Teapot button

Create menu > Standard Primitives > Teapot

Teapot produces a teapot. You can choose to make the whole teapot at once (the default), or any of its parts. Since the Teapot is a parametric object, you can choose which parts of the teapot to display after creation.



Examples of teapots

History of the Teapot

This teapot derives from the original data developed by Martin Newell in 1975. Beginning with a graph-paper sketch of a teapot that he kept on his desk, Newell calculated cubic Bezier splines on page 7922 to create a wireframe model. James Blinn, also at the University of Utah during this period, produced early renderings of exceptional quality using this model.

The teapot has since become a classic in computer graphics. Its complexly curved and intersecting surfaces are well suited to testing different kinds of material mappings and rendering settings on a real-world object.

Procedures

To create a teapot:

1 On the Create menu, choose Standard Primitives > Teapot.

2 In any viewport, drag to define a radius.

  As you drag, a teapot emerges with the pivot point at the center of its base.

3 Release the mouse to set the radius and create the teapot.

To create a teapot part:

1 In Parameters rollout > Teapot Parts group, turn off all parts except the one you want to create.

2 Create a teapot.

  The part you left on appears. The pivot point remains at the center of the teapot's base.

3 In Parameters rollout > Teapot Parts group, turn off all parts except the one you want.

The teapot has four separate parts: body, handle, spout, and lid. Controls are located in the Teapot Parts group of the Parameters rollout. You can check any combination of parts to create at the same time. The body alone is a ready-made bowl, or a pot with optional lid.

To turn a part into a teapot:

1 Select a teapot part in the viewport.

2 On the Modify panel > Parameters rollout, turn on all parts. (This is the default.)

  The whole teapot appears.

You can apply modifiers to any separate part. If you later turn on another part, the modifier affects the additional geometry as well.

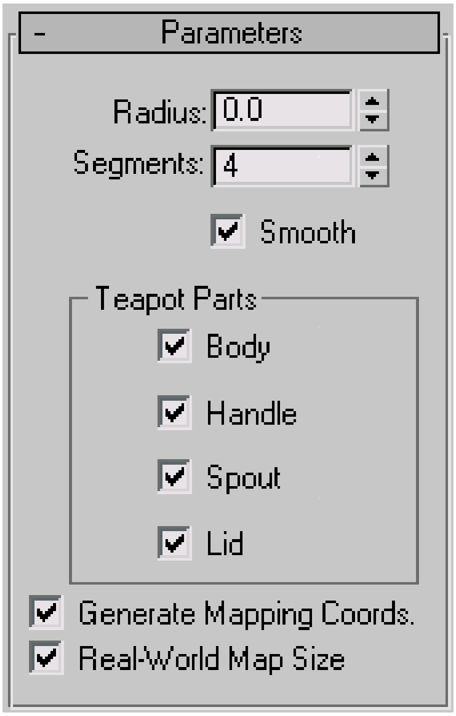
Interface

Creation Method rollout

Edge Draws a teapot from edge to edge. You can change the center location by moving the mouse.

Center Draws a teapot from the center out.

Parameters rollout



Radius Sets the radius of the teapot

Segments Sets the number of divisions for the teapot or its individual parts.

Smooth Blends faces of the teapot, creating a smooth appearance in rendered views.

Teapot Parts group

Turn check boxes on or off for teapot parts. By default, all are on, producing a complete teapot.

Generate Mapping Coords Generates coordinates for applying mapped materials to the teapot. Default=on.

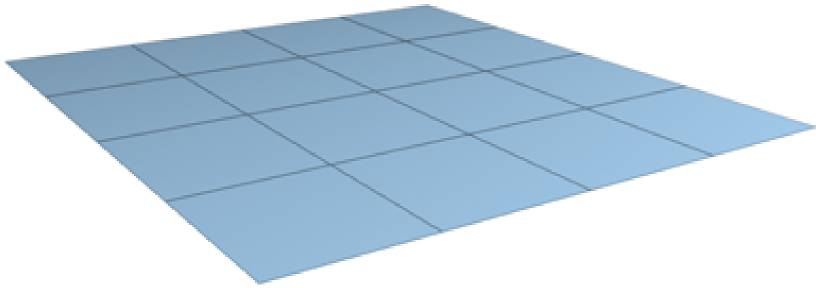
Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Plane Primitive

Create panel > Geometry button > Standard Primitives > Object Type rollout > Plane button

Create menu > Standard Primitives > Plane

The Plane object is a special type of flat polygon mesh that can be enlarged by any amount at render time. You can specify factors to magnify the size or number of segments, or both. Use the Plane object for creating a large-scale ground plane that doesn't get in the way when working in a viewport. You can apply any type of modifier to the plane object, such as Displace on page 1344 to simulate a hilly terrain.



Example of plane

Procedures

To create a plane:

1 On the Create menu choose Standard Primitives > Plane.

2 In any viewport, drag to create the Plane.

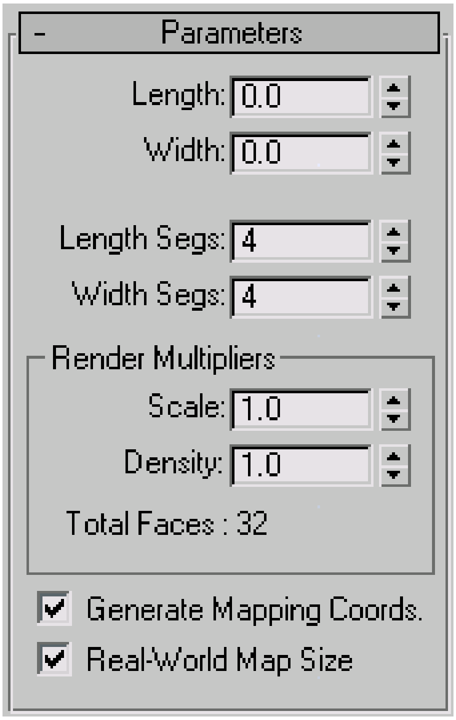
Interface

Creation Method rollout

Rectangle Creates the plane primitive from one corner to the diagonally opposite corner, interactively setting different values for length and width.

Square Creates a square plane where length and width are equal. You can change dimensions in the Parameters rollout subsequent to creation.

Parameters rollout



Length, **Width** Sets the length and width of the plane object. These fields act also as readouts while you drag the sides of the box. You can revise these values. Defaults= 0.0, 0.0.

Length Segs, **Width Segs** Sets the number of divisions along each axis of the object. Can be set before or after creation. By default, each side of the plane has four segments. When you reset these values, the new values become the default during a session.

Render Multipliers group

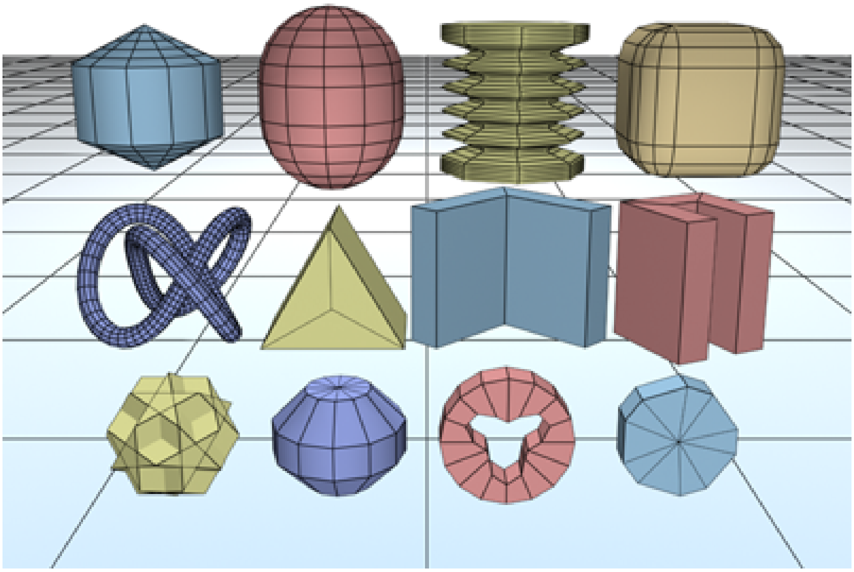
Render Scale Specifies the factor by which both length and width are multiplied at render time. Scaling is performed from the center outward.

Render Segs Specifies the factor by which the number of segments in both length and width are multiplied at render time.

Generate Mapping Coords Generates coordinates for applying mapped materials to the plane. Default=on.

Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

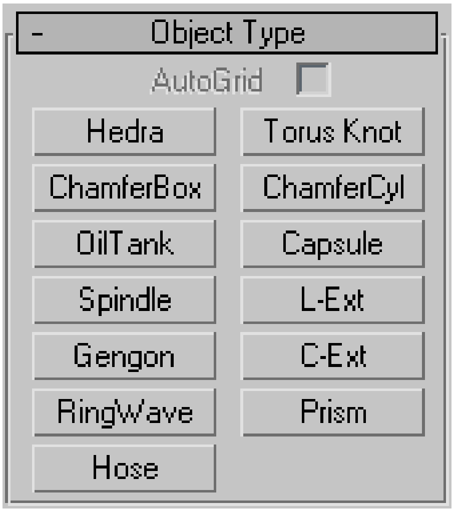
Extended Primitives



A collection of extended primitive objects

Extended Primitives are a collection of complex primitives for 3ds Max. The topics that follow describe each type of extended primitive and its creation parameters.

These primitives are available from the Object Type rollout on the Create panel and from the Create menu > Extended Primitives.



AutoGrid on page 2597

Hedra on page 445

Torus Knot on page 448

ChamferBox on page 452

ChamferCyl on page 456

OilTank on page 459

Capsule on page 462

Spindle on page 465

L-Ext on page 469

Gengon on page 471

C-Ext on page 474

RingWave on page 477

Prism on page 483

Hose on page 485

All primitives have name and color controls, and allow you to enter initial values from the keyboard. See these topics:

Object Name and Wireframe Color on page 7631

Creating Primitives from the Keyboard on page 406 (not applicable to Hedra, RingWave, or Hose)

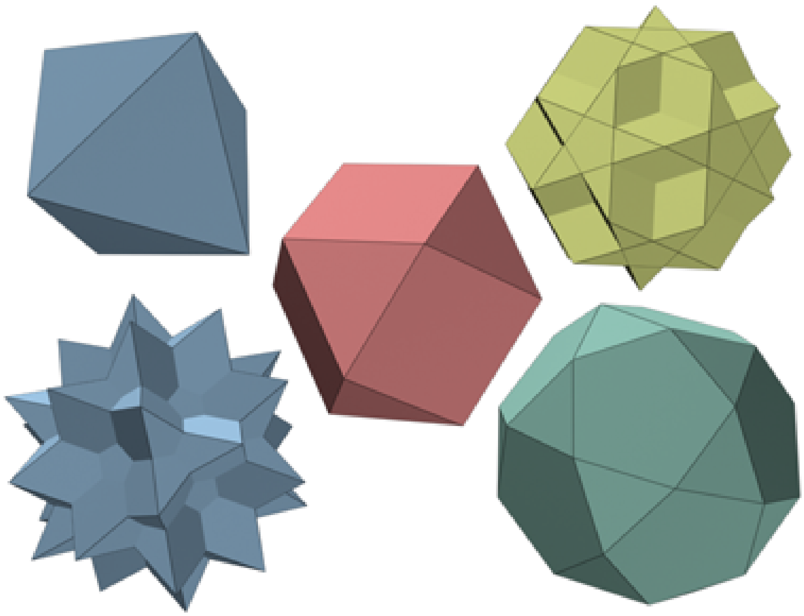
The remaining rollouts are covered in the topic for each primitive.

Hedra Extended Primitive

Create panel > Geometry button > Extended Primitives > Object Type rollout > Hedra button

Create menu > Extended Primitives > Hedra

Use Hedra to produce objects from several families of polyhedra.



Examples of hedra

Procedures

To create a polyhedron:

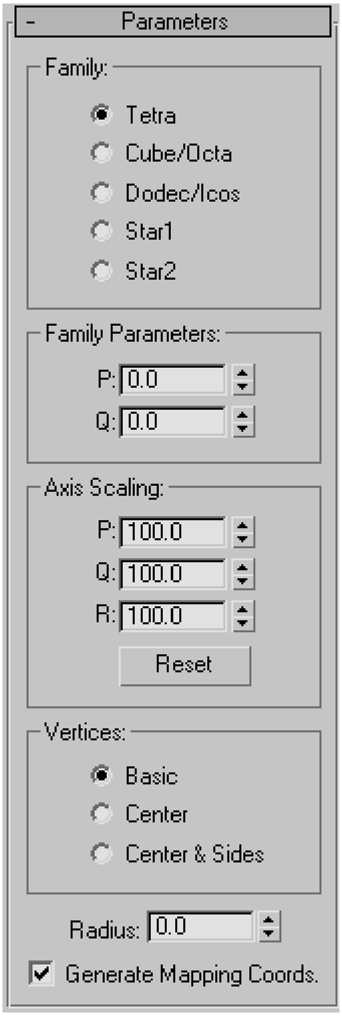
1 From the Create menu, choose Extended Primitives > Hedra.

2 In any viewport, drag to define a radius, then release to create the polyhedron.

As you drag, a polyhedron emerges from the pivot point.

3 Adjust the Family Parameter and Axis Scaling spinners to vary the Hedra's appearance.

Interface



Family group

Use this group to select the type of polyhedron to create.

Tetra Creates a tetrahedron.

Cube/Octa Creates a cubic or octahedral polyhedron (depending on parameter settings).

Dodec/Icos Creates a dodecahedron or icosahedron (depending on parameter settings).

Star1/Star2 Creates two different star-like polyhedra.

TIP You can animate between Hedra types. Turn on the Auto Key button, go to any frame, and change the Family check box. There is no interpolation between types; the model simply jumps from a star to a cube or tetrahedron, and so on.

Family parameters group

P, **Q** Interrelated parameters that provide a two-way translation between the vertices and facets of a polyhedron. They share the following:

- Range of possible values is 0.0 through 1.0.
- The combined total of the P and Q values can be equal to or less than 1.0.
- Extremes occur if either P or Q is set to 1.0; the other is automatically set to 0.0.
- Midpoint occurs when both P and Q are 0.

In the simplest terms, P and Q change the geometry back and forth between vertices and facets. At the extreme settings for P and Q, one parameter represents all vertices, the other represents all facets. Intermediate settings are transition points, with the midpoint an even balance between the two parameters.

Axis Scaling group

Polyhedra can have as many as three kinds of polygonal facets, such as triangle, square, or pentagon. These facets can be regular or irregular. If a polyhedron has only one or two types of facet, only one or two of the axis scaling parameters are active. Inactive parameters have no effect.

P, Q, R Controls the axis of reflection for one of the facets of a polyhedron. In practice, these fields have the effect of pushing their corresponding facets in and out. Defaults=100.

Reset Returns axes to their default setting.

Vertices group

Parameters in the Vertices group determine the internal geometry of each facet of a polyhedron. Center and Center & Sides increase the number of vertices in the object and therefore the number of faces. These parameters cannot be animated.

Basic Facets are not subdivided beyond the minimum.

Center Each facet is subdivided by placing an additional vertex at its center, with edges from each center point to the facet corners.

Center & Sides Each facet is subdivided by placing an additional vertex at its center, with edges from each center point to the facet corners, as well as to the center of each edge. Compared to Center, Center & Sides doubles the number of faces in the polyhedron.

NOTE If you scale the axis of the object, the Center option is used automatically, unless Center & Sides is already set.

To see the internal edges shown in the figure, turn off Edges Only on the Display command panel.

Radius Sets the radius of any polyhedron in current units.

Generate Mapping Coords Generates coordinates for applying mapped materials to the polyhedron. Default=on.

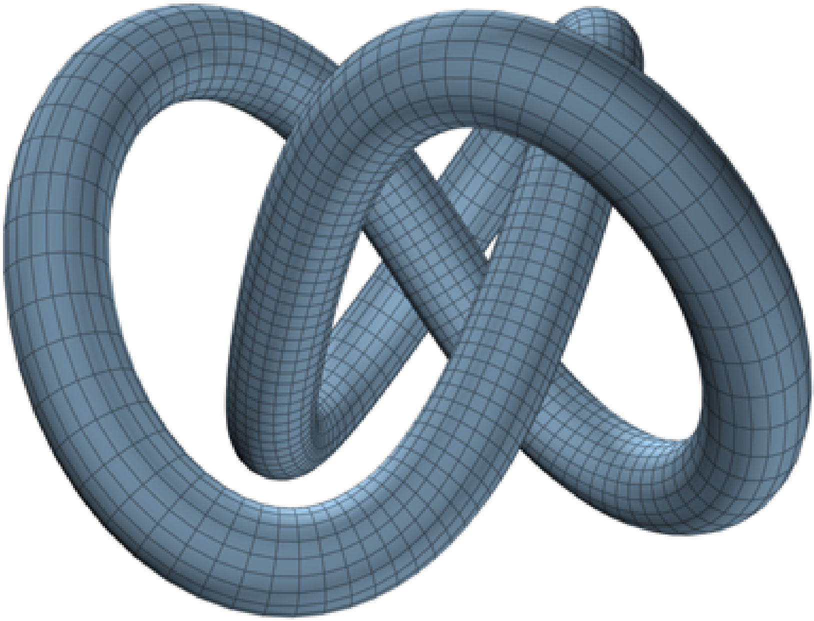
Torus Knot Extended Primitive

Create panel > Geometry > Extended Primitives > Object Type rollout > Torus Knot button

Create menu > Extended Primitives > Torus Knot

Use Torus Knot to create a complex or knotted torus by drawing 2D curves in the normal planes around a 3D curve. The 3D curve (called the Base Curve) can be either a circle or a torus knot.

You can convert a torus knot object to a NURBS surface on page 2304.



Example of torus knot

Procedures

To create a Torus Knot:

1. On the Create menu, choose Extended Primitives > Torus Knot.
2. Drag the mouse to define the size of the torus knot.
3. Click, then move the mouse vertically to define the radius.
4. Click again to finish the torus.
5. Adjust the parameters on the Modify panel.

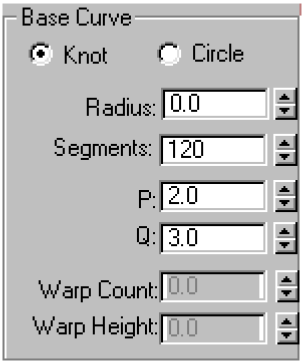
Interface

Creation Method rollout

Diameter Draws the object from edge to edge. You can change the center location by moving the mouse.

Radius Draws the object from the center out.

Parameters rollout > Base Curve group



Provides parameters that affect the base curve.

Knot/Circle With Knot, the torus interweaves itself, based on various other parameters. With Circle, the base curve is a circle, resulting in a standard torus if parameters such as Warp and Eccentricity are left at their defaults.

Radius Sets the radius of the base curve.

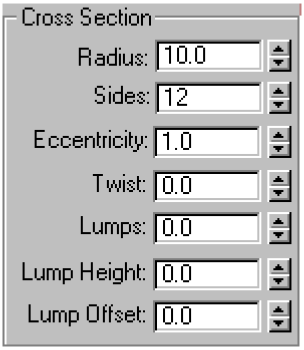
Segments Sets the number of segments around the perimeter of the torus.

P and Q Describes up-and-down (P) and around-the-center (Q) winding numbers. (Active only when Knot is chosen.)

Warp Count Sets the number of "points" in a star shape around the curve. (Active only when Circle is chosen.)

Warp Height Sets the height of the "points" given as a percentage of the base curve radius.

Parameters rollout > Cross Section group



Provides parameters that affect the cross section of the torus knot.

Radius Sets the radius of the cross section.

Sides Sets the number of sides around the cross section.

Eccentricity Sets the ratio of the major to minor axes of the cross section. A value of 1 provides a circular cross section, while other values create elliptical cross sections.

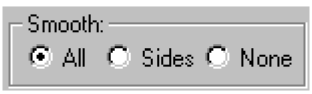
Twist Sets the number of times the cross section twists around the base curve.

Lumps Sets the number of bulges in the torus knot. Note that the Lump Height spinner value must be greater than 0 to see any effect.

Lump Height Sets the height of the lumps, as a percentage of the radius of the cross section. Note that the Lumps spinner must be greater than 0 to see any effect.

Lump Offset Sets the offset of the start of the lumps, measured in degrees. The purpose of this value is to animate the lumps around the torus.

Parameters rollout > Smooth group



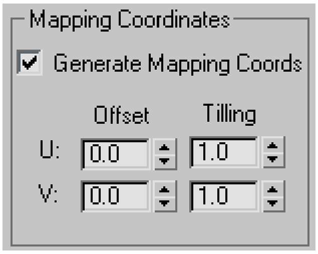
Provides options to alter the smoothing displayed or rendered of the torus knot. This smoothing does not displace or tesselate the geometry, it only adds the smoothing group information.

All Smoothes the entire torus knot.

Sides Smoothes only the adjacent sides of the torus knot.

None The torus knot is faceted.

Parameters rollout > Mapping Coordinates group



Provides methods of assigning and adjusting mapping coordinates.

Generate Mapping Coords Assigns mapping coordinates based on the geometry of the torus knot. Default=on.

Offset U/V Offset the mapping coordinates along U and V.

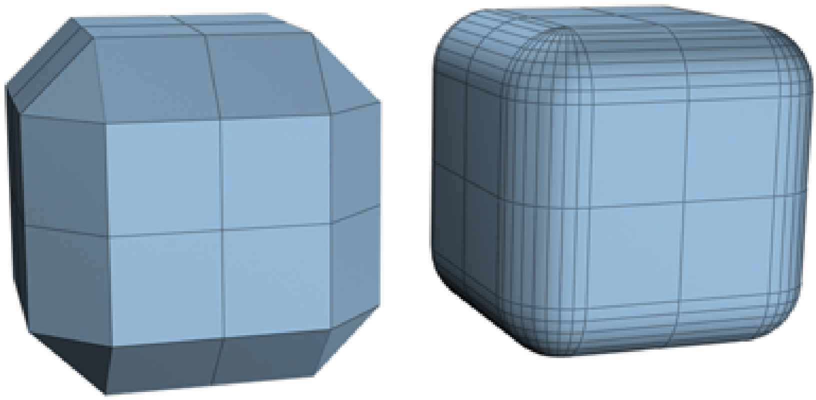
Tiling U/V Tile the mapping coordinates along U and V.

ChamferBox Extended Primitive

Create panel > Geometry button > Extended Primitives > Object Type rollout > ChamferBox button

Create menu > Extended Primitives > Chamfer Box

Use ChamferBox to create a box with beveled or rounded edges.



Examples of chamfered boxes

Procedures

To create a standard chamfered box:

1. From the Create menu, choose Extended Primitives > Chamfer Box.
2. Drag the mouse to define the diagonal corners of the base of the chamfered box. (Press Ctrl to constrain the base to a square.)
3. Release the mouse button, and then move the mouse vertically to define the height of the box. Click to set the height
4. Move the mouse diagonally to define the width of the fillet, or chamfer (toward the upper left increases the width; toward the lower right decreases it).
5. Click again to finish the chamfered box.

To create a cubic chamfered box:

1. On the Creation Method rollout, click Cube.

2 Beginning at the center of the cube, drag in a viewport to set all three dimensions simultaneously.

3 Release the button, and move the mouse to set the fillet or chamfer.

4 Click to create the object.

   You can change a cube's individual dimensions in the Parameters rollout.

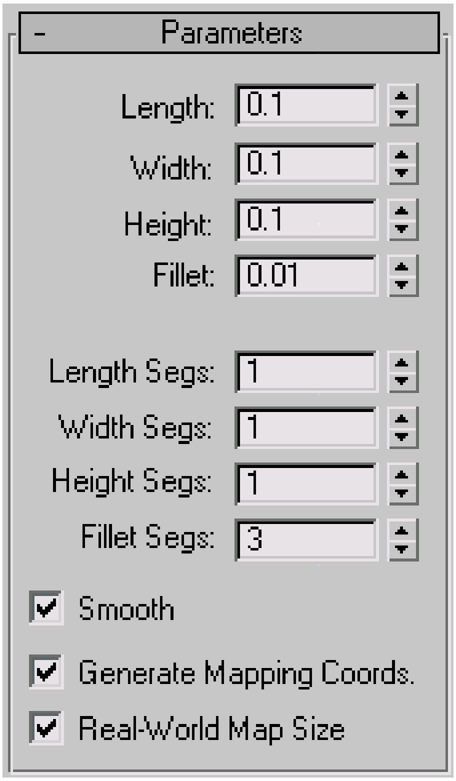
Interface

Creation Method rollout

Cube Forces length, width, and height to be equal. You can change a cube's individual dimensions in the Parameters rollout.

Box Creates a standard chamfered box primitive from one corner to the diagonally opposite corner, with individual settings for length, width, and height.

Parameters rollout



Length, **Width**, **Height** Sets the corresponding dimensions of the chamfered box.

Fillet Slices off the edges of the chamfered box. Higher values result in a more refined fillet on the edges of the chamfered box.

Length, **Width**, **Height Segs** Sets the number of divisions along the corresponding axis.

Fillet Segs Sets the number of segments in the filleted edges of the box. Adding fillet segments increases the edge roundness.

Smooth Blends the display of the faces of the chamfered box, creating a smooth appearance in rendered views.

Generate Mapping Coords Generates coordinates for applying mapped materials to the chamfered box. Default=on.

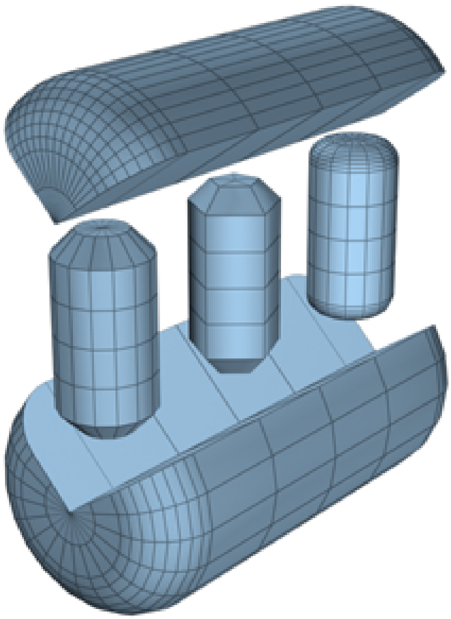
Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

ChamferCyl Extended Primitive

Create panel > Geometry button > Extended Primitives > Object Type rollout > ChamferCyl button

Create menu > Extended Primitives > Chamfer Cylinder

Use ChamferCyl to create a cylinder with beveled or rounded cap edges.



Examples of chamfered cylinders

Procedures

To create a chamfered cylinder:

1 From the Create menu, choose Extended Primitives > Chamfer Cylinder.

2. Drag the mouse to define the radius of the base of the chamfered cylinder.
3. Release the mouse button, and then move the mouse vertically to define the height of the cylinder. Click to set the height.
4. Move the mouse diagonally to define the width of the fillet, or chamfer (toward the upper left increases the width; toward the lower right decreases it).
5. Click to finish the cylinder.

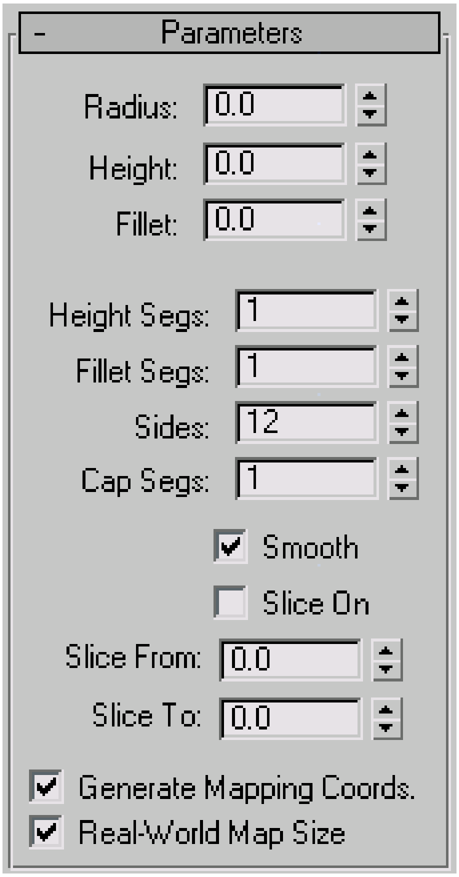
Interface

Creation Method rollout

Edge Draws the object from edge to edge. You can change the center location by moving the mouse.

Center Draws the object from the center out.

Parameters rollout



Radius Sets the radius of the chamfered cylinder.

Height Sets the dimension along the central axis. Negative values create the chamfered cylinder below the construction plane.

Fillet Chamfers the top and bottom cap edges of the chamfered cylinder. Higher numbers result in a more refined fillet along the cap edge.

Height Segs Sets the number of divisions along the corresponding axis.

Fillet Segs Sets the number of segments in the filleted edges of the cylinder. Adding fillet segments curves the edges, producing a filleted cylinder.

Sides Sets the number of sides around the chamfered cylinder. Higher numbers shade and render as true circles with Smooth on. Lower numbers create regular polygonal objects with Smooth off.

Cap Segs Sets the number of concentric divisions along the center of the chamfered cylinder's top and bottom

Smooth Blends the faces of the chamfered cylinder, creating a smooth appearance in rendered views.

Slice On Enables the Slice function. Default=off.

When you create a slice and then turn off Slice On, the complete chamfered cylinder reappears. You can use this check box to switch between the two topologies.

Slice From, **Slice To** Sets the number of degrees around the local Z axis from a zero point at the local X axis.

For both settings, positive values move the end of the slice counterclockwise; negative values move it clockwise. Either setting can be made first. When the ends meet, the whole chamfered cylinder reappears.

Generate Mapping Coords Generates coordinates for applying mapped materials to the chamfered cylinder. Default=on.

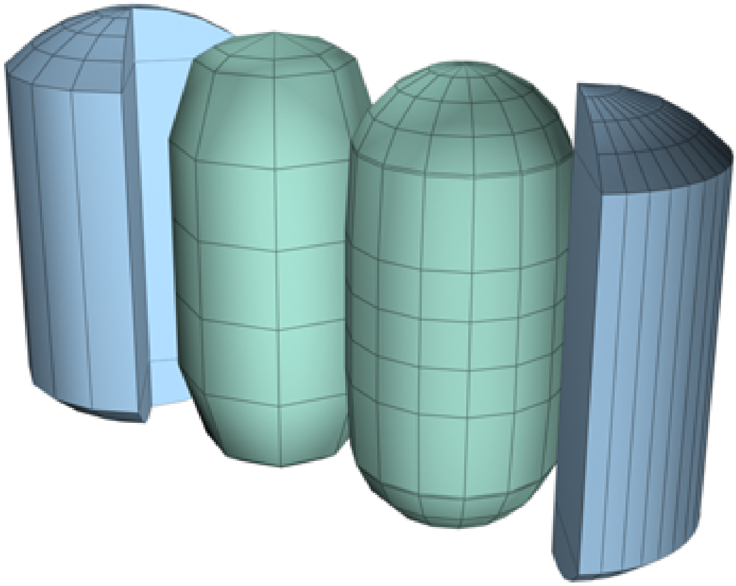
Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

OilTank Extended Primitive

Create panel > Geometry button > Extended Primitives > Object Type rollout > OilTank button

Create menu > Extended Primitives > Oil Tank

Use OilTank to create a cylinder with convex caps.



Examples of oil tanks

Procedures

To create an oil tank:

1. From the Create menu, choose Extended Primitives > Oil Tank.
2. Drag the mouse to define the radius of the base of the oil tank.
3. Release the mouse button, and then move the mouse vertically to define the height of the oil tank. Click to set the height.
4. Move the mouse diagonally to define the height of the convex caps (toward the upper left to increase the height; toward the lower right to decrease it).
5. Click again to finish the oil tank.

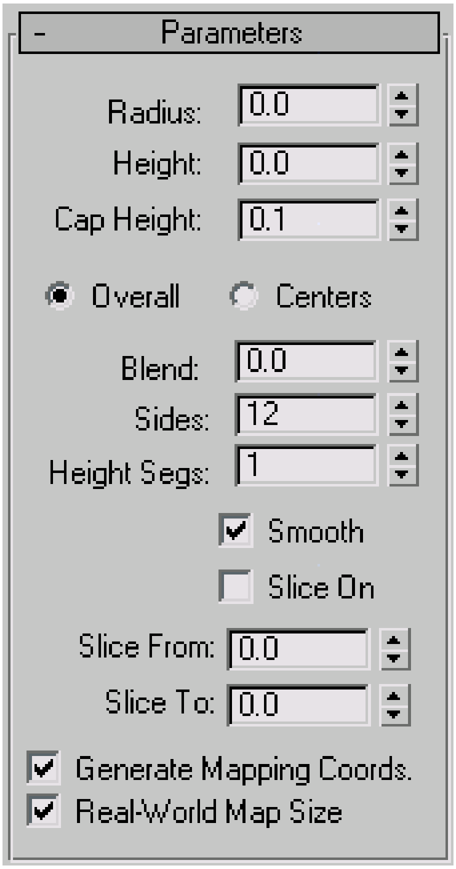
Interface

Creation Method rollout

Edge Draws the object from edge to edge. You can change the center location by moving the mouse.

Center Draws the object from the center out.

Parameters rollout



Radius Sets the radius of the oil tank.

Height Sets the dimension along the central axis. Negative values create the oil tank below the construction plane.

Cap Height Sets the height of the convex caps. The minimum value is 2.5% of the Radius setting. The maximum value is the Radius setting, unless the absolute value of the Height setting is less than the double Radius setting, in which case cap height cannot exceed ½ of the absolute value of the Height setting.

Overall/Centers Determines what the Height value specifies. Overall is the overall height of the object. Centers is the height of the midsection of the cylinder, not including its convex caps.

Blend When greater than 0, creates a bevel at the edge of the caps.

Sides Sets the number of sides around the oil tank. To create a smoothly rounded object, use a higher number of sides and turn Smooth on. To create an oil tank with flat sides, use a lower number of sides and turn Smooth off.

Height Segs Sets the number of divisions along the oil tank's major axis.

Smooth Blends the faces of the oil tank, creating a smooth appearance in rendered views.

Slice On Turns on the Slice function. Default=off.

When you create a slice and then turn off Slice On, the complete oil tank reappears. You can therefore use this check box to switch between the two topologies.

Slice From, Slice To Sets the number of degrees around the local Z axis from a zero point at the local X axis.

For both settings, positive values move the end of the slice counterclockwise; negative values move it clockwise. Either setting can be made first. When the ends meet, the whole oil tank reappears.

Generate Mapping Coords Generates coordinates for applying mapped materials to the oil tank. Default=on.

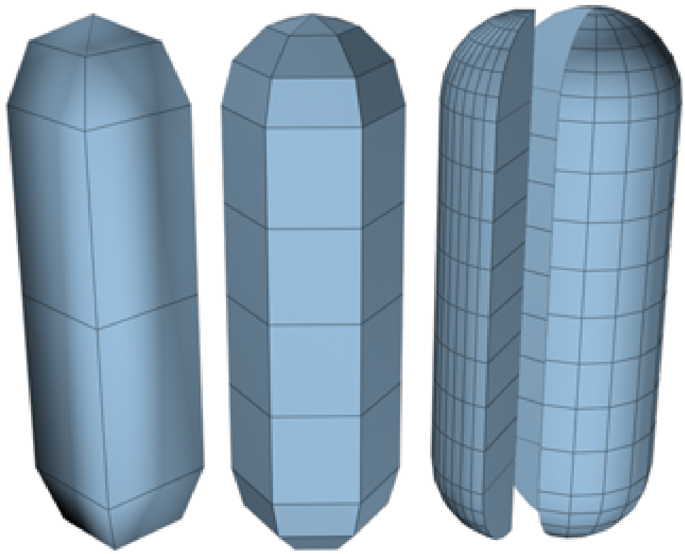
Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Capsule Extended Primitive

Create panel > Geometry button > Extended Primitives > Object Type rollout > Capsule button

Create menu > Extended Primitives > Capsule

Use Capsule to create a cylinder with hemispherical caps.



Examples of capsules

Procedures

To create a capsule:

1. From the Create menu, choose Extended Primitives > Capsule.
2. Drag the mouse to define the radius of the capsule.
3. Release the mouse button, and then move the mouse vertically to define the height of the capsule.
4. Click to set the height and finish the capsule.

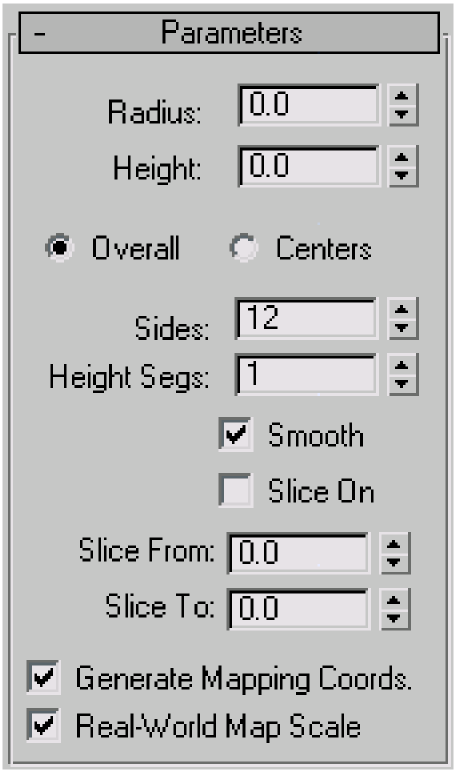
Interface

Creation Method rollout

Edge Draws the object from edge to edge. You can change the center location by moving the mouse.

Center Draws the object from the center out.

Parameters rollout



Radius Sets the radius of the capsule.

Height Sets the height along the central axis. Negative values create the capsule below the construction plane.

Overall/Centers Determines what the Height value specifies. Overall specifies the overall height of the object. Centers specifies the height of the midsection of the cylinder, not including its domed caps.

Sides Sets the number of sides around the capsule. Higher numbers shade and render as true circles with Smooth on. Lower numbers create regular polygonal objects with Smooth off.

Height Segs Sets the number of divisions along the capsule's major axis.

Smooth Blends the faces of the capsule, creating a smooth appearance in rendered views.

Slice On Turns on the Slice function. Default=off.

When you create a slice and then turn off Slice On, the complete capsule reappears. You can use this check box to switch between the two topologies.

Slice From, **Slice To** Sets the number of degrees around the local Z axis from a zero point at the local X axis.

For both settings, positive values move the end of the slice counterclockwise; negative values move it clockwise. Either setting can be made first. When the ends meet, the whole capsule reappears.

Generate Mapping Coords Generates coordinates for applying mapped materials to the capsule. Default=on.

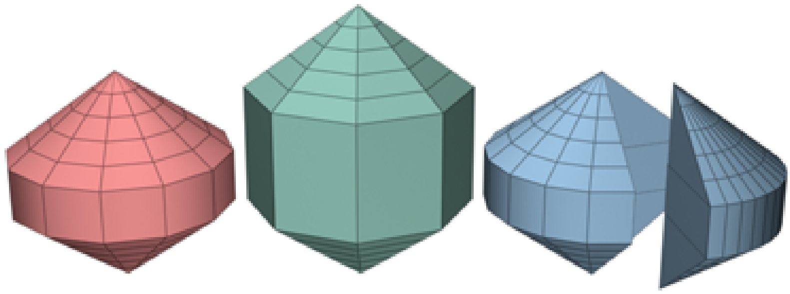
Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Spindle Extended Primitive

Create panel > Geometry button > Extended Primitives > Object Type rollout > Spindle button

Create menu > Extended Primitives > Spindle

Use the Spindle primitive to create a cylinder with conical caps.



Examples of spindles

Procedures

To create a spindle:

1. From the Create menu, choose Extended Primitives > Spindle.
2. Drag the mouse to define the radius of the base of the spindle.
3. Release the mouse button, and then move the mouse vertically to define the height of the spindle. Click to set the height.
4. Move the mouse diagonally to define the height of the conical caps (toward the upper left to increase the height; toward the lower right to decrease it).
5. Click again to finish the spindle.

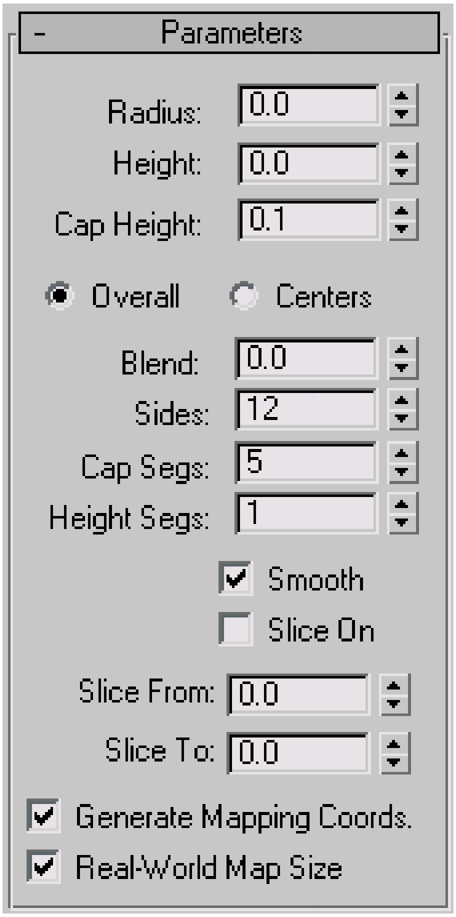
Interface

Creation Method rollout

Edge Draws the object from edge to edge. You can change the center location by moving the mouse.

Center Draws the object from the center out.

Parameters rollout



Radius Sets the radius of the spindle.

Height Sets the dimension along the central axis. Negative values create the spindle below the construction plane.

Cap Height Sets the height of the conical caps. The minimum value is 0.1; the maximum value is ½ the absolute value of the Height setting.

Overall/Centers Determines what the Height value specifies. Overall specifies the overall height of the object. Centers specifies the height of the midsection of the cylinder, not including its conical caps.

Blend When greater than 0, creates a fillet where the caps meet the body of the spindle.

Sides Sets the number of sides around the spindle. Higher numbers shade and render as true circles with Smooth on. Lower numbers create regular polygonal objects with Smooth off.

Cap Segs Sets the number of concentric divisions along the center of the spindle's top and bottom.

Height Segs Sets the number of divisions along the spindle's major axis.

Smooth Blends the faces of the spindle, creating a smooth appearance in rendered views.

Slice On Turns on the Slice function. Default=off.

When you create a slice and then turn off Slice On, the complete spindle reappears. You can therefore use this check box to switch between the two topologies.

Slice From, Slice To Sets the number of degrees around the local Z axis from a zero point at the local X axis.

For both settings, positive values move the end of the slice counterclockwise; negative values move it clockwise. Either setting can be made first. When the ends meet, the whole spindle reappears.

Generate Mapping Coords Sets up the required coordinates for applying mapped materials to the spindle. Default=on.

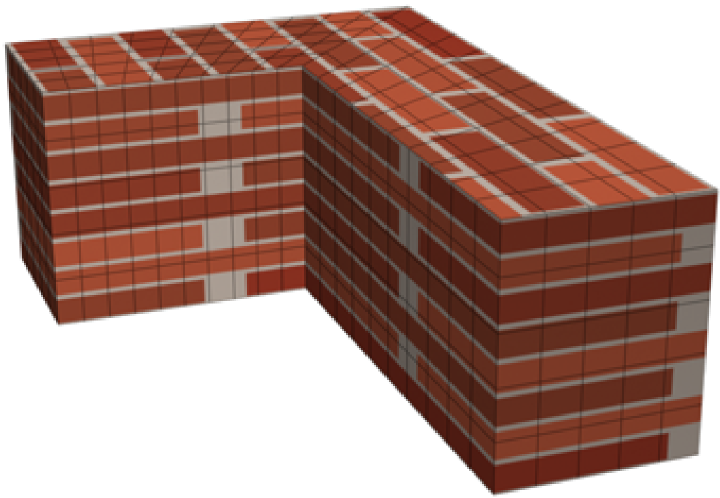
Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

L-Ext Extended Primitive

Create panel > Geometry button > Extended Primitives > Object Type rollout > L-Ext button

Create menu > Extended Primitives > L-Extrustion

Use L-Ext to create an extruded L-shaped object.



Example of L-Ext

Procedures

To create an L-Ext object:

1. From the Create menu, choose Extended Primitives > L-Ext.
2. Drag the mouse to define the base. (Press Ctrl to constrain the base to a square.)
3. Release the mouse and move it vertically to define the height of the L-extrusion.

4 Click, and then move the mouse vertically to define the thickness or width of the walls of the L-extrusion.

5 Click to finish the L-extrusion.

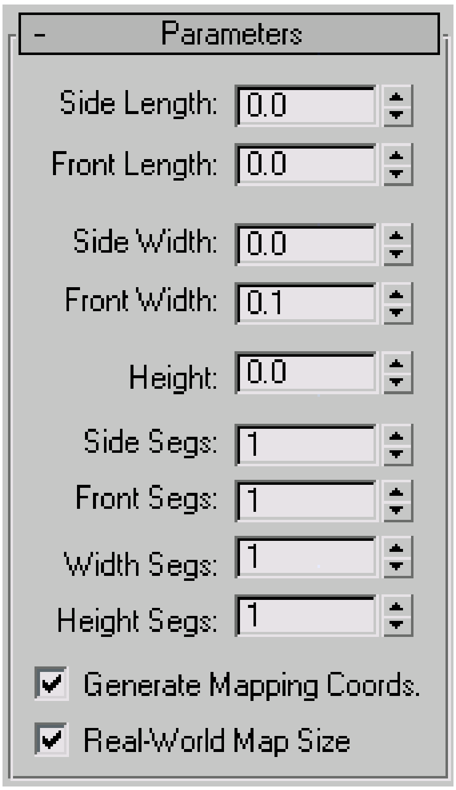
Interface

Creation Method rollout

Corners Draws the object from corner to corner. You can change the center location by moving the mouse.

Center Draws the object from the center out.

Parameters rollout



Side/Front Length Specify the lengths of each "leg" of the L.

Side/Front Width Specify the widths of each "leg" of the L.

Height Specifies the height of the object.

Side/Front Segs Specify the number of segments for a specific "leg" of the object.

Width/Height Segs Specify the number of segments for the overall width and height.

NOTE The object's dimensions (Back, Side, Front) are labeled as though it were created in the Top or Perspective viewports, and seen from the front in world space.

Generate Mapping Coords Sets up the required coordinates for applying mapped materials to the object. Default=on.

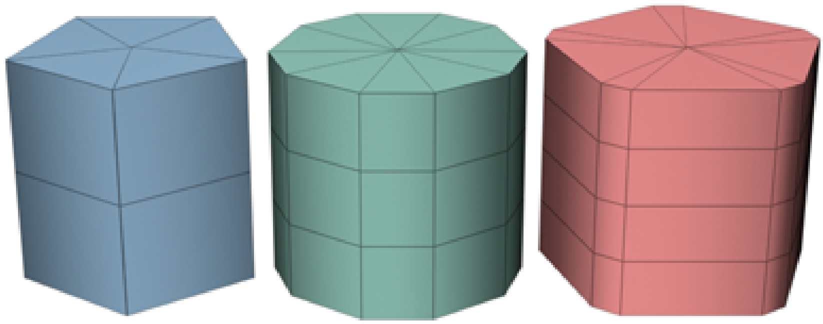
Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Gengon Extended Primitive

Create panel > Geometry button > Extended Primitives > Object Type rollout > Gengon button

Create menu > Extended Primitives > Gengon

Use Gengon to create an extruded, regular-sided polygon with optionally filleted side edges.



Examples of gengons

Procedures

To create a gengon:

1. From the Create menu, choose Extended Primitives > Gengon.
2. Set the Sides spinner to specify the number of side wedges in the gengon.
3. Drag the mouse to create the radius of the gengon.
4. Release the mouse button, then move the mouse vertically to define the height of the gengon. Click to set the height.
5. Move the mouse diagonally to specify the size of the chamfer along the side angles (toward the upper left to increase the size; toward the lower right to decrease it).
6. Click to finish the gengon.

TIP In the Parameters rollout, increase the Fillet Segs spinner to round the chamfered corners into fillets.

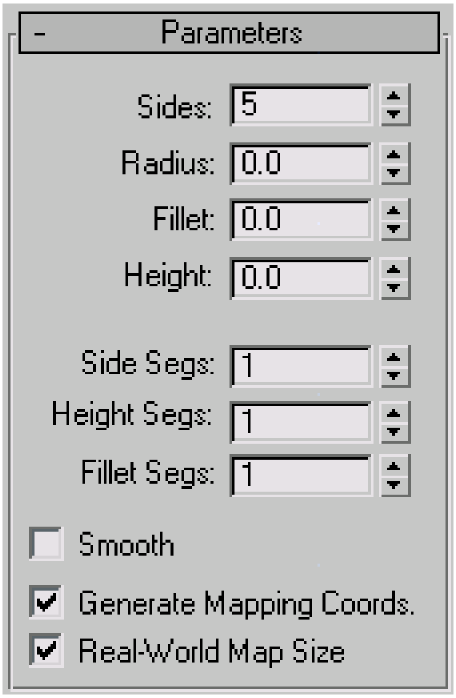
Interface

Creation Method rollout

Edge Draws the object from edge to edge. You can change the center location by moving the mouse.

Center Draws the object from the center out.

Parameters rollout



Sides Sets the number of sides around the gengon. Higher numbers shade and render as true circles with Smooth on. Lower numbers create regular polygonal objects with Smooth off.

Radius Sets the radius of the gengon.

Fillet Sets the width of the chamfered corners.

Height Sets the dimension along the central axis. Negative values create the gengon below the construction plane.

Side Segs Sets the number of divisions around the gengon.

Height Segs Sets the number of divisions along the gengon's major axis.

Fillet Segs Sets the number of divisions for the edge filleting. Increasing this setting will produce round, filleted corners instead of chamfers.

Smooth Blends the faces of the gengon, creating a smooth appearance in rendered views.

Generate Mapping Coords Sets up the required coordinates for applying mapped materials to the gengon. Default=on.

Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

C-Ext Extended Primitive

Create panel > Geometry > Extended Primitives > Object Type rollout > C-Ext button

Create menu > Extended Primitives > C-Extrusion

Use C-Ext to create an extruded C-shaped object.

Example of C-Ext

Procedures

To create a C-Ext object:

1. From the Create menu, choose Extended Primitives > C-Extrusion.
2. Drag the mouse to define the base. (Press Ctrl to constrain the base to a square.)
3. Release the mouse and move it vertically to define the height of the C-extrusion.
4. Click, and then move the mouse vertically to define the thickness or width of the walls of the C-extrusion.
5. Click to finish the C-extrusion.

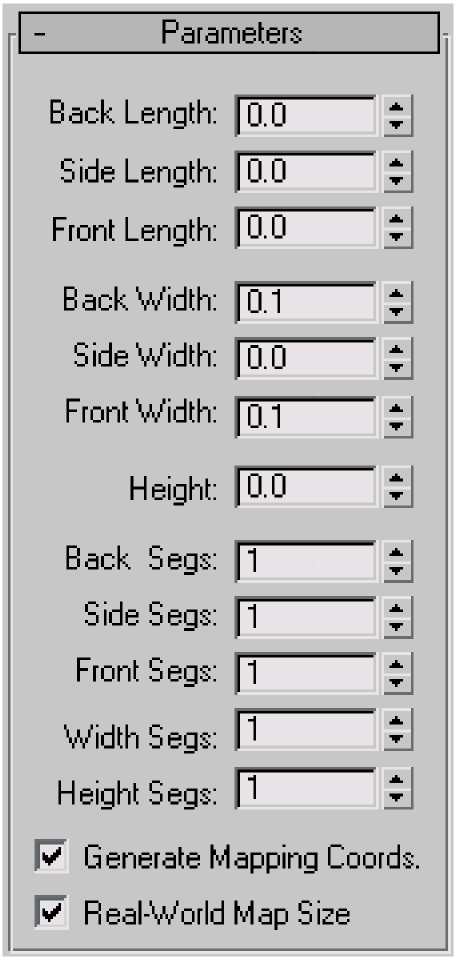
Interface

Creation Method rollout

Corners Draws the object from corner to corner. You can change the center location by moving the mouse.

Center Draws the object from the center out.

Parameters rollout



Back/Side/Front Length Specify the length of each of the three sides.

Back/Side/Front Width Specify the width of each of the three sides.

Height Specifies the overall height of the of the object.

Back/Side/Front Segs Specify the number of segments for a specific side of the object.

NOTE The object's dimensions (Back, Side, Front) are labeled as though it were created in the Top or Perspective viewports, and seen from the front in world space.

Width/Height Segs Set these to specify the number of segments for the overall width and height of the object.

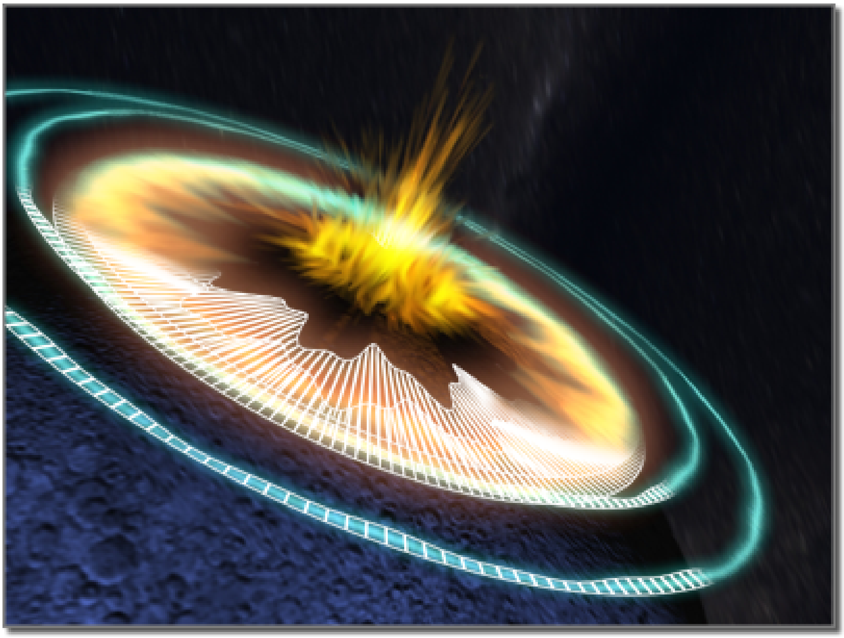
Generate Mapping Coords Sets up the required coordinates for applying mapped materials to the object. Default=on.

Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

RingWave Extended Primitive

Create panel > Geometry button > Extended Primitives > Object Type rollout > RingWave button

Create menu > Extended Primitives > RingWave



Example of ringwave

Use the RingWave object to create a ring, optionally with irregular inner and outer edges whose shapes can be animated. You can also animate the growth of the ringwave object, and you can use keyframing to animate all numeric settings. Use RingWave for various types of special-effects animation, for example, to depict the shock wave emanating from the explosion of a star or planet.

Procedures

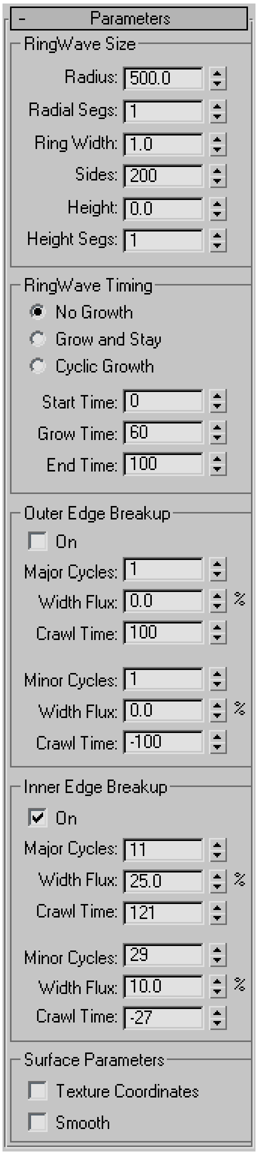
To create a basic animated ringwave:

1. On the menu bar choose Create > Extended Primitives > Ringwave.
2. Drag in a viewport to set the outer radius of the ringwave.
3. Release the mouse button, and then move the mouse back toward the center of the ring to set the inner radius.
4. Click to create the ringwave object.
5. Drag the time slider to see the basic animation. This is determined by the Inner Edge Breakup group > Crawl Time settings.

6 To animate the ring's growth, choose RingWave Timing group > Grow And Stay or Cyclic Growth.

Interface

Parameters rollout



RingWave Size group

Use these settings to change the ringwave's basic parameters.

Radius Sets the outside radius of the ringwave.

Radial Segs Sets the segment count between the inner and outer surfaces in the direction of the radius.

Ring Width Sets the mean ring width as measured inward from the outer radius.

Sides Sets the number of segments in the circumferential direction for both the inner, outer, and end (cap) surfaces.

Height Sets the height of the ringwave along its major axis.

TIP If you leave the Height at 0 for an effect like a shock wave, you will want to apply a two-sided material so that the ring can be seen from both sides.

Height Segs Sets the number of segments in the direction of the height.

RingWave Timing group

Use these settings for ringwave animation where the ringwave grows from nothing to its full size.

No Growth Sets a static ringwave, which appears at Start Time and disappears after End Time.

Grow and Stay Animates a single growth cycle. The ringwave begins growing at the Start Time and reaches its full size at Start Time plus Grow Time.

Cyclic Growth The ringwave grows repeatedly from the Start Time to Start Time plus Grow Time.

For example, if you set Start Time to 0 and Grow Time to 25, leaving End Time at the default value of 100, and choose Cyclic Growth, the ringwave grows from nothing to its full size four times over the course of the animation.

Start Time The frame number where the ringwave appears, and begins to grow if you choose Grow and Stay or Cyclic Growth.

Grow Time The number of frames after Start Time the ringwave takes to reach full size. Grow Time is available only if Grow and Stay or Cyclic Growth is chosen.

End Time The frame number after which the ringwave disappears.

Outer Edge Breakup group

Use these settings to change the shape of the ringwave's outer edge.

TIP For effects like shock waves, the ringwave typically has little or no breakup on the outer edge but relatively massive breakup on the inner edge.

On Turns on breakup of the outer edge. The remaining parameters in this group are active only when this is on. Default=off.

Major Cycles Sets the number of major waves around the outer edge.

Width Flux Sets the size of the major waves, expressed as a percentage of the unmodulated width.

Crawl Time Sets the number of frames each major wave takes to move around the outer circumference of the RingWave.

Minor Cycles Sets the number of random-sized smaller waves in each major cycle.

Width Flux Sets the average size of the smaller waves, expressed as a percentage of the unmodulated width.

Crawl Time Sets the number of frames each minor wave takes to move across its respective major wave.

Inner Edge Breakup group

Use these settings to change the shape of the ringwave's inner edge.

On Turns on the breakup of the inner edge. The remaining parameters in this group are active only when this is on. Default=on.

Major Cycles Sets the number of major waves around the inner edge.

Width Flux Sets the size of the major waves, expressed as a percentage of the unmodulated width.

Crawl Time Sets the number of frames each major wave takes to move around the inner circumference of the RingWave.

Minor Cycles Sets the number of random-sized smaller waves in each major cycle.

Width Flux Sets the average size of the smaller waves, expressed as a percentage of the unmodulated width.

Crawl Time Sets the number of frames each minor wave takes to move across its respective major wave.

NOTE Negative values in the Crawl Time parameters change the direction of the wave. To produce interference patterns, use Crawl Time settings of opposite sign for major and minor waves, but similar Width Flux and Cycles settings.

TIP To produce the best "random" results, use prime numbers for major and minor cycles that differ by a multiple of two to four. For example, a major wave of 11 or 17 cycles using a width flux of 50 combined with a minor wave of 23 or 31 cycles with a width flux of 10 to 20 makes a nice random-appearing edge.

Texture Coordinates Sets up the required coordinates for applying mapped materials to the object. Default=on.

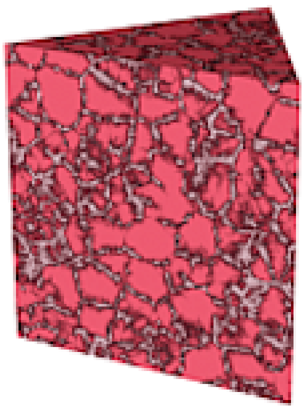
Smooth Applies smoothing to the object by setting all polygons to smoothing group 1. Default=on.

Prism Extended Primitive

Create panel > Geometry button > Extended Primitives > Object Type rollout > Prism button

Create menu > Extended Primitives > Prism

Use Prism to create a three-sided prism with independently segmented sides.



Example of a prism

Procedures

To create a prism with an isosceles triangle as its base:

1. Choose Isosceles on the Creation Method rollout.
2. Drag horizontally in the viewport to define the length of Side 1 (along the X axis). Drag vertically to define the length of Sides 2 and 3 (along the Y axis).

   (To constrain the base to an equilateral triangle, press Ctrl before performing this step.)
3. Release the mouse, and move it vertically to define the height of the prism.
4. Click to complete the prism.
5. On the Parameters rollout, alter the length of the sides as needed.

To create a prism with a scalene or obtuse triangle at its base:

1. Choose Base/Apex in the Creation Method rollout.
2. Drag horizontally in the viewport to define the length of Side 1 (along the X axis). Drag vertically to define the length of Sides 2 and 3 (along the Y axis).
3. Click, and then move the mouse to specify the placement of the apex of the triangle. This alters the length of sides 2 and 3, and the angles of the corners of the triangle.
4. Click, and then move the mouse vertically to define the height of the prism.
5. Click to complete the prism.

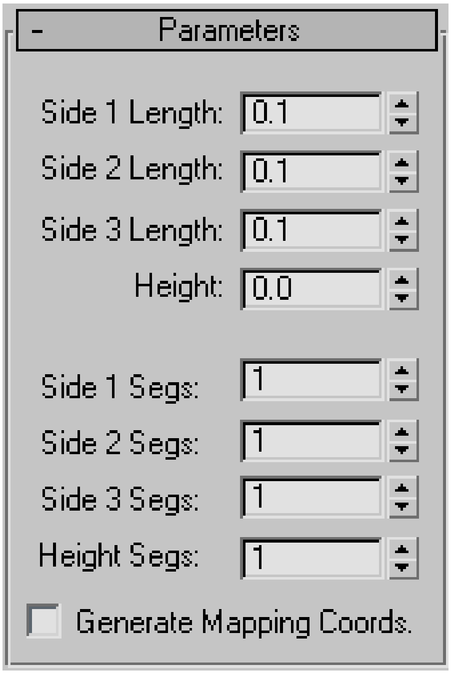
Interface

Creation Method rollout

Isosceles Draws a prism with an isosceles triangle at its base.

Base/Apex Draws a prism with a scalene or obtuse triangle at its base.

Parameters rollout



Side (n) Length Sets the length of triangle's corresponding side (and thus the triangle's corner angles).

Height Sets the dimension of the prism's central axis.

Side (n) Segs Specifies the number of segments for each side of the prism.

Height Segs Sets the number of divisions along the prism's central axis.

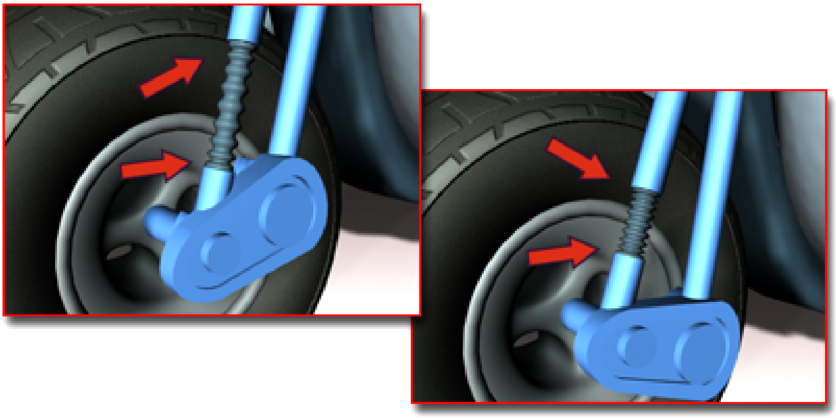
Generate Mapping Coordinates Sets up the required coordinates for applying mapped materials to the prism. Default=off.

Hose Extended Primitive

Create panel > Geometry > Extended Primitives > Object Type rollout > Hose button

Create menu > Extended Primitives > Hose

The Hose object is a flexible object that you can connect between two objects, whereupon it reacts to their movement. It's similar to Spring on page 890, but does not have dynamics properties. You can specify the overall diameter and length of the hose, the number of turns, and the diameter and shape of its "wire."



Hose models a workable spring on a motorcycle

Procedures

To create a hose:

1 From the menu bar, choose Create > Extended Primitives > Hose.

2 Drag the mouse to define the radius of the hose.

3 Release the mouse, and then move it to define the length of the hose.

4 Click to finish the hose.

To bind a hose to two objects:

1 Add a hose and two other objects. Select the hose.

2 In the Modify panel > Hose Parameters rollout > End Point Method group, choose Bound To Object Pivots.

3 In the Binding Objects group, click Pick Top Object, and then select one of the two objects.

4 In the Binding Objects group, click Pick Bottom Object, and then select the second of the two objects.

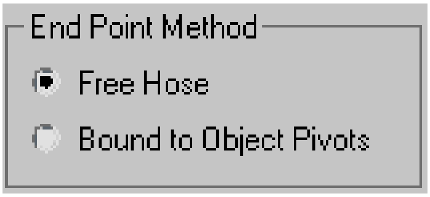
The two ends of the hose attach themselves to the two objects.

5 Move one of the objects.

The hose adjusts itself to remain attached to both objects.

Interface

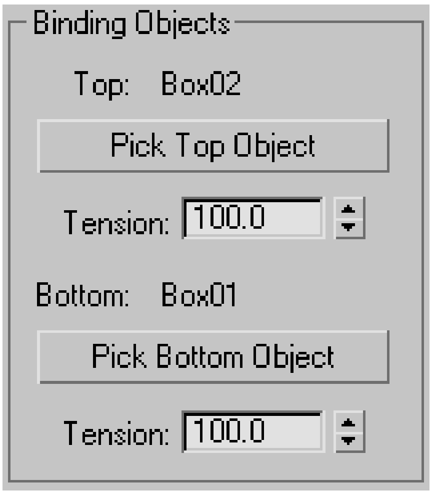
Hose Parameters rollout > End Point Method group



Free Hose Choose this when using the hose as a simple object that's not bound to other objects.

Bound to Object Pivots Choose this when binding the hose to two objects, using the buttons in the Binding Objects group.

Hose Parameters rollout > Binding Objects group



Available only when Bound To Object Pivots is chosen. Use the controls to pick the objects to which the hose is bound and to set the tension between them. "Top" and "Bottom" are arbitrary descriptors; the two bound objects can have any positional relationship to each other.

Each end point of the hose is defined by the center of the overall diameter. This end point is placed at the pivot point of the object to which it is bound. You can adjust the relative position of the binding object to the hose by transforming the binding object while the Affect Object Only button is turned on in the Hierarchy panel > Adjust Pivot rollout.

Top (label) Displays the name of the "top" binding object.

Pick Top Object Click this button and then select the "top" object.

Tension Determines the tension of the hose curve near the Top object as it reaches for the Bottom object. Lower the tension to have the bend occur closer to the Top object, raise the tension to have the bend occur further away from the Top object. Default=100.

Bottom (label) Displays the name of the "bottom" binding object.

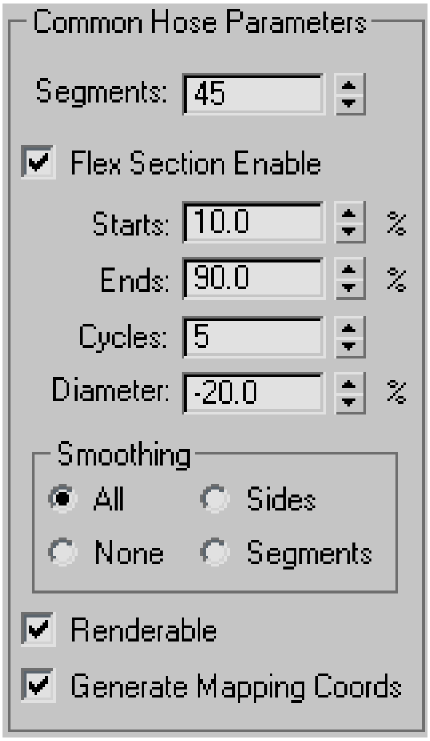
Pick Bottom Object Click this button and then select the "bottom" object.

Tension Determines the tension of the hose curve near the Bottom object as it reaches for the Top object. Lower the tension to have the bend occur closer to the Bottom object, raise the tension to have the bend occur further away from the Bottom object. Default=100.

Hose Parameters rollout > Free Hose Parameters group

Height Use this field to set the straight-line height or length of the hose when it is not bound. This is not necessarily the actual length of the hose. Available only when Free Hose is chosen.

Hose Parameters rollout > Common Hose Parameters group



Segments The total number of segments in the hose's length. Increase this setting for a smooth profile when the hose is curved. Default=45.

Flex Section Enable When on, lets you set the following four parameters for the central, flexible section of the hose. When off, the hose's diameter is uniform throughout its length.

Starts The percentage of the hose length from the starting extremity of the hose at which the flex section begins. By default, the starting end of the hose is the end at which the object pivot appears. Default=10%.

Ends The percentage of the hose length from the end extremity of the hose at which the flex section ends. By default, the end extremity of the hose is opposite the end at which the object pivot appears. Default=90%.

Cycles The number of corrugations in the flex section. The number of visible cycles is limited by the number of segments; if Segments isn't high enough to support the number of cycles, then not all cycles will appear. Default=5.

TIP To set the appropriate number of segments, first set Cycles, and then increase Segments until the number of visible cycles stops changing.

Diameter The relative width of the "outside" parts of the cycles. At negative settings, these are smaller than the overall hose diameter. At positive settings, these are larger than the overall hose diameter. Default=-20%. Range=-50% to 500%.

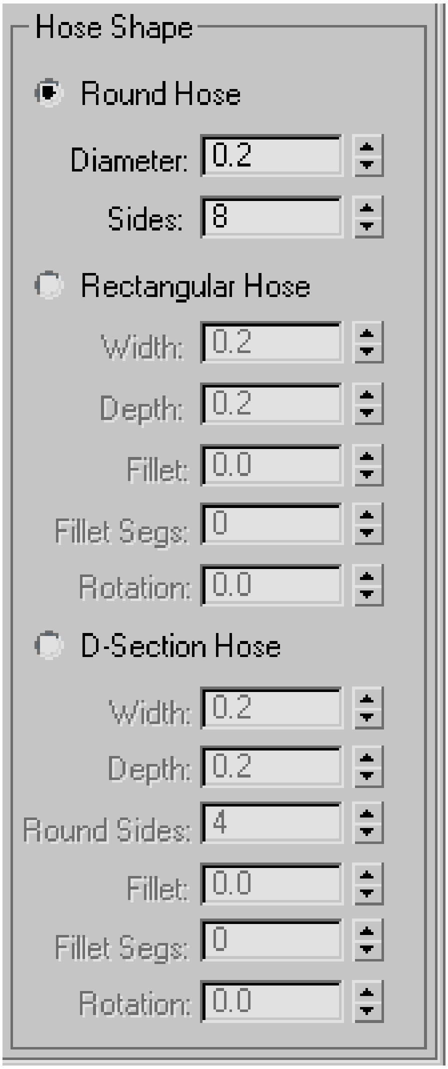
Smoothing Defines the geometry that gets smoothed. Default=All:

- **All** The entire hose is smoothed.
- **Sides** Smoothing is applied along the length of the hose but not around its circumference.
- **None** No smoothing is applied.
- **Segments** Smoothing is applied only on the inner section of the hose.

Renderable When on, the hose is rendered using the specified settings. When off, the hose is not rendered. Default=on.

Generate Mapping Coords Sets up required coordinates for applying mapped materials to the hose. Default=on.

Hose Parameters rollout > Hose Shape group



Sets the shape of the hose cross section. Default=Round Hose.

Round Hose Sets a circular cross section.

Diameter The maximum width of the hose at the ends.

Sides The number of sides of the hose. A Sides setting of 3 gives a triangular cross section; 4 gives a square cross section; and 5 gives a pentagonal cross section. Increase Sides for a circular cross section. Default=8.

Rectangular Hose Lets you specify different settings for width and depth.

Width The width of the hose.

Depth The height of the hose.

Fillet The amount by which the cross-section corners are rounded. For this to be visible, Fillet Segs must be set to 1 or higher. Default=0.

Fillet Segs The number of segments across each filleted corner. A Fillet Segs setting of 1 cuts the corner straight across; use higher settings for rounded corners. Default=0.

Rotation The orientation of the hose along its long axis. Default=0.

D-Section Hose Similar to Rectangular Hose, but rounds one side for a D-shaped cross-section.

Width The width of the hose.

Depth The height of the hose.

Round Sides The number of segments on the rounded side. Increase for a smoother profile. Default=4.

Fillet The amount by which the two cross-section corners opposite the rounded side are rounded. For this to be visible, Fillet Segs must be set to 1 or higher. Default=0.

Fillet Segs The number of segments across each filleted corner. A Fillet Segs setting of 1 cuts the corner straight across; use higher settings for rounded corners. Default=0.

Rotation The orientation of the hose along its long axis. Default=0.

Architectural Objects

3ds Max provides an array of architectural objects, useful as building blocks for models of homes, businesses, and similar projects. These include:

AEC Extended Objects on page 493: Foliage, Railing, and Wall

Stairs on page 532

[Doors](#) on page 562

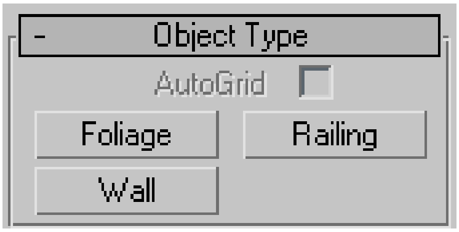
[Windows](#) on page 577

AEC Extended Objects

Create panel > Geometry > AEC Extended

Create menu > AEC Objects

AEC Extended objects are designed for use in the architectural, engineering, and construction fields. Use Foliage to create plants, Railing to create railings and fences, and Wall to create walls.



Interface

Foliage on page 499

Railing on page 507

Wall on page 516

The Object Name and Wireframe Color rollout on page 7631 in each AEC Extended object's creation panel functions identically. The remaining rollouts are covered in each object's topic.

Working with AEC Design Elements

3ds Max includes such features as Foliage, Doors, Windows, Stairs, Railing, and Wall to make exploring three-dimensional design ideas much easier.

This section provides general information about these features. For detailed explanations and procedures, see the topics listed below:

Doors on page 562

Windows on page 577

Stairs on page 532

Railing on page 507

Wall on page 516

Foliage on page 499

Doors and Windows

3ds Max supplies a number of parametric window and door objects that you can place into wall openings to add realism to an architectural model. These objects let you control details like trim and panel fill in your model.

TIP Use Snaps on page 2671 for added precision when adding doors and windows.

When you create a new door or window, you must select four points in the scene that define the size and orientation of the rectangle that will be the door or window. You may find it easier to select these points in a given sequence, depending on your scene and views of the scene.

If you already have a rectangular hole you want to fill, you can still create a door or window to your specifications by using the following procedure.

To create a door or window:

1. Set up an angled User view so that you can see the bottom and one vertical edge of the opening and its full height.
2. Set the appropriate object snaps, such as Vertex or Endpoint. This helps make the model more precise.
3. After clicking Window or Door, choose one of two Creation Methods: Width/Depth/Height or Width/Height/Depth.
4. Make parameter adjustments to define details.

The width and orientation of the door/window is always defined by the first mouse click and subsequent mouse drag. Depending on the creation method you use, either the height or depth of the object is defined next.

If you have no object snaps set and are working in a Perspective or User Viewport, using the Width/Depth/Height Creation Method creates an upright Door or Window. The Width/Height/Depth Creation Method creates the object as if it were lying on its side.

Allowing Non-vertical Jambs

The Allow Non-vertical Jambs toggle is useful for creating doors or windows that do not fit in a vertical plane, such as a skylight window in a sloping roof. By default, this toggle is off, making the third point in the creation sequence either directly above (Width/Height/Depth) or on the same horizontal plane (Width/Depth/Height) with the second point.

When you turn on Allow Non-vertical Jambs, the third point in the creation sequence falls wherever you choose and the fourth point is added by the program. Its offset from the plane is determined by the first three points.

Using the Width/Height/Depth Creation Method in Perspective and User viewports with Allow Non-vertical Jambs off can be an efficient way to create doors and windows with Object Snaps. However, it can also be confusing at first. Keep in mind that the third point you define, the Height, is interpreted as a point on the home grid until you indicate a point higher or lower than the grid. If you are using an Object Snap setting, 3ds Max might not know you mean a point off the grid unless you bring the cursor in proximity to a nonplanar point to which it can snap.

Additional Parameters

There are additional parameters specific to each door and window type that control overall dimension parameters, as well as detailed parameters for sub-object components such as mullions, trim, and panels within leaves. See Doors on page 562 and Windows on page 577 for more information on these parameters.

Animating Doors and Windows

Certain door and window creation parameters, including the Open parameter, can be animated. See Doors on page 562 and Windows on page 577 for more information.

Creating Stairs and Railings

3ds Max contains four types of stair objects: spiral stairs on page 540, U-type stairs on page 555 with an intermediate landing, L-type stairs on page 534 with a landing at the bend in the stair, and straight stairs on page 549 with no intermediate landing. A complementary Railing object can be used to create any number of handrail designs that follow along a spline path.

For more information, see Stairs on page 532.

The Railing Object

Use the Railing button on the Create panel in the to produce railing objects. Railing components include rails, AEC Extended category on page 493posts, and fencing. Fencing includes pickets (balusters) or solid-filled material (such as glass or wood strips).

You can create a railing in two ways: specify the orientation and height of the railing, or pick a spline path and apply the railing to that path. The spline path with a railing is called a rail path. Later, if you edit the rail path, the Railing object automatically updates to follow the changes you make. Rail paths can occupy three-dimensional space.

When you create the lower rails, posts, and fencing components of a Railing object, you use a special version of the Spacing Tool to specify the spacing of those components. The program displays the Spacing Tool dialog for each railing component: Lower Rail, Post Spacing, or Picket Spacing. For more information on the Spacing Tool, see Spacing Tool on page 996.

For details on Railing parameters and information on creating a Railing object, see Railing on page 507.

Creating Walls

Use the Wall button on page 516 on the Create panel, in the AEC Extended category, to produce straight-wall objects. A wall object is made up of sub-object segments that you can edit with the Modify panel.

You can:

- Break or insert wall segments to create separate wall objects.
- Delete wall segments.
- Connect two wall objects.

When you create two wall segments that meet at a corner, 3ds Max removes any duplicate geometry. This “cleaning up” of the corners might involve trimming. 3ds Max cleans up only the first two wall segments of a corner, not other wall segments that might share the corner. 3ds Max does not clean up intersections.

You can edit the segments of a wall using sub-object selection mode on the Modify panel. For example, you can define a wall’s height profile. 3ds Max moves the active grid to the plane of the wall you’re editing. This allows you to snap to the profile vertices in the plane of the wall.

If you move, scale, or rotate the wall object, the linked door and window moves, scales, or rotates along with the wall. If you move the linked door or window along the wall, using the door or window's Local coordinate system and activating Restrict to XY Plane in the Axis Constraints toolbar on page 955, the opening will follow. Also, if you change a door or window's overall width and height in the Modify panel, the hole will reflect those changes.

Usage Tips

The following are a few tips for working with wall objects:

- Use the Top viewport when creating wall objects.
- Single walls with many windows and doors can slow down snap calculations and movement of the wall object. To speed up insertion and editing, use multiple walls instead of a single wall.
- You can speed up performance in a scene with many walls, windows, and doors by collapsing them. First save an uncollapsed version for any future parametric changes you might want to make. Then right-click the wall and pick Select Children from the right-click menu. Next use Collapse in the Utility rollout to collapse them all.

For complete information, see Wall on page 516.

To create a wall:

1. On the Create panel, in the AEC Extended category, click Wall.
2. Use Customize > Units Setup to establish precision, and then set the parameters for the Width, Height, and Justification of the wall.
3. In any viewport, click, release the mouse, drag the wall segment to the length you want and click again.

   This creates a wall segment. You can end the wall or you can continue to create another wall segment.
4. To complete the wall, right-click, or to add another wall segment, drag the next wall segment to the length you want and click again.

   If you create a room by ending a segment at the end of another segment of the same wall object, the program displays the Weld Point dialog. This dialog lets you convert the two end vertices into a single vertex, or keep the two end vertices separate.

5 If you want the wall segments to be welded at a corner (when you move one wall, the other wall stays at the corner), click Yes. Otherwise, click No.

6 Right-click to complete the wall, or continue to add another wall segment.

To attach separate walls:

1 Select a wall object.

2 On the Modify panel, click Attach, and then pick another wall object.

  The two wall objects become part of the same wall object, but are not physically connected.

  Attach stays active, and you can continue clicking wall segments to attach. To stop attaching, click the Attach button or right-click in the active viewport.

  To attach multiple wall objects simultaneously to the selected wall object, click Attach Multiple on the Modify panel to open the Attach Multiple dialog. This works the same as the Select From Scene dialog on page 228, except that it shows only wall objects; choose multiple walls to attach, and then click the Attach button.

To connect vertices in a wall:

This method lets you connect two separate wall sections with a new segment.

TIP It is easier to work with wall vertices in wireframe view mode.

---

1 Select a wall object that has more than one section. Typically you would use Attach to create such an object.

2 In the modifier stack on page 7635, go to the Vertex sub-object level.

3 Click Connect and point the mouse over an end vertex until the cursor changes to a cross.

4 Click once over the end vertex.

5 Move the cursor to another end vertex, and then click to connect the two segments.

To insert a vertex in a wall:

It is easier to work with wall vertices in wireframe view mode.

1 Select a wall segment.

2 In the modifier stack on page 7635, go to the Vertex sub-object level.

3 Click Insert.

   A highlighted line appears along the bottom of the wall, showing where you can insert vertices.

4 Click anywhere on the highlighted line to insert a vertex.

   The new vertex is attached to the mouse cursor.

5 Move the mouse to position the vertex, and then click to place it.

   Now the mouse is attached to one of the new segments.

6 Move the mouse along the segment and click to add vertices.

7 Right-click to finish working on this segment. You can now insert vertices in other segments, or right-click again to exit Insert mode.

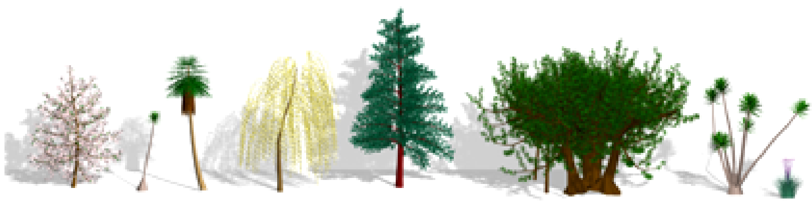
Foliage

Create panel > Geometry > AEC Extended > Foliage button

Create menu > AEC Objects > Foliage

Foliage produces various types of plant objects such tree species. 3ds Max generates mesh representations to create fast, efficient, and good-looking plants.

You control height, density, pruning, seed, canopy display, and level of detail. The seed on page 505 option controls creation of different representations of the same species. You can create millions of variations of the same species, so each object can be unique. With the viewport canopy mode on page 506 option, you can control the amount of plant detail, reducing the number of vertices and faces 3ds Max uses to display the plant.

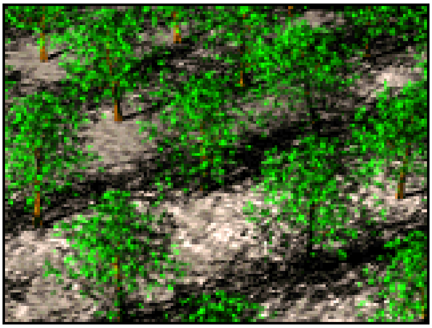


Some of the plants that can be created from the standard library

Tips

- Use the Spacing tool on page 996 to place plants along a path.

- Use vertex or face snapping (see Snaps Settings on page 2662) to position plants on a surface.



Using the Spacing tool to distribute trees along paths

Procedures

To add plants to a scene:

1. Click the Favorite Plants rollout > Plant Library button to display the Configure Palette dialog.
2. Double-click the row for each plant you want to add or remove from the Palette and click OK.
3. On the Favorite Plants rollout, select a plant and drag it to a location in a viewport. Alternatively, select a plant in the rollout and then click in the viewport to place the plant.
4. On the Parameters rollout, click the New button to display different seed variations of the plant.
5. Adjust the remaining parameters to show elements of the plants, such as leaves, fruit, branches, and if you want, to view the plant in canopy mode.

Interface

Object Name and Wireframe Color rollout

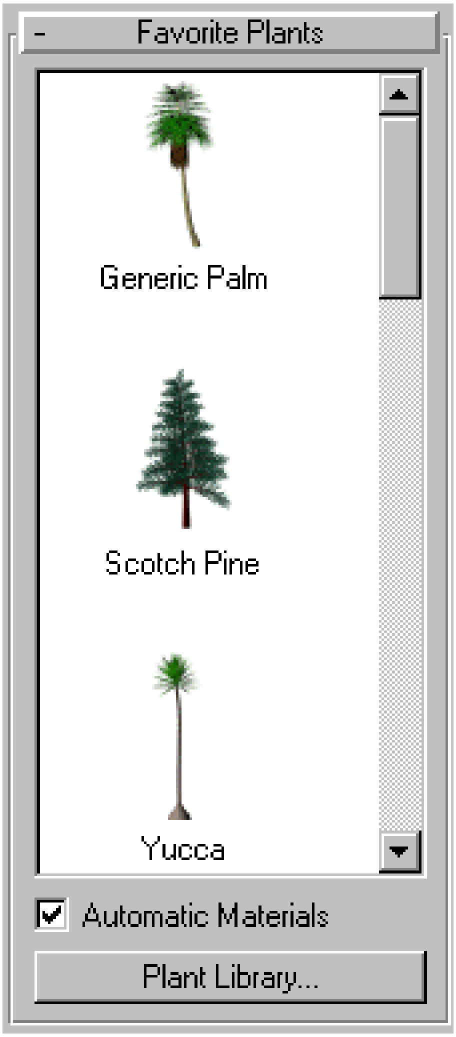
This rollout lets you set the foliage object's name, color, and default material. For detailed information, see Object Name and Wireframe Color on page 7631.

When Favorite Plants rollout > Automatic Materials is on, each plant is assigned its own default material. For more information, see Favorite Plants rollout, following.

Keyboard Entry rollout

See Creating Primitives from the Keyboard on page 406.

Favorite Plants rollout



The palette displays the plants currently loaded from the Plant Library on page 503. There are three ways to add a plant to the scene:

- Use keyboard entry.
- Click the icon in the Favorite Plants list and then click a location in a viewport. Double-click the icon to place the plant at the world origin.
- Drag the plant from the palette and drop it into a viewport.

Automatic Materials Assigns default materials for the plant. To modify these material assignments, use the Material Editor on page 5284. Select the plant in the viewport, and click Main toolbar > Material Editor. Click the Get Material button on page 5341 to display the Material/Map Browser. Under Browse From, choose Selected. Then, from the list pane, double-click the material list item for the plant to display the materials in the Basic Parameters rollout of the Material Editor.

If you turn off Automatic Materials, 3ds Max assigns no materials to the object, unless the Name And Color rollout > Default Material check box is on and a default material is assigned. This way you can specify a particular default material for *all* foliage objects. For more information, see Object Name and Wireframe Color on page 7631.

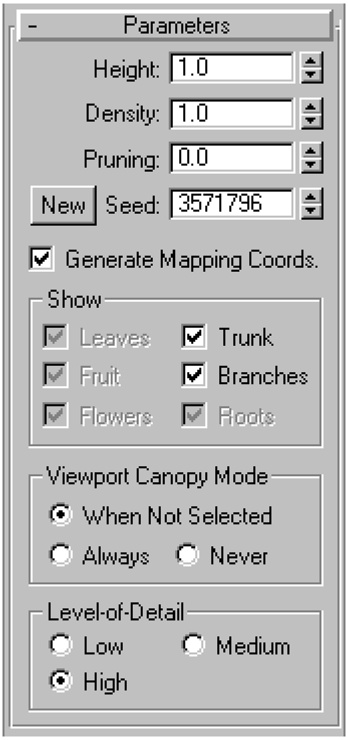
When on, Automatic Materials overrides the Default Material settings.

NOTE Even if Automatic Materials is off, 3ds Max still assigns material IDs to the foliage objects, so that the object is ready for a multi/sub-object material.

Plant Library Displays the Configure Palette dialog. Using this window, you can view information on the available plants including their names, whether they're in the palette, their scientific names, types, descriptions, and the approximate number of faces per object. You can also add and remove plants from the palette, and clear the palette, which removes all plants from the palette.

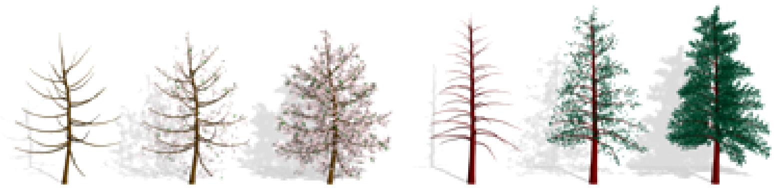
TIP To quickly add or remove a plant from the palette, double-click its row in the Configure Palette dialog. The Fav. (Favorite Plants) column entry switches between "no" and "yes." Click OK to accept the changes and exit the window.

Parameters rollout



Height Controls the approximate height of the plant. 3ds Max applies a random noise factor to the height of all of the plants. Therefore, the actual height of a plant, as measured in the viewports, won't necessarily match the setting given in the Height parameter.

Density Controls the amount of leaves and flowers on the plant. A value of 1 displays a plant with all its leaves and flowers, .5 displays a plant with half its leaves and flowers, and 0 displays a plant with no leaves or flowers.



Two trees with varying foliage densities

Pruning Applies only to plants with branches. Removes branches that lie below an invisible plane parallel to the construction plane. A value of 0 prunes nothing, a value of .5 prunes the plant at a plane halfway up its height from the construction plane, and a value of 1 prunes everything possible from the plant. What 3ds Max prunes from the plant depends on the type of plant. The trunk is never pruned.

Three pairs of trees, showing different values of pruning

New Displays a random variation of the current plant. 3ds Max displays the seed value in the numeric field next to the button.

TIP Click the New button repeatedly until you find the variation you want. This is often easier than trying to adjust the tree using modifiers.

Seed A value between 0 and 16,777,215 representing the possible variations of branch and leaf placement and shape and angle of the trunk of the current plant.

Generate Mapping Coords Applies default mapping coordinates on page 8034 to the plant. Default=on.

Show group

Controls the display of leaves, fruit, flowers, trunk, branches, and roots of plants. Available options depend on the type of plant you select. For example, if a plant doesn't have fruit, 3ds Max disables that option. Turning off options reduces the number of vertices and faces displayed.

Viewport Canopy Mode group

In 3ds Max, the canopy of a plant is a shell covering the outermost parts of the plant, such as the leaves or the tips of the branches and trunk. The term derives from "forest canopy." Use reasonable parameters when you create many plants and want to optimize display performance.

Because this setting applies only to the plant's representation in the viewports, it has no effect on how 3ds Max renders the plant. For information on how 3ds Max renders the plant, see Level-of-Detail on page 506.

When Not Selected Displays the plant in canopy mode when it's not selected.

Always Always displays the plant in canopy mode.

Never Never displays the plant in canopy mode. 3ds Max displays all the features of the plant.

Level-of-Detail group

Controls how 3ds Max renders the plant. For information on how 3ds Max displays the plant in the viewports, see Viewport Canopy Mode on page 506.

Low Renders the plant canopy, providing the lowest level of detail.

Medium Renders a reduced-face-count version of the plant. How 3ds Max reduces the face count varies from plant to plant, but it usually involves removing smaller elements of the plant or reducing the number of faces in the branches and trunk.

High Renders all the faces of the plant, providing the highest level of detail.

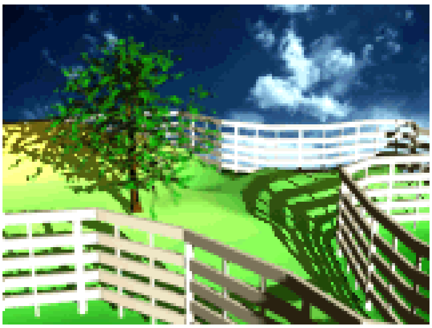
TIP Set the parameters before creating multiple plants. This can avoid slowing down the display, and might reduce editing you have to do on the plants.

Railing

Create panel > Geometry > AEC Extended > Railing button

Create menu > AEC Objects > Railing

Components of the railing object include rails, posts, and fencing. Fencing includes either pickets (balusters) or solid-filled material, such as glass or wood strip.



Railings used to create fences in a field.

You can create a railing object either by specifying the orientation and height of the railing, or by picking a spline path and applying the railing to that path. When 3ds Max applies railing to a spline path, the latter is called a *rail path*. Later, if you edit the rail path, the railing object automatically updates to follow the changes you made. You can use three-dimensional splines as rail paths.

When you create the lower rails, posts, and fencing components of a railing, you use the Spacing tool on page 996 to specify the spacing of those components. 3ds Max names the Spacing tool dialog for each railing component: Lower Rail Spacing, Post Spacing, or Picket Spacing.

TIP Use Railing to create complete railings for stairs. See Stairs on page 532 for more information.

Railings and Materials

By default, 3ds Max assigns five different material IDs to railings. The *aectemplates.mat* material library includes *Rail-Template*, a multi/sub-object material on page 5720 designed to be used with railings. Each component of the railing/material is listed below along with its corresponding Material ID.

Material ID	Railing/Material Component
1	Lower rails
2	Posts of the railing
3	Solid fill of the railing
4	Top of the railing
5	Pickets of the railing

NOTE 3ds Max does not automatically assign a material to the railing object. To use the included material, open the library and then assign the material to your object.

Procedures

The following procedures describe how to create railings combining each of the components: upper rail, lower rails, posts, picket fencing, and solid filled fencing.

You can create a railing object in any viewport, but for best results, use a Perspective, Camera, or Top viewport.

To create a railing:

1 Click and drag the railing to the desired length.

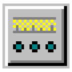
2 Release the mouse button, and then move the mouse vertically to set the height. Click to finish.

   By default, 3ds Max creates the top rail along with two posts, a lower rail at half the railing height, and two evenly spaced pickets.

3 If you need to, change any of the parameters to adjust the segments, length, profile, depth, width, and height of the rail.

To adjust lower rails:

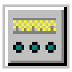
1 To modify the lower rail, or add more, choose an option from the Lower Rail(s) group > Profile list.

2  Specify the depth and width for the lower rails and then click the Lower Rail(s) > Spacing button.

3 Specify the number of lower rails you want using the Count option. Click Close to apply your changes. For more information on spacing options in this dialog, see Spacing Tool on page 996.

To create posts:

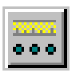
1 If you want to modify the posts, or add more, choose an option from the Profile list under the Posts rollout.

2  Specify the depth and width of the posts and how much they should extend above the top rail. Then click the Posts rollout > Spacing button.

3 Specify the number of posts you want using the Count option. Click Close to apply your changes. For more information on spacing options in this dialog, see Spacing Tool on page 996.

To create picket fencing:

1 Choose Fencing rollout > Type list > Pickets. The Solid Fill options will be unavailable.

2  Choose an option from the Profile list, specify the depth and width of the pickets, and then click the Picket rollout > Spacing button.

3 Specify the number of pickets you want using the Count option. Click Close to apply your changes. For more information on spacing options in this dialog, Spacing Tool on page 996.

To create solid-fill fencing:

1 Choose Fencing rollout > Type list > Solid Fill. (The options under Picket are unavailable).

2 Under Solid Fill, adjust the options for Thickness and offsets.

To create railings along a spline path:

Before you can create railings along a spline path, you need to create a spline, or use an existing spline from your scene.

1 Click Create panel > Geometry > AEC Extended > Railing.

2 Click Pick Railing Path, then select a spline in your scene.

   Since the number of segments is 1 by default, the upper rail extends for one segment between the start and end of the spline.

3 Change the amount of segments using the Modify panel > Segment setting.

   The higher the segment value, the more closely the railing approximates the spline shape.

4 If you want the railing to contain corners where the spline does, turn on Respect Corners.

5 Complete the remainder of the railing options as described in the preceding procedures.

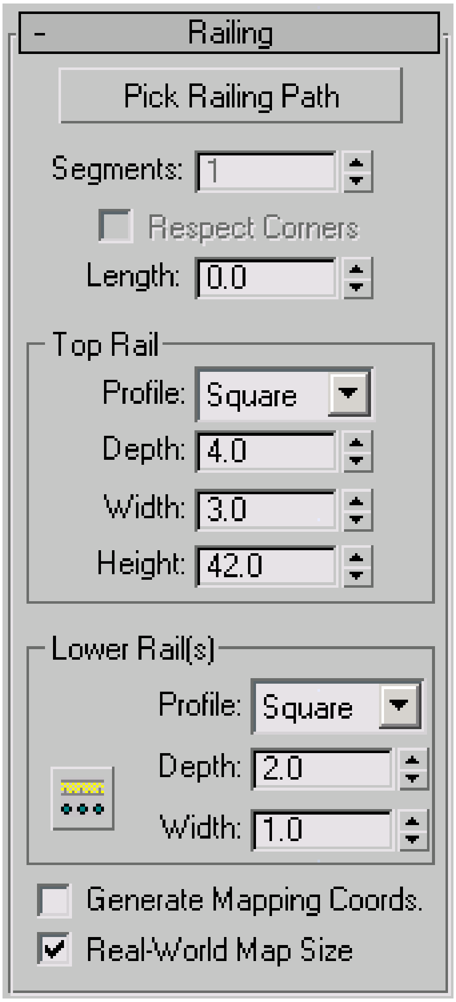
   Thereafter, the spline is associated with the railing; any changes you make to the spline shape are reflected in the railing.

Interface

Name and Color rollout

This rollout lets you set the selected railing's name and color. For detailed information, see Object Name and Wireframe Color on page 7631.

Railing rollout



Pick Railing Path Click this, and then click a spline in the viewport to use as the railing path. 3ds Max uses the spline as the path along which to apply the railing object.

If you edit the spline you've used as a railing path, the railing adjusts to the changes you make. 3ds Max doesn't immediately recognize 2D Shapes from a linked AutoCAD drawing. To recognize Shapes from a linked AutoCAD drawing, edit the Shape with Edit Spline on page 1447 in the Modify panel.

TIP When you create a railing using a closed spline for the rail path, open the Post Spacing dialog on page 996, turn off Start Offset and End Offset, and lock End Offset. This will ensure that 3ds Max properly creates the railing with any fill, pickets, and posts you specify.

NOTE Railing objects that use Pick Path do not stay on the path when substituted using the Substitute modifier. Substituted externally referenced railings do not undo when railings are associated with a path.

Segments Sets the number of segments of the railing object. Available only when you're using a railing path.

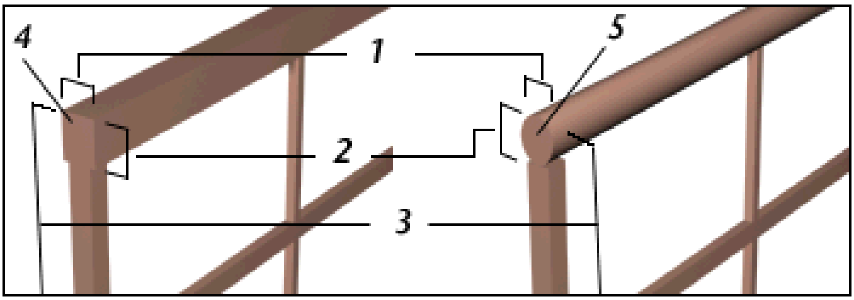
For a close approximation to a railing path, increase the number of segments. Be aware that a high number of segments increases file size and slows down the rendering speed. You might use fewer segments when the spline path has a low curvature (or none) and fewer segments provide an adequate approximation.

Respect Corners Puts corners in the railing to match the corners of the railing path.

Length Sets the length of the Railing object. When you drag the mouse, the length displays in the edit box.

Top Rail group

The defaults produce a top rail component, consisting of one segment by the length you specify, a square profile, four units deep, three units wide, and the height you specify.



1. Width

2. Depth

3. Height

4. Profile for the square top rail

5. Profile for the round top rail

Profile Sets the cross-section shape of the top rail.

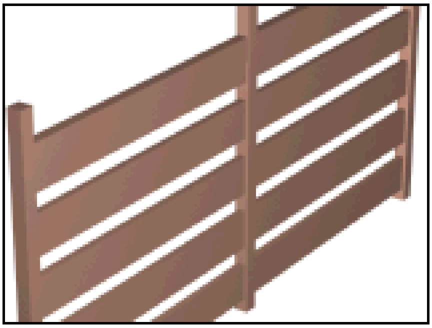
Depth Sets the depth of the top rail.

Width Sets the width of the top rail.

Height Sets the height of the top rail. During creation, you can drag the top rail to the height you want using the mouse in the viewport. Or you can enter the height amount from the keyboard or use the spinners.

Lower Rail(s) group

Controls the profile, depth, width, and spacing between the lower rails. You specify how many lower rails you want using the Lower Rail Spacing button.

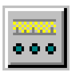


A railing with the rails defined by their profile, depth, and width as planks.

Profile Sets the cross-section shape of the lower rails.

Depth Sets the depth of the lower rails.

Width Sets the width of the lower rails.

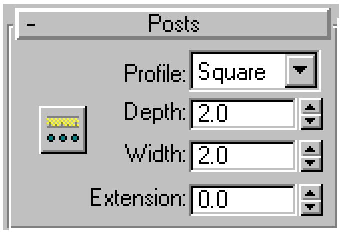
 **Lower Rail Spacing** Sets the spacing of the lower rails. When you click this button, the Lower Rail Spacing dialog displays. Specify the number of lower rails you want using the Count option. For more information on spacing options in this dialog, see Spacing Tool on page 996.

Generate Mapping Coords Assigns mapping coordinates on page 8034 to the railing object.

NOTE If a visible viewport is set to a non-wireframe or non-bounding-box display, Generate Mapping Coordinates is on for all primitives to which you apply a material containing a map with Show Map In Viewport on. If all viewports are set to wireframe or bounding box, 3ds Max turns on Generate Mapping Coordinates for primitives containing mapped materials at render time.

Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Posts rollout



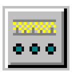
Controls the profile, depth, width, extension, and spacing between the posts. You specify how many posts you want using the Post Spacing button.

Profile Sets the cross-section shape of the posts: none, Square, or Round.

Depth Sets the depth of the posts.

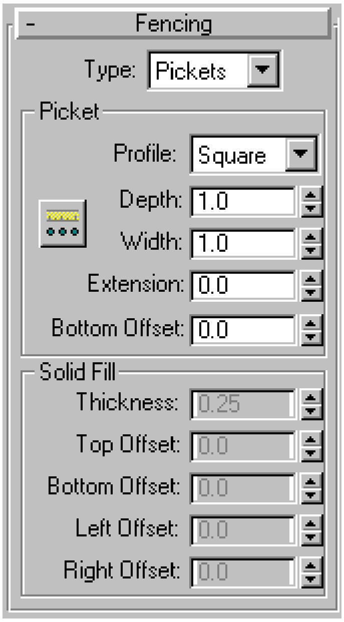
Width Sets the width of the posts.

Extension Sets the amount the posts extend above the bottom of the top railing.

 **Post Spacing** Sets the spacing of the posts. When you click this button, the Post Spacing dialog displays. Specify the number of posts you want using the Count option. For more information on spacing options in this dialog, see Spacing Tool on page 996.

TIP Setting Profile to (none) makes an "invisible" post. You might want to do this to create a railing with gaps between solid fill fencing. Or you could use it to make a railing with openings between groups of pickets. This is different from setting the post count to 0 in the Post Spacing dialog.

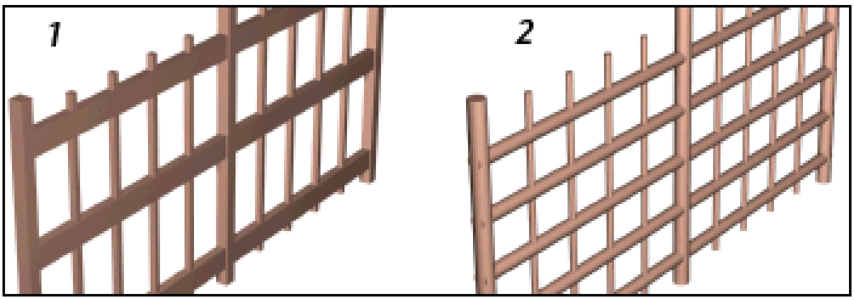
Fencing rollout



Type Sets the type of fencing between the posts: none, Pickets, or Solid Fill.

Picket group

Controls the profile, depth, width, and spacing between the pickets. Specify how many pickets you want using the Picket Spacing button. Available only when you set Type to Pickets.



1. A railing with pickets using a square profile

2. A railing with pickets using a round profile

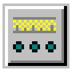
Profile Sets the cross-section shape of the pickets.

Depth Sets the depth of the pickets.

Width Sets the width of the pickets.

Extension Sets the amount the pickets extend above the bottom of the top railing.

Bottom Offset Sets the amount the pickets are offset from the bottom of the railing object.

 **Picket Spacing** Sets the spacing of the pickets. When you click this button, the Picket Spacing dialog displays. Specify the number of pickets you want using the Count option. For more information on spacing options in this dialog, see Spacing Tool on page 996.

Solid Fill group

Controls the thickness and offsets of the solid fill between the posts. Available only when you set Type to Solid.

Thickness Sets the thickness of the solid fill.

Top Offset Sets the offset of the solid fill from the bottom of the top rail.

Bottom Offset Sets the offset of the solid fill from the bottom of the railing object.

Left Offset Sets the offset between the solid fill and the adjacent left post.

Right Offset Sets the offset between the solid fill and the adjacent right post.

Wall

Create panel > Geometry > AEC Extended > Object Type rollout > Wall button

Create menu > AEC Objects > Wall

The Wall object is made up of three sub-object types that you can edit in the Modify panel. Similarly to the way you edit splines, you can edit the wall object on page 526, its vertices on page 527, its segments on page 528, and its profile on page 530.

When you create two wall segments that meet at a corner, 3ds Max removes any duplicate geometry. This "cleaning up" of the corners might involve trimming. 3ds Max cleans up only the first two wall segments of a corner, not any other wall segments that might share the corner. 3ds Max does not clean up intersections.

Inserting Doors and Windows in a Wall

3ds Max can automatically make openings for doors and windows in a wall. At the same time, it links the door or window to the wall as it child. The most effective way of doing both is to create the doors and windows directly on a wall segment by snapping to the faces, vertices, or edges of the wall object.

If you move, scale, or rotate the wall object, the linked door or window moves, scales, or rotates along with the wall. If you move the linked door or window along the wall, using the door or window's local coordinate system and constraining motion to the XY plane on page 7503, the opening will follow. Also, if you change a door or window's overall width and height on the Modify panel, the hole will reflect those changes.

For further information, see the procedure To create and place a window or door in a wall on page ?.

Walls and Materials

By default, 3ds Max assigns five different material IDs to walls. The *aectemplates.mat* material library includes *Wall-Template*, a multi/sub-object material on page 5720 designed to be used with walls. Each component of the wall/material is listed below along with its corresponding Material ID.

Material ID	Wall/Material Component
1	Vertical ends of the wall
2	Outside of the wall
3	Inside of the wall
4	Top of the wall, including any edges cut out of the wall
5	Bottom of the wall

NOTE 3ds Max does not automatically assign a material to the wall object. To use the included material, open the library and then assign the material to your object.

NOTE The definitions of slots 2 and 3 are interchangeable; inside and outside simply depend on your point of view, and how you created the wall.

See also:

- Editing Wall Objects on page 526

Tips

- To make a passageway through a wall you can perform a Boolean operation on page 757 with the wall as Operand A, and another object, such as a box or an extruded archway shape, as Operand B. The wall will still be accessible at the Boolean sub-object level. Then, you can add a window or door in the passageway, and link on page 3342 it as a child of the wall.
- Single walls with many windows and doors can become slow to use because of the amount of boolean calculations used. To speed up movement and editing, you might consider using multiple walls instead of a single wall.
- You can speed up performance in a scene with many walls, windows and doors by collapsing them. First save an uncollapsed version for any future parametric changes you might want to make. Then double-click the wall to select it and its children. Next use Convert To from the right-click menu to convert them to an editable mesh, and so on.

Procedures

To create a wall:

You can create a wall in any viewport, but for vertical walls, use a Perspective, Camera, or Top viewport.

1. Set parameters for the Width, Height, and Justification of the wall.
2. In a viewport, click and release, move the mouse to set the desired length for the wall segment, and click again.

   This creates a wall segment. You can end the wall by right-clicking or you can continue to create another wall segment.

3 To add another wall segment, move the mouse to set the length of the next wall segment and click again.

If you create a room by ending a segment at the end of another segment of the same wall object, 3ds Max displays the Weld Point dialog. This dialog lets you convert the two end vertices into a single vertex, or to keep the two end vertices distinct.

4 If you want the wall segments to be welded at that corner so that when you move one wall, the other wall stays correct at the corner, click Yes. Otherwise, click No.

5 Right-click to end the wall, or continue to add other wall segments.

To attach separate walls:

1 Select a wall object.

2 On the Modify panel, click Attach, and then pick another wall object.

The two wall objects become part of the same wall object, but are not physically connected.

Attach stays active, and you can continue clicking wall segments to attach. To stop attaching, click the Attach button or right-click in the active viewport.

To attach multiple wall objects simultaneously to the selected wall object, click Attach Multiple on the Modify panel to open the Attach Multiple dialog. This works the same as the Select From Scene dialog on page 228, except that it shows only wall objects; choose multiple walls to attach, and then click the Attach button.

To connect vertices in a wall:

This method lets you connect two separate wall sections with a new segment.

TIP It is easier to work with wall vertices in wireframe view mode.

---

1 Select a wall object that has more than one section. Typically you would use Attach to create such an object.

2 In the modifier stack on page 7635, go to the Vertex sub-object level.

3 Click Connect and point the mouse over an end vertex until the cursor changes to a cross.

4 Click once over the end vertex.

5 Move the cursor to another end vertex, and then click to connect the two segments.

To insert a vertex in a wall:

It is easier to work with wall vertices in wireframe view mode.

1 Select a wall segment.

2 In the modifier stack on page 7635, go to the Vertex sub-object level.

3 Click Insert.

A highlighted line appears along the bottom of the wall, showing where you can insert vertices.

4 Click anywhere on the highlighted line to insert a vertex.

The new vertex is attached to the mouse cursor.

5 Move the mouse to position the vertex, and then click to place it.

Now the mouse is attached to one of the new segments.

6 Move the mouse along the segment and click to add vertices.

7 Right-click to finish working on this segment. You can now insert vertices in other segments, or right-click again to exit Insert mode.

To detach and reorient a copy of a wall segment:

TIP It is easier to work with wall vertices in wireframe view mode.

---

1 Select a wall.

2 In the modifier stack on page 7635, go to the Segment sub-object level.

3 Select a wall segment.

4 Turn on both Reorient and Copy, and then click Detach.

5 Enter a name for the new wall object in the Detach dialog or click OK to accept the default name.

3ds Max copies the original wall's Local coordinate system on page 8026 when it makes the copy of the detached segment. It places the new object so that its Local coordinate system is coincident with the World space origin on page 8174.

To add a gable point to a wall profile or adjust for uneven terrain:

TIP It is easier to work with wall vertices in wireframe view mode.

---

1 Select a wall.

2 In the modifier stack on page 7635, go to the Profile sub-object level.

3 Select a wall profile by clicking a wall segment.

   A grid appears.

4 To add a gable point procedurally, set the height and click Create Gable.

   If you prefer to add the profile point manually, click Insert, click a point on the highlighted top profile, drag the new point into place and then release where you want to place the new gable point. You can move profile points you create with Insert only within the plane of the wall segment, and you cannot move them below the original top edge.

   If you want to adjust the profile for uneven terrain below a wall, click Insert, pick the highlighted bottom profile and add points as necessary.

   If you want to extend multiple segments uniformly downward below floor level, do the following: At the Segment sub-object level, select the segments and, on the Edit Segment rollout, enter a negative Bottom Offset value to move the segments downward. Add the absolute value of the Bottom Offset setting back to the Height value to bring the top of the wall height back up and make it flush with the other wall segments.

To apply a texture to a wall:

Walls are created with five different material IDs on page 8038 for their various parts.

The *aectemplates.mat* material library includes *Wall-Template*, a Multi/Sub-Object material designed for use with walls. You can copy or copy and modify this template, or create your own material as follows:

1 Create a Multi/Sub-Object material on page 5720 using five textures for the following Material IDs:
   - Slot #1 is the material for the vertical ends on the wall
   - Slot #2 is the material for the outside of the wall
   - Slot #3 is the material for the inside of the wall
   - Slot #4 is the material for the top of the wall, as well as any inside edges cut out of the wall
   - Slot #5 is the material for the bottom of the wall

   NOTE The definitions of slots 2 and 3 are interchangeable; inside and outside simply depend on your point of view, and how you created the wall.

2 If the top and bottom surfaces of the wall aren't visible in the rendered scene, you can use a three-sided material instead. The inside and outside of the wall are relative to the direction in which the wall was created. To swap a texture between slots in the Material Editor, drag one of the textures over the other slot in the Basic Parameters rollout of the Multi/Sub-Object material, and then choose Swap.

3 For greater control in tiling across the wall surface, apply a Map Scaler world-space modifier on page 1190 to the wall. Then adjust the scale of the map in the Map Scaler's Parameters rollout.

To create and place a window or door in a wall:

For best results, perform this procedure in a wireframe viewport.

1 Create a window on page 577 or door on page 562 (hereafter referred to as "window" for brevity) directly on an existing wall. You can define the window's exact dimensions after insertion. Use edge snap on page 2662 for the first snaps to place and align the window on the wall and to establish its exact depth. Snap to and then click the near top edge of the wall to start creation. Drag to another edge snap point on the near top edge of the wall and release to align the window with the wall segment and to set its width. Snap to the rear top edge of the wall to set the proper depth and click. Move the cursor downward and click to define the window

height. This final click doesn't require a snap, as it simply defines a rough height.

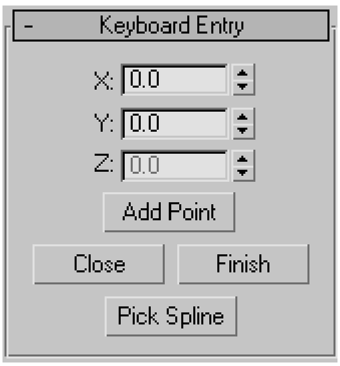
2. The window should now be cut out of the wall. On the Modify panel for windows or doors, set the correct width and height. Change the depth if it's different from the snap depth you set above.
3. Use vertex snap to move the window or door from a reference point to a known point on the wall segment. Then

   Next, use relative offset values from this new position to accurately locate the window or door. As an example, following the next two steps, you could move a window from its top left corner to the top left corner of the wall segment so that you can then move it 3 feet to the right and 2 feet down.
4. With the window or door selected, set the coordinate system to Local.
5. On the Coordinate Display on page 7541, activate Offset mode and then enter the offset distances on the X axis for horizontal and the Y axis for vertical.

NOTE For best results, do not position an inserted window or door at the bottom of a wall.

Interface

Keyboard Entry rollout



X Sets the coordinate position along the X axis for the start point of a wall segment in the active construction plane.

Y Sets the coordinate position along the Y axis for the start point of a wall segment in the active construction plane.

Z Sets the coordinate position along the Z axis for the start point of a wall segment in the active construction plane.

Add Point Adds the point from the X, Y, and Z coordinate values you enter.

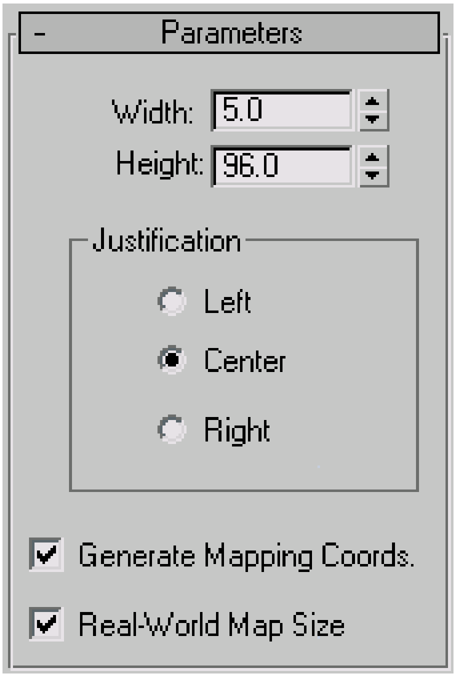
Close Ends creation of the wall object and creates a segment between the end point of the last segment and the start point of the first segment, to make a closed wall.

Finish Ends creation of the wall object, leaving it open ended.

Pick Spline Lets you use a spline as the wall path. Click this, and then click a spline in the viewport to use as the wall path. 3ds Max uses the spline as the path along which to apply the wall object. 3ds Max doesn't immediately recognize 2D Shapes from a linked AutoCAD drawing. To recognize Shapes from a linked AutoCAD drawing, edit the Shape with Edit Spline on page 1447 from the Modify panel.

NOTE If you designate a curved spline as the path, 3ds Max creates straight wall segments that approximate the spline as closely as possible, with one wall segment per spline segment.

Parameters rollout



The defaults produce a wall object 5 units wide, 96 units high, and justified at the center of the wall.

Width Sets the thickness of the wall. Range=0.01 unit to 100,000 units. Default=5.

Height Sets the height of the wall. Range=0.01 unit to 100,000 units. Default=96.

Justification group

Left Justifies the wall at the left edge of its baseline (the line between the wall's front and back sides, which is equal to the wall thickness). If you turn Grid Snap on, the left edge of the wall's baseline snaps to the grid line.

Center Justifies the wall at the center of its baseline. If you turn Grid Snap on, the center of the wall's baseline snaps to the grid line. This is the default.

Right Justifies the wall at the right edge of its baseline. If you turn Grid Snap on, the right edge of the wall's baseline snaps to the grid line.

Generate Mapping Coords Assigns mapping coordinates on page 8034 to the wall. Default=on.

Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Editing Wall Objects

Select a wall object. > Modify panel

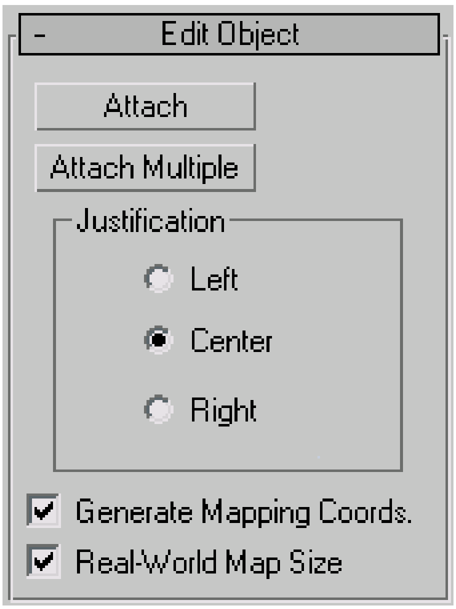
The following reference describes the Wall options on the Modify panel. It's generally easiest to edit wall objects in wireframe mode.

See also:

- Wall on page 516

Interface

Edit Object rollout



This rollout appears when you select a wall object at the object level; other rollouts, discussed below appear at the different sub-object levels.

Attach Attaches another wall in a viewport to the selected wall by a single pick. The object you attach must also be a wall. 3ds Max applies the material of the selected wall to the wall being attached.

Attach Multiple Attaches other walls in a viewport to the selected wall. Click this button to open the Attach Multiple dialog, which lists all the other wall objects in the scene. Select the walls you want to attach from the list and click the Attach button. 3ds Max applies the material of the selected wall to the walls being attached.

Justification group

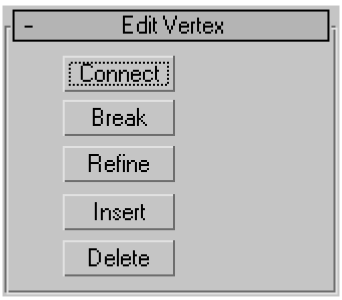
See Justification on page 525.

Generate Mapping Coords. Assigns mapping coordinates on page 8034 to the wall. Default=on.

Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Edit Vertex rollout

Appears at the Vertex sub-object level. Each wall segment has two vertices; one in each bottom corner. In wireframe views, wall vertices appear as + symbols. Connected segments in the same wall object each share a vertex. Moving a wall vertex has the effect of scaling attached segments as well as rotating them about their other vertices. You cannot rotate or scale wall vertices.



Connect Lets you connect any two vertices, creating a new linear segment between the vertices.

Click this button, click a vertex, and then click a second vertex on a different segment. When you move the cursor over a valid second vertex, the mouse icon changes to a Connect icon.

Break Lets you disconnect segments at a shared vertex.

TIP Select a vertex shared between wall segments, and then click the Break button. The segments become disconnected, and each has its own end vertex at the position of the previously shared vertex.

Refine Adds a vertex to the position along a wall segment that you click. When you move the cursor over a valid Refine point, the mouse icon changes to a Refine icon.

Insert Inserts one or more vertices, creating additional segments. When you move the cursor over the a valid Insert point, the mouse icon changes to an Insert icon. Right-click to stop inserting new vertices and segments.

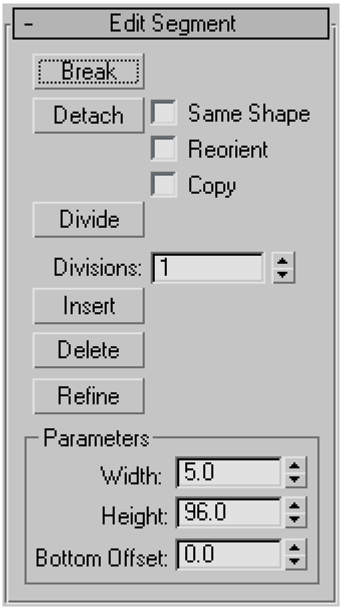
Delete Deletes the currently selected vertex or vertices, including any segments in between.

Deleting vertices shared by two or more segments doesn't create a gap, but rather results in a single segment connecting vertices adjacent to those being deleted.

Edit Segment rollout

This rollout appears when you select a wall object and then access Segment sub-object level.

Each wall segment is defined by, and effectively connects, two wall vertices. Moving a segment is the same as moving its two vertices in tandem. It has the effect of scaling adjacent wall segments as well as rotating them about their other vertices. You can scale a wall segment horizontally only (any Scale function does this). You cannot rotate a segment.



Break Specifies a break point in a wall segment.

You needn't select a segment first. When you move the cursor over the object, the mouse icon changes to a Break icon. The position you select on the segment becomes two coincident vertices, and 3ds Max breaks the segment in two.

Detach Detaches wall segments you select and creates a new wall object out of them.

Same Shape Detaches the wall segment keeping it part of the same wall object. If you also turn on Copy, 3ds Max places a detached copy of the segment in the same location.

Reorient Detaches the wall segment, copies the object's Local coordinate system on page 8026, and places the segment so that its object Local coordinate system is coincident with the World space origin on page 8174. If you also turn on Copy, 3ds Max detaches a copy of the segment and leaves the original segments in place.

Copy Copies the detached wall segment rather than moving it.

Divide Subdivides each segment by the number of vertices specified in the Divisions spinner. Select one or more segments, set the Divisions spinner, and then click Divide.

Divisions Sets the number by which to divide the segment.

Insert Provides the same function as the Insert button on page 528 in Vertex sub-object selection. Inserts one or more vertices, creating additional segments. When you move the cursor over the a valid Insert point, the mouse icon changes to an Insert icon. Right click to stop inserting new vertices and segments.

Delete Deletes any selected wall segments in the current wall object.

Refine Provides the same function as the Refine button on page 528 at the Vertex sub-object level. Adds a vertex to the position along a wall segment you select. When you move the cursor over a valid Refine point, the mouse icon changes to a Refine icon.

Parameters group

Width Changes the width of a selected segment or segments.

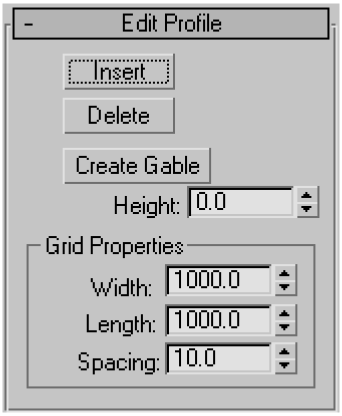
Height Changes the height of a selected segment or segments.

Bottom Offset Sets the distance of the bottom of the selected segment or segments from the floor.

Edit Profile rollout

This rollout appears when you select a wall object and then access Profile sub-object level.

The term "profile" refers to the outline of a wall segment's top and bottom edges. When in Profile sub-object mode, the selected wall object's inner horizontal edges appear dark orange. Click any of these edges to select the corresponding segment, highlight it in red, and place a temporary active grid in the plane of the segment. At that point, you can insert and delete vertices along the horizontal edges, move an inserted vertex along the grid to change the profile, create gables, and change the grid properties.



Insert Inserts a vertex so that you can adjust the profile of the selected wall segment.

Use this option to adjust the profile of walls under gables or to align walls to a slope. When you move the cursor over the selected segment, the mouse icon changes to an Insert icon. Click to insert a new profile point, then drag and release to position and place it. You can add new profile points to both the top and the bottom of the wall, but you cannot position profile points below the original top edge or above the original bottom edge.

Delete Deletes the selected vertices on the profile of the selected wall segment.

Create Gable Creates a gable by moving the center point of the top profile of the selected wall segment to a height you specify.

Select the segment, set the height, and then click Create Gable.

Height Specifies the height of a gable.

Grid Properties group

The grid constricts profile point insertion and movement to the plane of the wall and allows you to snap to grid points on the plane of the wall.

Width Sets the width of the active grid.

Length Sets the length of the active grid.

Spacing Sets the size of the smallest square in the active grid.

Stairs

Create panel > Geometry > Stairs

Create menu > AEC Objects

You can create four different types of stairs in 3ds Max:

Spiral Stair on page 540

Straight Stair on page 549

L-Type Stair on page 534

U-Type Stair on page 555

Railings and Materials

By default, 3ds Max assigns seven different material IDs to stairs. The *aectemplates.mat* material library includes *Stair-Template*, a multi/sub-object material on page 5720 designed to be used with stairs. Each component of the stair/material is listed below along with its corresponding Material ID.

Material ID	Railing/Material Component
1	Treads of the stairs
2	Front riser of the stairs
3	Bottom, back, and sides of the risers of the stairs
4	Center pole of the stairs
5	Handrails of the stairs
6	Carriage of the stairs
7	Stringers of the stairs

NOTE 3ds Max does not automatically assign a material to the stairs object. To use the included material, open the library and then assign the material to your object.

Procedures

To create railings on stairs:

1 Create the stairs. See individual stair-type topics for more information.

2 In the Generate Geometry group, turn on Rail Path > Left and Right.

   3ds Max places left and right rail paths above the stairs.

3 In the Railings rollout, set Height to **0.0**.

4 Click Create panel > AEC Extended > Railing on page 507 to create the first railing.

5 Click Railing rollout > Pick Railing Path and select one of the rail paths on the stairs.

6 Adjust the railing parameters.

   3ds Max remembers the parameters you set. When you create the next railing, it will have the same parameters as you set for the first railing.

7 Right-click to end the creation of the first railing.

8 Click Railing again to create the second railing.

9 Click Pick Railing Path and select the other rail path on the stairs.

Interface

Object Type rollout

Stair Selection Buttons Click one of these to specify the type of stairs you want to create.

Name and Color rollout

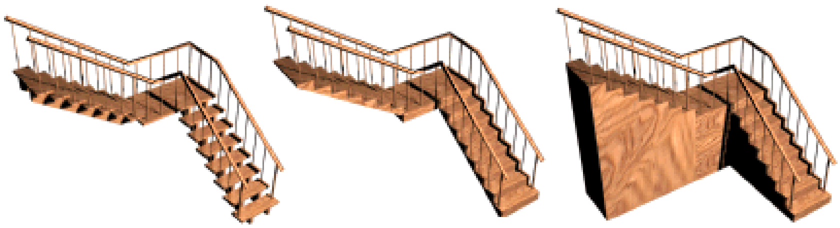
This rollout lets you set the stairs object's name and color. For detailed information, see Object Name and Wireframe Color on page 7631.

L-Type Stair

Create panel > Geometry > Stairs > L-Type Stair button

Create menu > AEC Objects > L-Type Stair

The L-Type Stair object lets you create a staircase with two flights at right angles to each other.



Types of L-type stair: open, closed, and boxed

L-type stairs have two flights at right angles, and a landing.

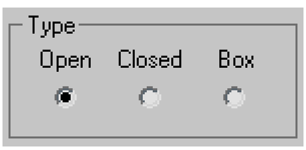
Procedures

To create L-Type stairs:

1. In any viewport, drag to set the length for the first flight. Release the mouse button, then move the cursor and click to set the length, width, and direction for the second flight.
2. Move the cursor up or down to define the rise of the stairs, then click to end.
3. Adjust the stairs by using the options in the Parameters rollout.

Interface

Parameters rollout > Type group

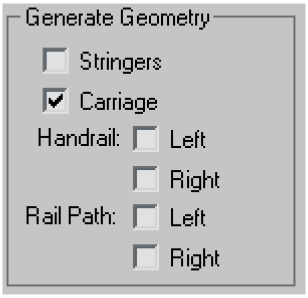


Open Creates an open riser stair, as shown on the left in the illustration above.

Closed Creates a closed riser stair, as shown in the center in the illustration above.

Box Creates a stair with closed risers and closed stringers on both sides, as shown on the right in the illustration above.

Generate Geometry group



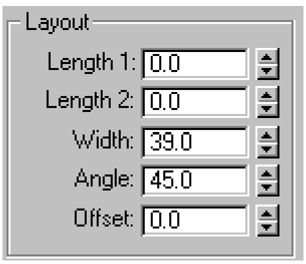
Stringers Creates stringers along the ends of the treads of the stairs. To modify the stringers' depth, width, offset and spring from the floor, see Stringers rollout on page 538.

Carriage Creates an inclined, notched beam under the treads which supports the steps or adds support between the stringers of the stairs. You might also know this as a *carriage piece*, *a horse*, or a *rough string*. See Carriage rollout on page 539 to modify the parameters.

Handrail Creates left and right handrails. See Railings rollout on page 540 to modify the handrails' height, offset, number of segments, and radius.

Rail Path Creates left and right paths you can use to install railings on the stairs. See Stairs on page 532 for the instructions on how to do this.

Layout group



Length 1 Controls the length of the first flight of stairs.

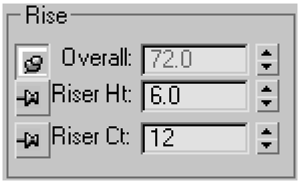
Length 2 Controls the length of the second flight of stairs.

Width Controls the width of the stairs, including the steps and the landing.

Angle Controls the angle of the second flight from the landing. Range=-90 to 90 degrees.

Offset Controls the distance of the second flight from the landing. The length of the landing adjusts accordingly.

Rise group

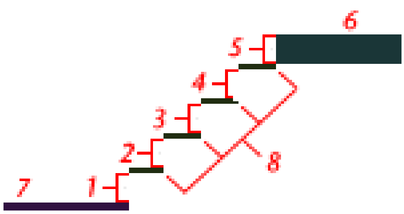


3ds Max keeps one Rise option locked while you adjust the other two. To lock an option, you click a push pin. To unlock an option you click a raised push pin. 3ds Max locks the spinner value of the parameter with the depressed push pin and allows the spinner values of the parameter with the raised push pins to change.

Overall Controls the height of the flight of stairs.

Riser Ht Controls the height of the risers.

Riser Ct Controls the number of risers. There will always be one more riser than steps. This *implied* riser is between the top step of the stair and the upper floor.



Linear stair with five risers

1 through 4. Risers

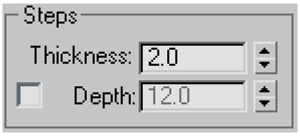
5. The implied riser

6. The upper floor you snap to

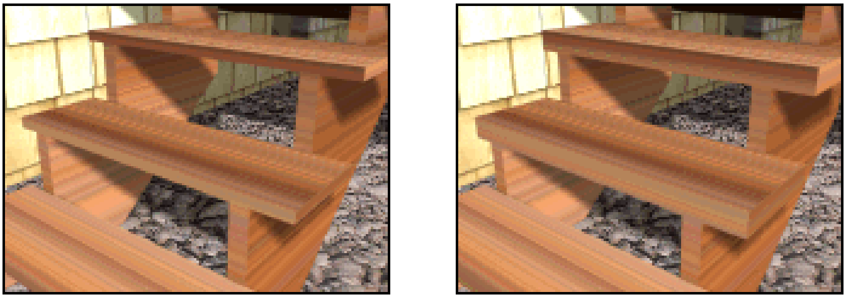
7. The lower floor you snap to

8. The steps

Steps group

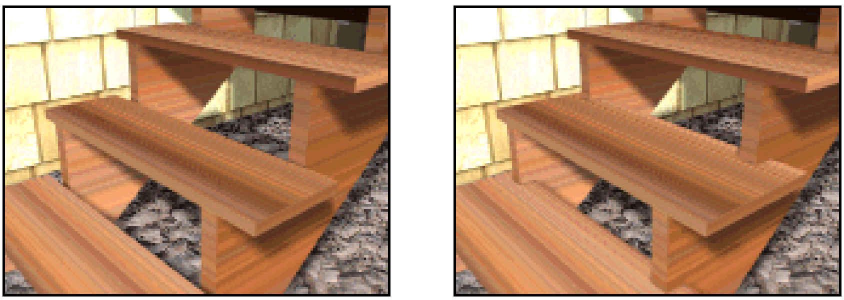


Thickness Controls the thickness of the steps.



Step thickness variance between two stairs

Depth Controls the depth of the steps.



Step depth variance between two stairs

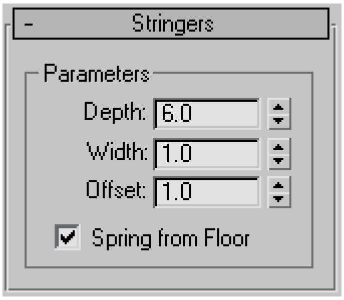
Generate Mapping Coords Applies default mapping coordinates on page 8034 to the stairs.

NOTE If a visible viewport is set to a non-wireframe or non-bounding-box display, Generate Mapping Coordinates is on for all primitives to which you apply a material containing a map with Show Map In Viewport on. If all viewports are set to wireframe or bounding box, 3ds Max turns on Generate Mapping Coordinates for primitives containing mapped materials at render time.

Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Stringers rollout

These controls are available only when you turn on Stringers on the Parameters rollout > Generate Geometry group.

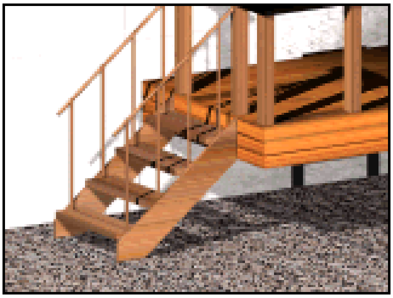


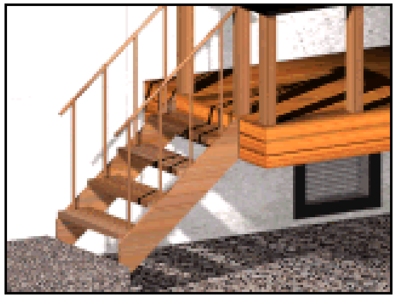
Depth Controls how far down the stringers reach toward the floor.

Width Controls the width of the stringers.

Offset Controls the vertical distance of the stringers from the floor.

Spring from Floor Controls whether the stringer starts at the floor, flush with the start of the first riser, or if the stringer extends below the floor. You control the amount the stringer extends below the floor with the Offset option.



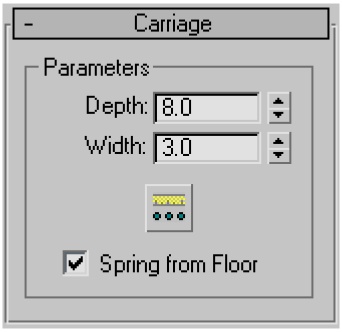


Left: The stringer extending below the floor. (Spring From Floor off.)

Right: The stringer springing from the floor. (Spring From Floor on.)

Carriage rollout

These controls are available only when you turn on Carriage on the Parameters rollout > Generate Geometry group.

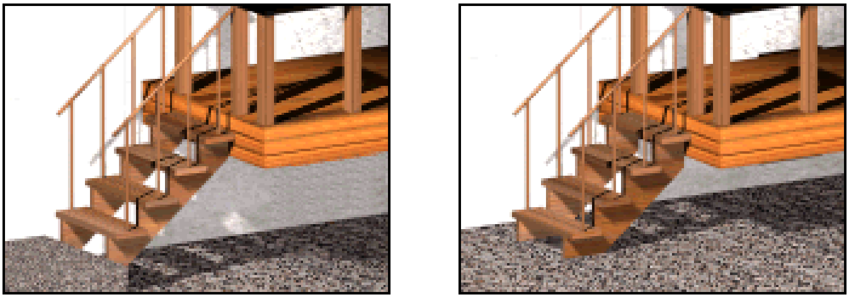


Depth Controls how far down the carriage reaches toward the floor.

Width Controls the width of the carriage.

Carriage Spacing Sets the spacing of the carriage. When you pick this button, the Carriage Spacing dialog displays. Specify the number of carriages you want using the Count option. For more information on spacing options in this dialog, see Spacing Tool on page 996.

Spring from Floor Controls whether the carriage starts at the floor, flush with the start of the first riser, or if the carriage extends below the floor. You control the amount the carriage extends below the floor with the Offset option.

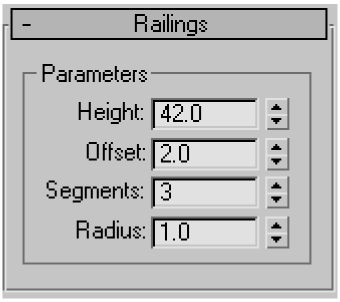


Left: The carriage springing from the floor. (Spring From Floor on.)

Right: The carriage extending below the floor. (Spring from Floor off.)

Railings rollout

These controls are available only when you turn on one or more of the Handrail or Rail Path options on the Parameters rollout > Generate Geometry group. Also, Segments and Radius aren't available if neither of the Handrail options is on.



Height Controls the height of the railings from the steps.

Offset Controls the offset of the railings from the ends of the steps.

Segments Controls the number of segments in the railings. Higher values display smoother railings.

Radius Controls the thickness of the railings.

Spiral Stair

Create panel > Geometry > Stairs > Spiral Stair button

Create menu > AEC Objects > Spiral Stair

The Spiral Stair object lets you specify the radius and number of revolutions, add stringers and a center pole, and more.

Types of spiral stair: open, closed, and boxed

Spiral stairs wind around a center

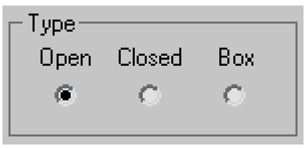
Procedures

To create spiral stairs:

1. In any viewport, click for the start point of the stairs, and drag to the specify the radius you want.
2. Release the mouse button, move the cursor up or down to specify the overall rise, and click to end.
3. Adjust the stairs with options in the Parameters rollout.

Interface

Parameters rollout > Type group

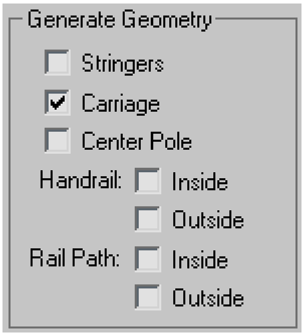


Open Creates an open riser stair, as shown on the left of the illustration above.

Closed Creates a closed riser stair, as shown in the center of the illustration above.

Box Creates a stair with closed risers and closed stringers on both sides, as shown on the right of the illustration above.

Generate Geometry group



Stringers Creates stringers along the ends of the treads of the stairs. To modify the stringers' depth, width, offset and spring from the floor, see Stringers rollout on page 545.

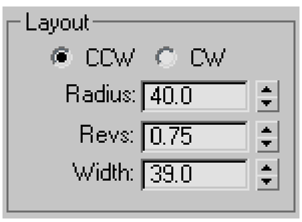
Carriage Creates an inclined, notched beam under the treads which supports the steps or adds support between the stringers of the stairs. You might also know this as a *carriage piece*, *a horse*, or a *rough string*. See Carriage rollout on page 546 to modify the parameters.

Center Pole Creates a pole at the center of the spiral. See Center Pole rollout on page 547 to modify the parameters of the pole.

Handrail Creates inside and outside handrails. See Railings rollout on page 548 to modify the handrails' height, offset, number of segments, and radius.

Rail Path Creates inside and outside paths which you can use to install railings on the stairs. See Stairs on page 532 for the instructions on how to do this.

Layout group



CCW Orients the spiral stairs to be a right-hand flight of stairs.

CW Orients the spiral stairs to be a left-hand flight of stairs.

Left: CCW (counterclockwise) right-hand spiral stairs. The arrow indicates "Up."

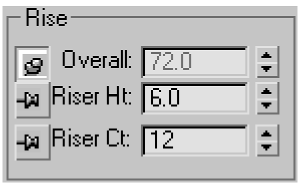
Right: CW (clockwise) left-hand spiral stairs. The arrow indicates "Up."

Radius Controls the size of the radius of the spiral.

Revs Controls the number of revolutions in the spiral.

Width Controls the width of the spiral stairs.

Rise group

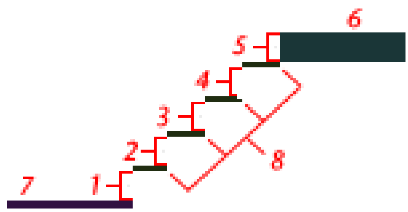


3ds Max keeps one Rise option locked while you adjust the other two. To lock an option, click a pushpin button. To unlock an option, click a raised pushpin. 3ds Max locks the spinner value of the parameter with the depressed pushpin and allows the spinner values of the parameter with the raised pushpins to change.

Overall Controls the height of the flight of stairs.

Riser Ht Controls the height of the risers.

Riser Ct Controls the number of risers. There will always be one more riser than steps. This *implied* riser is between the top step of the stair and the upper floor.



Linear stair with five risers

1 through 4. Risers

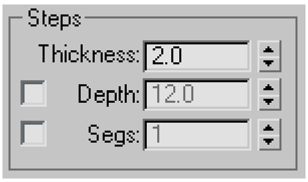
5. The implied riser

6. The upper floor you snap to

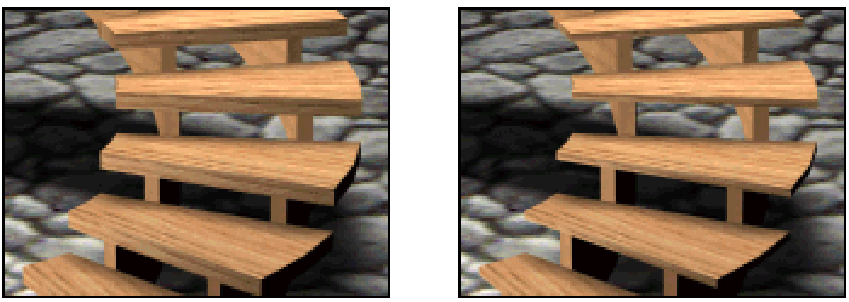
7. The lower floor you snap to

8. The steps.

Steps group



Thickness Controls the thickness of the steps.



Step thickness variance between two stairs

Depth Controls the depth of the steps.

Step depth variance between two stairs

Segs Controls the number of segments 3ds Max uses to construct the steps.

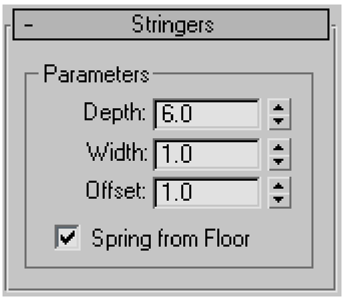
Generate Mapping Coords Applies default mapping coordinates on page 8034 to the stairs.

NOTE If a visible viewport is set to a non-wireframe or non-bounding-box display, Generate Mapping Coordinates is on for all primitives to which you apply a material containing a map with Show Map In Viewport on. If all viewports are set to wireframe or bounding box, 3ds Max turns on Generate Mapping Coordinates for primitives containing mapped materials at render time.

Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Stringers rollout

These controls are available only when you turn on Stringers on the Parameters rollout > Generate Geometry group.

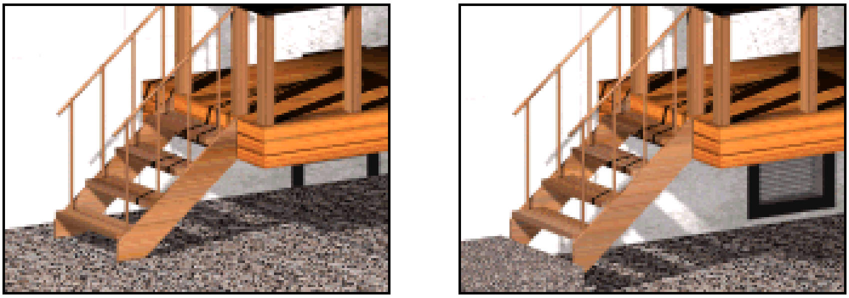


Depth Controls how far down the stringers reach toward the floor.

Width Controls the width of the stringers.

Offset Controls the vertical distance of the stringers from the floor.

Spring from Floor Controls whether the stringer starts at the floor, flush with the start of the first riser, or if the stringer extends below the floor. You control the amount the stringer extends below the floor with the Offset option.

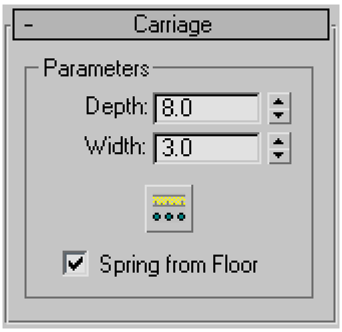


Left: The stringer extending below the floor. (Spring From Floor turned off.)

Right: the stringer springing from the floor. (Spring From Floor turned on.)

Carriage rollout

These controls are available only when you turn on Carriage on the Parameters rollout > Generate Geometry group.

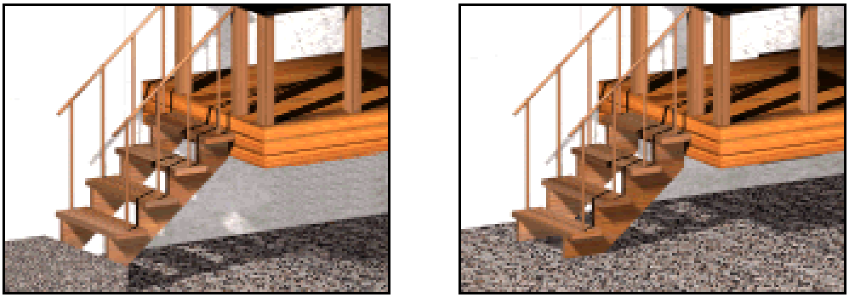


Depth Controls how far down the carriage reaches toward the floor.

Width Controls the width of the carriage.

Carriage Spacing Sets the spacing of the carriage. When you pick this button, the Carriage Spacing dialog displays. Specify the number of carriages you want using the Count option. For more information on spacing options in this dialog, see Spacing Tool on page 996.

Spring from Floor Controls whether the carriage starts at the floor, flush with the start of the first riser, or if the carriage extends below the floor. You control the amount the carriage extends below the floor with the Offset option.

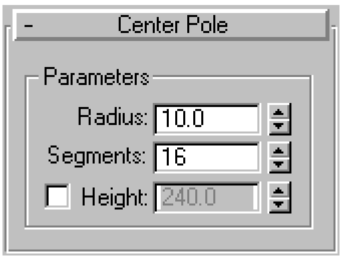


Left: The carriage springing from the floor. (Spring From Floor turned on.)

Right: The carriage extending below the floor. (Spring From Floor turned off.)

Center Pole rollout

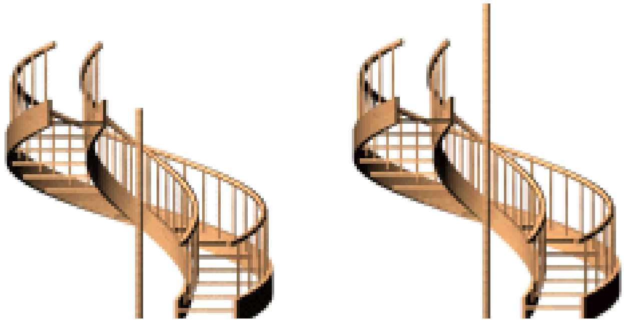
These controls are available only when you turn on Center Pole on the Parameters rollout > Generate Geometry group.



Radius Controls the radius size of the center pole.

Segments Controls the number of segments in the center pole. Higher values display a smoother pole.

Height The spinner controls the height of the center pole. Turning on Height lets you adjust the height of the pole independently of the stairs. Turning off Height makes the spinner unavailable and locks the top of the pole to the top of the implied last riser. Typically, this riser would attach to the fascia of a landing.

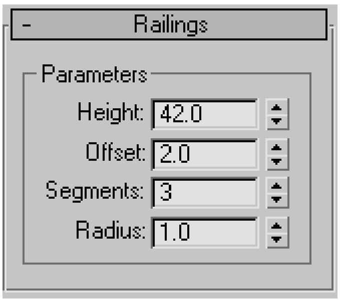


Left: The center pole locked to the top of the implied last riser. (Height turned off.)

Right: The center pole adjusted to the height you specify. (Height turned on.)

Railings rollout

These controls are available only when you turn on one or more of the Handrail or Rail Path options on the Parameters rollout > Generate Geometry group. Also, Segments and Radius aren't available if neither of the Handrail options is on.



Height Controls the height of the railings from the steps.

Offset Controls the offset of the railings from the ends of the steps.

Segments Controls the number of segments in the railings. Higher values display smoother railings.

Radius Controls the thickness of the railings.

Straight Stair

Create panel > Geometry > Stairs > Straight Stair button

Create menu > AEC Objects > Straight Stair

The Straight Stair object lets you create a simple staircase, with optional stringers, carriage, and handrail.

Types of straight stair: open, closed, and boxed

Straight stairs have a single flight.

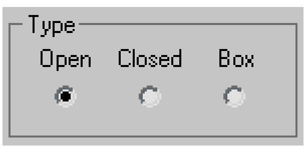
Procedures

To create straight stairs:

1. In any viewport, drag to set the length. Release the mouse button, then move the cursor and click to set the width you want.
2. Move the cursor up or down to define the rise of the stairs, and click to end.
3. Adjust the stairs with the options in the Parameters rollout.

Interface

Parameters rollout > Type group

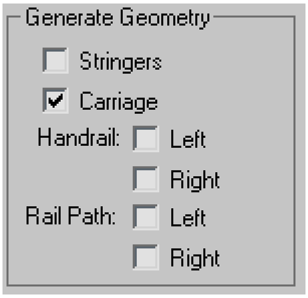


Open Creates an open riser stair as shown on the left of the illustration above.

Closed Creates a closed riser stair as shown in the center of the illustration above.

Box Creates a stair with closed risers and closed stringers on both sides as shown on the right of the illustration above.

Generate Geometry group



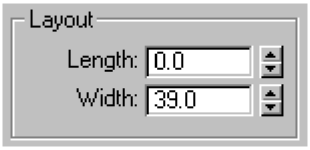
Stringers Creates stringers along the ends of the treads of the stairs. To modify the stringers' depth, width, offset and spring from the floor, see Stringers rollout on page 553.

Carriage Creates an inclined, notched beam under the treads which supports the steps or adds support between the stringers of the stairs. You might also know this as a *carriage piece*, *a horse*, or a *rough string*. See Carriage rollout on page 554 to modify the parameters.

Handrail Creates left and right handrails. See Railings rollout on page 555 to modify the handrails' height, offset, number of segments, and radius.

Rail Path Creates left and right paths you can use to install railings on the stairs. See Stairs on page 532 for the instructions on how to do this.

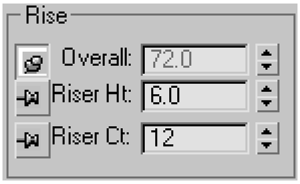
Layout group



Length Controls the length of the stairs.

Width Controls the width of the stairs.

Rise group

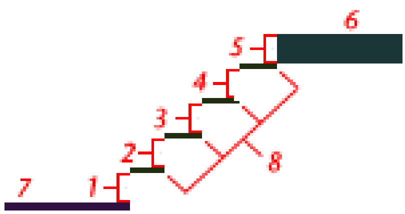


3ds Max keeps one Rise option locked while you adjust the other two. To lock an option, you click a push pin. To unlock an option you click a raised push pin. 3ds Max locks the spinner value of the parameter with the depressed push pin and allows the spinner values of the parameter with the raised push pins to change.

Overall Controls the height of the flight of stairs.

Riser Ht Controls the height of the risers.

Riser Ct Controls the number of risers. There will always be one more riser than steps. This *implied* riser is between the top step of the stair and the upper floor.



Linear stair with five risers

1 through 4. Risers

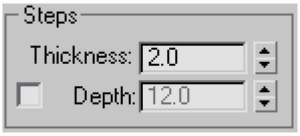
5. The implied riser

6. The upper floor you snap to

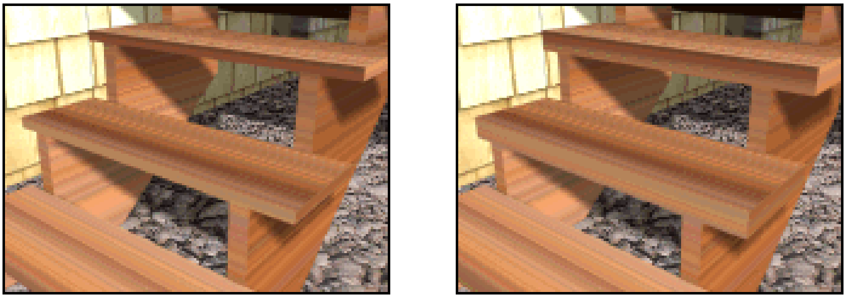
7. The lower floor you snap to

8. The steps.

Steps group

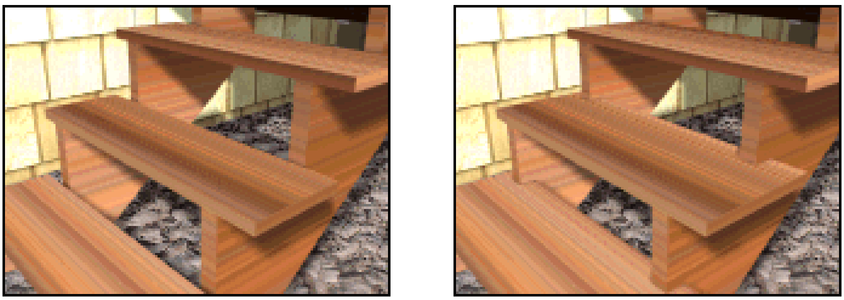


Thickness Controls the thickness of the steps.



Step thickness variance between two stairs

Depth Controls the depth of the steps.



Step depth variance between two stairs

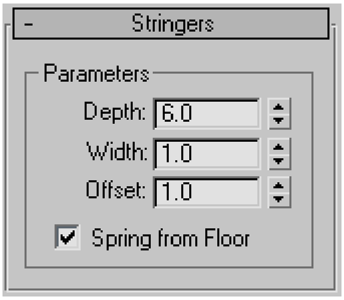
Generate Mapping Coords Applies default mapping coordinates on page 8034 to the stairs.

NOTE If a visible viewport is set to a non-wireframe or non-bounding-box display, Generate Mapping Coordinates is on for all primitives to which you apply a material containing a map with Show Map In Viewport on. If all viewports are set to wireframe or bounding box, 3ds Max turns on Generate Mapping Coordinates for primitives containing mapped materials at render time.

Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Stringers rollout

These controls are available only when you turn on Stringers on the Parameters rollout > Generate Geometry group.

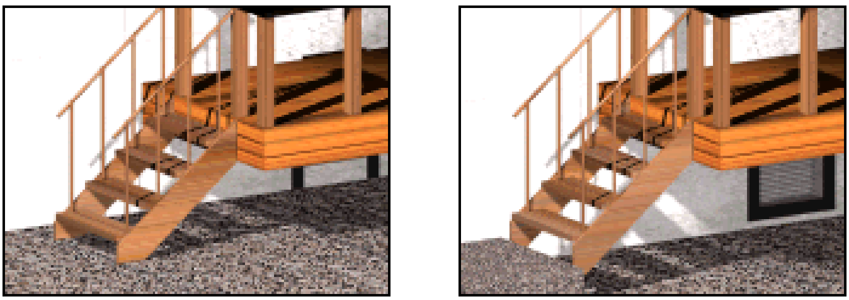


Depth Controls how far down the stringers reach toward the floor.

Width Controls the width of the stringers.

Offset Controls the vertical distance of the stringers from the floor.

Spring from Floor Controls whether the stringer starts at the floor, flush with the start of the first riser, or if the stringer extends below the floor. You control the amount the stringer extends below the floor with the Offset option.

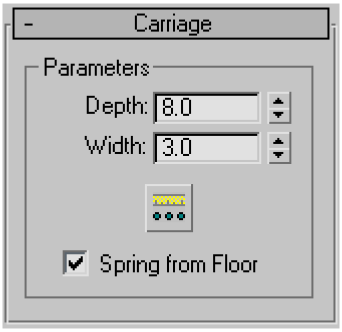


Left: The stringer extending below the floor. (Spring From Floor off.)

Right: The stringer springing from the floor. (Spring From Floor on.)

Carriage rollout

These controls are available only when you turn on Carriage on the Parameters rollout > Generate Geometry group.

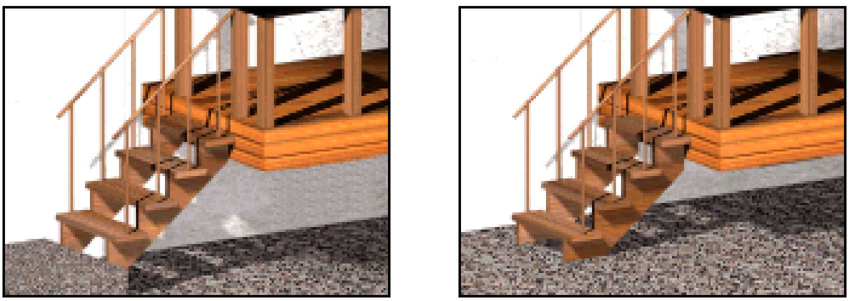


Depth Controls how far down the carriage reaches toward the floor.

Width Controls the width of the carriage.

Carriage Spacing Sets the spacing of the carriage. When you pick this button, the Carriage Spacing dialog displays. Specify the number of carriages you want using the Count option. For more information on spacing options in this dialog, see Spacing Tool on page 996.

Spring from Floor Controls whether the carriage starts at the floor, flush with the start of the first riser, or if the carriage extends below the floor. You control the amount the carriage extends below the floor with the Offset option.

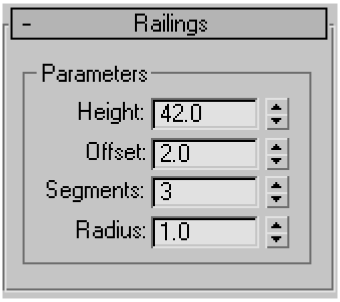


Left: The carriage springing from the floor. (Spring From Floor on.)

Right: The carriage extending below the floor. (Spring From Floor off.)

Railings rollout

These controls are available only when you turn on one or more of the Handrail or Rail Path options on the Parameters rollout > Generate Geometry group. Also, Segments and Radius aren't available if neither of the Handrail options is on.



Height Controls the height of the railings from the steps.

Offset Controls the offset of the railings from the ends of the steps.

Segments Controls the number of segments in the railings. Higher values display smoother railings.

Radius Controls the thickness of the railings.

U-Type Stair

Create panel > Geometry > Stairs > U-Type Stair button

Create menu > AEC Objects > U-Type Stair

The U-Type Stair object lets you create a two-flight staircase, with the two flights parallel to each other and a landing between them.

Types of U-type stair: open, closed, and boxed

U-type stairs have two flights in opposite directions, and a landing.

Procedures

To create U-Type stairs:

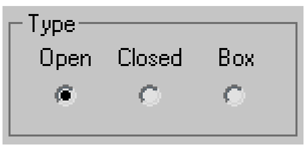
1 In any viewport, drag to set the length for the first flight. Release the mouse button, then move the cursor and click to set the width of the landing, or the distance separating the two flights.

2 Click and move the cursor up or down to define the rise of the stairs, then click to end.

3 Adjust the stairs by using the options in the Parameters rollout.

Interface

Parameters rollout > Type group

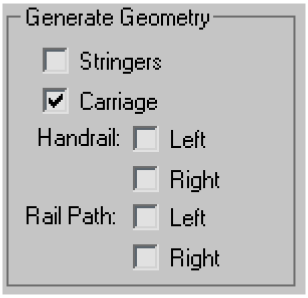


Open Creates an open riser stair as shown on the left in the illustration above.

Closed Creates a closed riser stair as shown in the center in the illustration above.

Box Creates a stair with closed risers and closed stringers on both sides as shown on the right in the illustration above.

Generate Geometry group



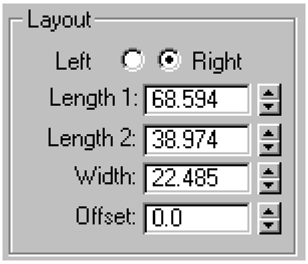
Stringers Creates stringers along the ends of the treads of the stairs. To modify the stringers' depth, width, offset and spring from the floor, see Stringers rollout on page 560.

Carriage Creates an inclined, notched beam under the treads which supports the steps or adds support between the stringers of the stairs. You might also know this as a carriage piece, a horse, or a roughstring. See Carriage rollout on page 561 to modify the parameters.

Handrail Creates left and right handrails. See Railings rollout on page 562 to modify the handrails' height, offset, number of segments, and radius.

Rail Path Creates left and right paths you can use to install railings on the stairs. See Stairs on page 532 for the instructions on how to do this.

Layout group



Left/Right Controls the position of the two flights (Length 1 and Length 2) relative to each other. If you select left, then the second flight is on the left from the landing. If you select right, then the second flight is the right from the landing.

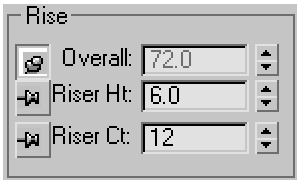
Length 1 Controls the length of the first flight of stairs.

Length 2 Controls the length of the second flight of stairs.

Width Controls the width of the stairs, including the steps and the landing.

Offset Controls the distance separating the two flights and thus the length of the landing.

Rise group

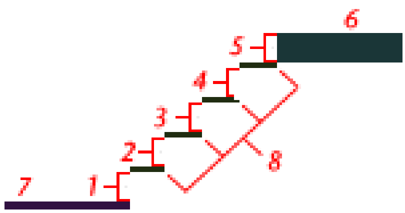


3ds Max keeps one Rise option locked while you adjust the other two. To lock an option, you click a push pin. To unlock an option you click a raised push pin. 3ds Max locks the spinner value of the parameter with the depressed push pin and allows the spinner values of the parameter with the raised push pins to change.

Overall Controls the height of the flight of stairs.

Riser Ht Controls the height of the risers.

Riser Ct Controls the number of risers. There will always be one more riser than steps. This *implied* riser is between the top step of the stair and the upper floor.



Linear stair with five risers

1 through 4. Risers

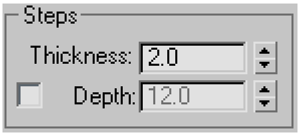
5. The implied riser

6. The upper floor you snap to

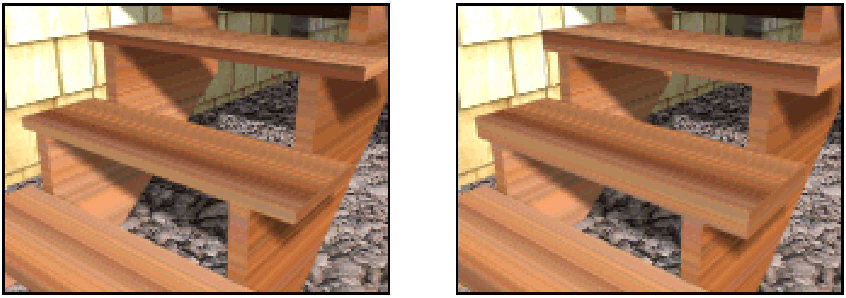
7. The lower floor you snap to

8. The steps.

Steps group

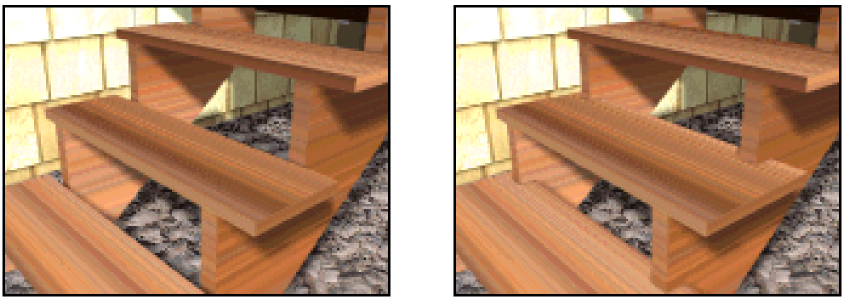


Thickness Controls the thickness of the steps.



Step thickness variance between two stairs

Depth Controls the depth of the steps.



Step depth variance between two stairs

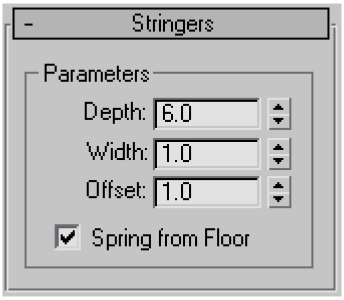
Generate Mapping Coords Applies default mapping coordinates on page 8034 to the stairs.

NOTE If a visible viewport is set to a non-wireframe or non-bounding-box display, Generate Mapping Coordinates is on for all primitives to which you apply a material containing a map with Show Map In Viewport on. If all viewports are set to wireframe or bounding box, 3ds Max turns on Generate Mapping Coordinates for primitives containing mapped materials at render time.

Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Stringers rollout

These controls are available only when you turn on Stringers on the Parameters rollout > Generate Geometry group.

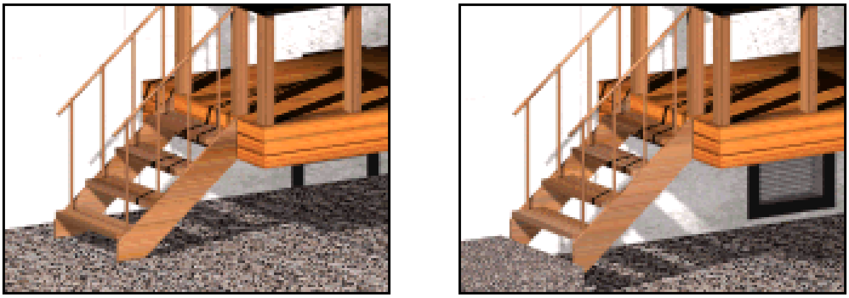


Depth Controls how far down the stringers reach toward the floor.

Width Controls the width of the stringers.

Offset Controls the vertical distance of the stringers from the floor.

Spring from Floor Controls whether the stringer starts at the floor, flush with the start of the first riser, or if the stringer extends below the floor. You control the amount the stringer extends below the floor with the Offset option.

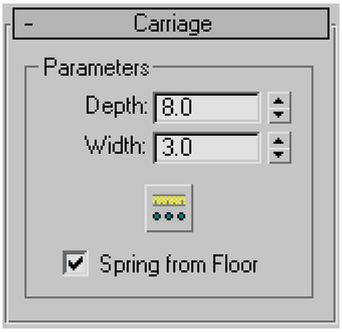


Left: The stringer extending below the floor. (Spring From Floor off.)

Right: The stringer springing from the floor. (Spring From Floor on.)

Carriage rollout

These controls are available only when you turn on Carriage on the Parameters rollout > Generate Geometry group.

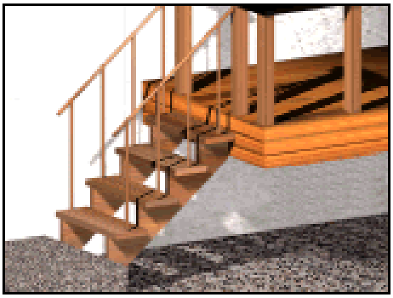


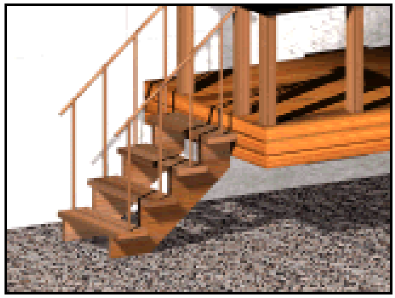
Depth Controls how far down the carriage reaches toward the floor.

Width Controls the width of the carriage.

Carriage Spacing Sets the spacing of the carriage. When you pick this button, the Carriage Spacing dialog displays. Specify the number of carriages you want using the Count option. For more information on spacing options in this dialog, see Spacing Tool on page 996.

Spring from Floor Controls whether the carriage starts at the floor, flush with the start of the first riser, or if the carriage extends below the floor. You control the amount the carriage extends below the floor with the Offset option.



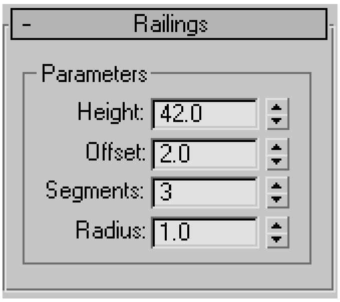


Left: The carriage springing from the floor. (Spring From Floor on.)

Right: The carriage extending below the floor. (Spring From Floor off.)

Railings rollout

These controls are available only when you turn on one or more of the Handrail or Rail Path options on the Parameters rollout > Generate Geometry group. Also, Segments and Radius aren't available if neither of the Handrail options is on.



Height Controls the height of the railings from the steps.

Offset Controls the offset of the railings from the ends of the steps.

Segments Controls the number of segments in the railings. Higher values display smoother railings.

Radius Controls the thickness of the railings.

Doors

Create panel > Geometry > Doors

Create menu > AEC Objects

The door models provided let you control details of a door's appearance. You can also set the door to be open, partially open, or closed and you can animate the opening.

Different door types in a model of a house

There are three kinds of doors. The Pivot door on page 571 is the familiar door that is hinged on one side only. The Bifold door on page 575 is hinged in the middle as well as the side, like many closet doors. You can also make these kinds of doors a set of double doors. The Sliding door on page 573 has a fixed half and a sliding half.

The topic for each kind of door describes its unique controls and behavior. Most door parameters are common to all kinds of doors, and are described here.

Doors and Materials

By default, 3ds Max assigns five different material IDs to doors. The *aectemplates.mat* material library includes *Door-Template*, a multi/sub-object material designed to be used with doors. Each component of the door/material is listed below along with its corresponding Material ID.

Material ID	Door/Material Component
1	Front
2	Back
3	Inner Bevel (used for glazing when Panels set to Glass or Beveled).
4	Frame
5	Inner Door

NOTE 3ds Max does not automatically assign a material to the door object. To use the included material, open the library and then assign the material to your object.

Making an Opening for a Door

To make an opening in a wall, you can perform a Boolean operation on page 757 with the wall as Operand A, and another object, such as a box, as Operand B. Then, you can create and add a door in the opening, and link on page 3342 it, if you choose, as a child of the wall.

NOTE Using snaps, you can insert a door in a wall object, automatically linking the two and creating a cutout for the door. See the procedure To create and place a window or door in a wall: on page 522.

Procedures

To create a door:

1 On the Object Type rollout, click the button for the type of door you want to create.

2 Choose options as needed, such as changing the default creation method. Turn off Create Frame to eliminate the door frame. Turn on Allow Non-vertical Jambs if you want an inclined door.

3. Drag the mouse in the viewport to create the first two points, defining the width and angle of the base of the door.
4. Release the mouse and move to adjust the depth of the door (default creation method), and then click to set.

   By default, the depth is perpendicular to the line between the first two points and parallel to the active grid.
5. Move the mouse to adjust the height, and then click to finish.

   The height is perpendicular to the plane defined by the first three points and perpendicular to the active grid.

   You can adjust the Height, Width, and Depth values on the Parameters rollout.

On the Creation Method rollout, you can change the creation order to width-height-depth instead of width-depth-height.

To create a door material:

1. Create a door or select an existing door.
2. Open the Material Editor, and select a slot for the material.
3. Click the Type button below the Material Editor toolbar.

   The Material/Map Browser dialog opens.
4. In the Material list, double-click the Multi/Sub-Object item, and then on the Replace Material dialog that appears, choose either option and click OK.
5. On the Multi/Sub-Object Basic Parameters rollout, click Set Number and change Number Of Materials to 5. Click OK.
6. Optionally, change the sub-material names to those specified in the above table on page 564.
7. Edit the material as you would any Multi/Sub-Object material.

To animate a door:

You can animate a door opening and closing by keyframing the Open setting.

1. Create a door or select an existing door.

   If using an existing door, also access the Modify panel.

2 Set the Parameters rollout > Open parameter to the amount you want the door to be open at the start of the animation. If you want it to be closed, set it to **0**.

3 Click the Auto Key button and advance to the first keyframe.

4 Change the Open setting.

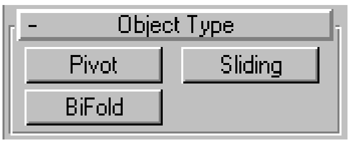
5 Continue moving to any additional keyframes and changing the Open setting as necessary.

6 Play the animation.

Interface

The topic for each kind of door describes its unique controls and behavior. Most door parameters are common to all kinds of doors, and are described here.

Object Type rollout



There are three kinds of doors in 3ds Max:

Pivot The familiar door type that is hinged on one side only. See Pivot Door on page 571.

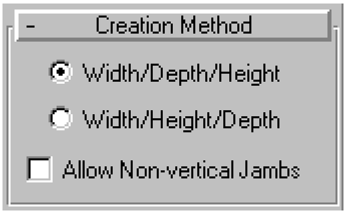
Sliding Has a fixed half and a sliding half. See Sliding Door on page 573.

BiFold Hinged in the middle as well as the side, like many closet doors. You can also use this type of door to make a set of double doors. See BiFold Door on page 575.

Name and Color rollout

See Object Name and Wireframe Color on page 7631.

Creation Method rollout



You define each type of door with four points: Drag the first two, followed by two move-click sequences. The Creation Method setting determines the order in which these actions define the door's dimensions.

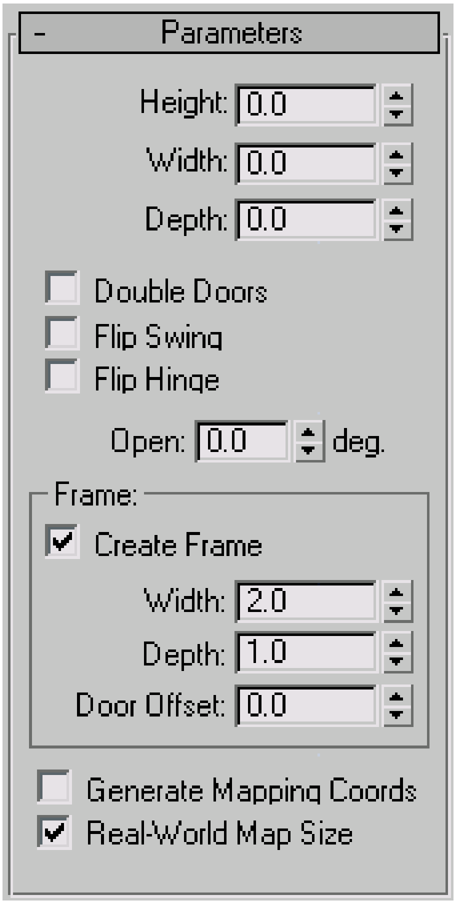
Width/Depth/Height The first two points define the width and angle of the base of the door. You set these points by dragging in a viewport, as the first step in creating a door. The first point, where you click and hold before dragging, defines a point on the jamb at the hinge for single-pivot and bifold doors (both jambs have hinges on double doors, and sliding doors have no hinge). The second point, where you release the button after dragging, specifies the width of the door, as well as the direction from one jamb to the other. This lets you align the door with a wall or opening when you place it. The third point, where you click after moving the mouse, specifies the depth of the door, and the fourth click, where you click after moving the mouse again, specifies the height.

Width/Height/Depth Works like the Width/Depth/Height option, except that the last two points create first the height and then the depth.

NOTE With this method, the depth is perpendicular to the plane set by the first three points. Thus, if you draw the door in the Top or Perspective viewport, the door lies flat on the active grid.

Allow Non-vertical Jambs Lets you create tilted doors. Set snaps on page 2652 to define points off the construction plane. Default=off.

Parameters rollout



Height Sets the overall height of the door unit.

Width Sets the overall width of the door unit.

Depth Sets the depth of the door unit.

Open With Pivot doors, specifies in degrees the extent to which the door is open. With Sliding and BiFold doors, Open specifies the percent that the door is open.

Frame group

This rollout has controls for the door-jamb frame. Though part of the door object, the frame behaves as if it were part of the wall. It doesn't move when you open or close the door.

Create Frame This is turned on as a default to display the frame. Turn this off to disable display of the frame.

Width Sets the width of the frame parallel to the wall. Available only when Create Frame is on.

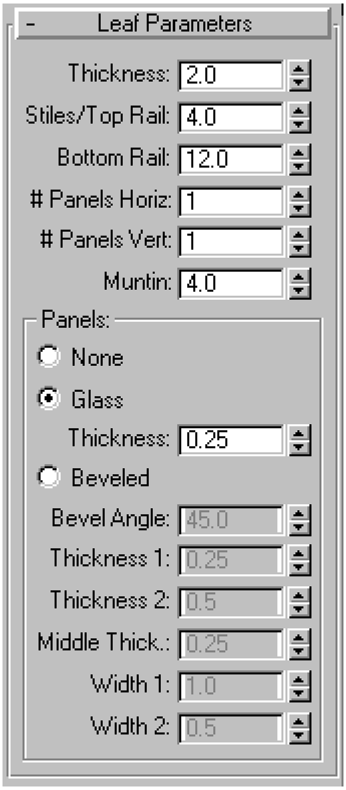
Depth Sets the depth of the frame as it projects from the wall. Available only when Create Frame is on.

Door Offset Sets the location of the door relative to the frame. At 0.0, the door is flush with one edge of the trim. Note that this can be a positive or negative value. Available only when Create Frame is on.

Generate Mapping Coords Assigns mapping coordinates to the door.

Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Leaf Parameters rollout



Provides controls that affect the door itself (as opposed to the door unit, which includes the frame). You can adjust the dimensions of the door, add panels, and adjust the dimensions and placement of those panels. The total number of panels for each door element is the number of horizontal divisions times the number of vertical divisions. Pivot doors have a single door element unless they are double doors. BiFold doors have two door elements, or four if they are double doors. Sliding doors have two door elements.

Thickness Sets the thickness of the door.

Stiles/Top Rail Sets the width of the panel framing on the top and sides. This setting is apparent only if the door is paneled.

Bottom Rail Sets the width of the panel framing at the base of the door. This setting is apparent only if the door is paneled.

 # Panels Horiz. Sets the number of panel divisions along the horizontal axis.

 # Panels Vert. Sets the number of panel divisions along the vertical axis.

Muntin Sets the width of the separations between the panels.

Panels group

Determines how panels are created in the door.

None The door has no paneling.

Glass Creates glass panels with no beveling.

Thickness Sets the thickness of the glass panels.

Beveled Choose this to have beveled panels.
The remaining spinners affect the beveling of the panels.

Bevel Angle Specifies the angle of the bevel between the outer surface of the door and the surface of the panel.

Thickness 1 Sets the outer thickness of the panel.

Thickness 2 Sets the thickness where the bevel begins.

Middle Thick. Sets the thickness of the inner part of the panel.

Width 1 Sets the width where the bevel begins.

Width 2 Sets the width of the inner part of the panel.

Pivot Door

Create panel > Geometry > Doors > Pivot button

Create menu > AEC Objects > Pivot Door

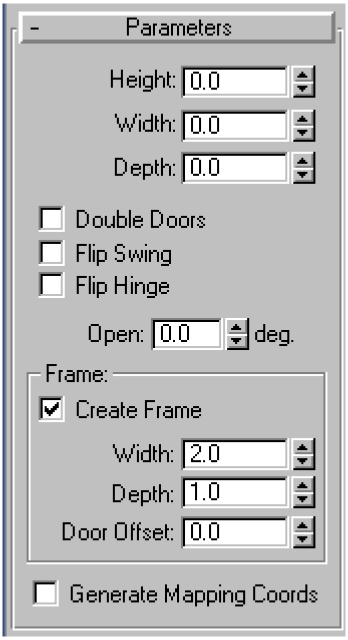
The Pivot door is hinged on one side only. You can also make the door a double door, with two door elements, each hinged on its outer edge.

Single and double pivot doors

This topic describes only controls and behavior unique to the Pivot door. Most door parameters are common to all kinds of doors; see Doors on page 562.

Interface

Parameters rollout



The Parameters rollout contains three check boxes specific to Pivot doors.

Double Doors Makes a double door.

Flip Swing Changes the direction the door swings.

Flip Hinge Places the door hinges on the opposite side of the door. This option is unavailable for double doors.

Sliding Door

Create panel > Geometry > Doors > Sliding button

Create menu > AEC Objects > Sliding Door

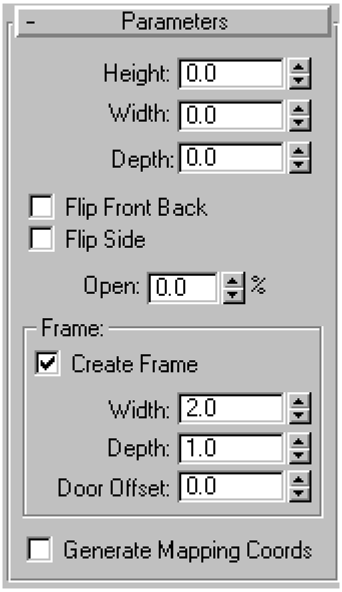
The Sliding door slides as if on a track or railing. It has two door elements: one remains stationary while the other moves.

Sliding doors with different numbers of panels

This topic describes only controls and behavior unique to the Sliding door. Most door parameters are common to all kinds of doors; see Doors on page 562.

Interface

Parameters rollout



Flip Front Back Changes which element is in front, compared to the default.

Flip Side Changes the current sliding element to the stationary element, and vice versa.

BiFold Door

Create panel > Geometry > Doors > BiFold button

Create menu > AEC Objects > BiFold Door

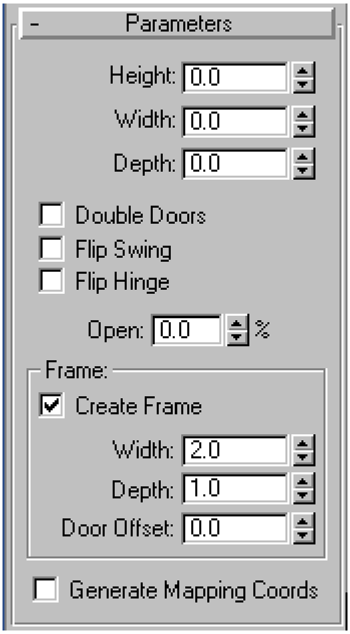
The BiFold door is hinged in the middle as well as on the side. It has two door elements. You can also make the door a double door, with four door elements.

Single and double bifold doors

This topic describes only controls and behavior unique to the BiFold door. Most door parameters are common to all kinds of doors; see Doors on page 562.

Interface

Parameters rollout



The Parameters rollout contains three check boxes specific to BiFold doors.

Double Doors Makes the door a double door, with four door elements, meeting in the center.

Flip Swing Makes the door swing in the opposite direction from the default.

Flip Hinge Makes the door hinged on the opposite side from the default. Flip Hinge is unavailable when Double Doors is on.

Windows

Create panel > Geometry > Windows

Create menu > AEC Objects

The window object lets you control details of a window's appearance. You can also set the window to be open, partially open, or closed, and you can animate the opening over time.

Different types of windows in a model of a house

3ds Max offers six kinds of windows:

- The Casement window on page 587 has one or two door-like sashes that swing inward or outward.
- The Pivoted window on page 591 pivots at the center of its sash, either vertically or horizontally.
- The Projected window on page 593 has three sashes, two of which open like awnings in opposite directions.
- The Sliding window on page 595 has two sashes, one of which slides either vertically or horizontally.
- The Fixed window on page 589 doesn't open.
- The Awning window on page 584 has a sash that is hinged at the top.

Windows and Materials

By default, 3ds Max assigns five different material IDs to windows. The *aectemplates.mat* material library includes *Window-Template*, a multi/sub-object material designed to be used with windows. Each component of the window/material is listed below along with its corresponding Material ID.

Material ID	Window/Material Component
1	Front Rails
2	Back Rails
3	Panels (glazing), with 50% opacity
4	Front Frame
5	Back Frame

NOTE 3ds Max does not automatically assign a material to the window object. To use the included material, open the library and then assign the material to your object.

Making an Opening for a Window

To make an opening in a wall, you can perform a Boolean operation on page 757 with the wall as Operand A, and another object, such as a box, as Operand B. Then, you can create and add a window in the opening, and link on page 3342 it, if you choose, as a child of the wall.

NOTE Using snaps, you can insert a window in a wall object, automatically linking the two and creating a cutout for the window. See the procedure To create and place a window or door in a wall: on page 522.

Procedures

To create a window:

1 On the Object Type rollout, click the button for the type of window you want to create.

2 Choose options as needed, such as changing the default creation method. Turn on Allow Non-vertical Jambs if you want an inclined window.

3 Drag the mouse in the viewport to create the first two points, defining the width and angle of the base of the window.

4 Release the mouse and move to adjust the depth of the window (default creation method), and then click to set.

   By default, the depth is perpendicular to the line between the first two points and parallel to the active grid.

5 Move the mouse to adjust the height, and then click to finish.

   The height is perpendicular to the plane defined by the first three points and perpendicular to the active grid.

   You can adjust the height, width, and depth values on the Parameters rollout.

In the Creation Method rollout, you can change the creation order to width-height-depth instead of width-depth-height.

To create a window material:

1 Create a window or select an existing window.

2 Open the Material Editor, and select a slot for the material.

3 Click the Type button below the Material Editor toolbar.

   The Material/Map Browser dialog opens.

4 In the Material list, double-click the Multi/Sub-Object item, and then on the Replace Material dialog that appears, choose either option and click OK.

5 On the Multi/Sub-Object Basic Parameters rollout, click Set Number and change Number Of Materials to 5. Click OK.

6 Optionally, change the sub-material names to those specified in the above table on page 579.

7 Edit the material as you would any Multi/Sub-Object material.

To animate a window:

You can animate a window opening and closing by keyframing the Open setting.

1 Create a window or select an existing window.

2 If using an existing window, also access the Modify panel.

3 Set the Parameters rollout > Open parameter to the amount you want the window to be open at the start of the animation. If you want it to be closed, set it to **0**.

4 Click the Auto Key button on page 7549 to turn it on, and advance to the first keyframe.

5 Change the Open setting.

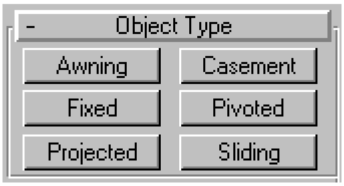
6 Continue moving to any additional keyframes and changing the Open setting as necessary.

7 Play the animation.

Interface

Most window parameters are common to all kinds of windows, and are described here. The topic for each window type describes its unique controls and behavior.

Object Type rollout



Six types of window are available in 3ds Max:

Awning Has a sash that is hinged at the top. See Awning on page 584.

Casement Has one or two door-like sashes that swing inward or outward. See Casement on page 587.

Fixed Doesn't open. See Fixed on page 589.

Pivoted Pivots at the center of its sash, either vertically or horizontally. See Pivoted on page 591.

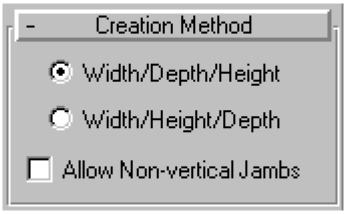
Projected Has three sashes, two of which open like awnings in opposite directions. See Projected on page 593.

Sliding Has two sashes, one of which slides vertically or horizontally. See Sliding on page 595.

Name and Color rollout

See Object Name and Wireframe Color on page 7631.

Creation Method rollout



You define each type of window with four points: Drag the first two, followed by two move-click sequences. The Creation Method setting determines the order in which these actions define the window's dimensions.

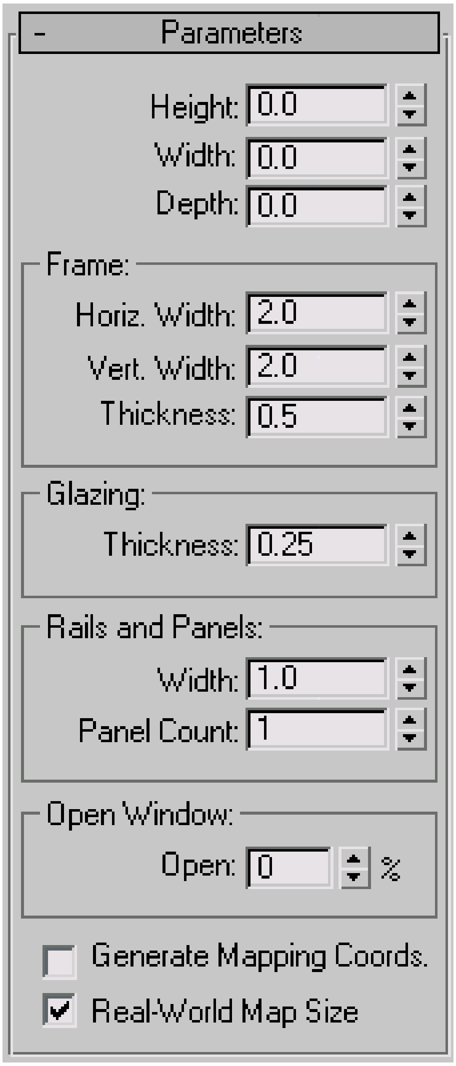
Width/Depth/Height The first two points define the width and angle of the base of the window. You set these points by dragging in a viewport, as the first step in creating a window. This lets you align the window with a wall or opening when you place it. The third point, where you click after moving the mouse, specifies the depth of the window, and the fourth click, where you click after moving the mouse again, specifies the height.

Width/Height/Depth Works like the Width/Depth/Height option, except that the last two points create first the height and then the depth.

NOTE With this method, the depth is perpendicular to the plane set by the first three points. Thus, if you draw the window in the Top or Perspective viewport, the door lies flat on the active grid.

Allow Non-vertical Jambs Select to create tilted windows. Set snaps on page 2652 to define points off the construction plane. Default=off.

Parameters rollout



Height/Width/Depth Specifies the overall dimensions of the window.

Frame group

Horiz. Width Sets the width of the horizontal part of the window frame (at the top and bottom). This setting also affects the glazed portion of the window's width.

Vert. Width Sets the width of the vertical part of the window frame (at the sides). This setting also affects the glazed portion of the window's height.

Thickness Sets the thickness of the frame. This also controls the thickness of casements or railings on the window's sashes.

Glazing group

Thickness Specifies the thickness of the glass.

Generate Mapping Coordinates Creates the object with the appropriate mapping coordinates on page 8034 already applied.

Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Awning Window

Create panel > Geometry > Windows > Awning button

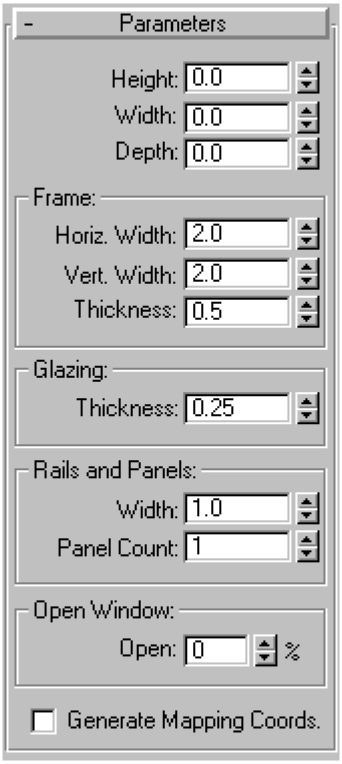
Create menu > AEC Objects > Awning Window

The Awning window has one or more sashes that are hinged at the top.

Awning window

Interface

Parameters rollout



The topic for each kind of window describes its unique controls and behavior. Some window parameters are common to all kinds of windows; see Windows on page 577.

Rails and Panels group

Width Sets the width (depth) of the rails in the sashes.

Panel Count Sets the number of sashes in the window. If you use more than one sash, each is hinged at its top edge. Range=1 to 10.

Open Window group

Open Specifies the percent the window is open. This control is animatable.

Casement Window

Create panel > Geometry > Windows > Casement button

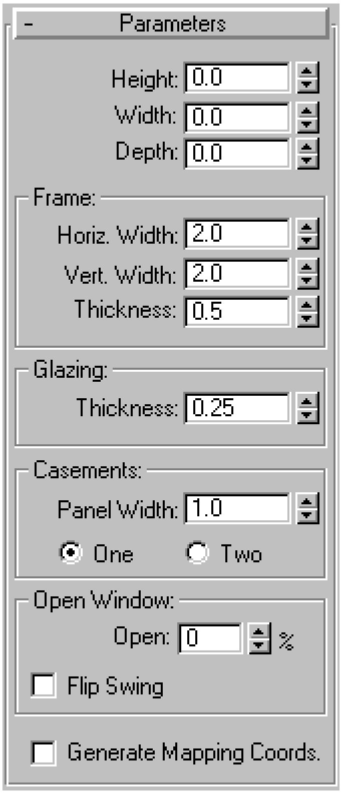
Create menu > AEC Objects > Casement Window

The Casement window has one or two sashes that are hinged on the side, like a door.

Casement window

Interface

Parameters rollout



The topic for each kind of window describes its unique controls and behavior. Some window parameters are common to all kinds of windows; see Windows on page 577.

Casements group

Panel Width Changes the size of the glazed panel within each sash.

One/Two Specifies the number of window panels: one or two. Using two panels creates a window like a double door; each panel is hinged on its outside side edge.

Open Window group

Open Specifies the percent that the window is open. This control is animatable.

Flip Swing Turn this on to have the sashes open in the opposite direction.

Fixed Window

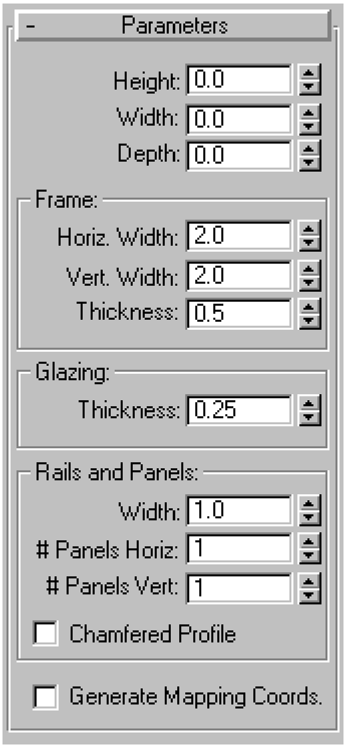
Create panel > Geometry > Windows > Fixed button

Create menu > AEC Objects > Fixed Window

Fixed windows do not open, thus have no Open Window control. In addition to the standard window object parameters, the Fixed window provides the Rails And Panels group of settings for subdividing the window.

Fixed windows

Interface



Parameters rollout

The topic for each kind of window describes its unique controls and behavior. Some window parameters are common to all kinds of windows; see Windows on page 577.

Rails and Panels group

Width Sets the width (depth) of the rails in the sashes.

 # Panels Horiz Sets the number of horizontal divisions in the window.

 # Panels Vert Sets the number of vertical divisions in the window.

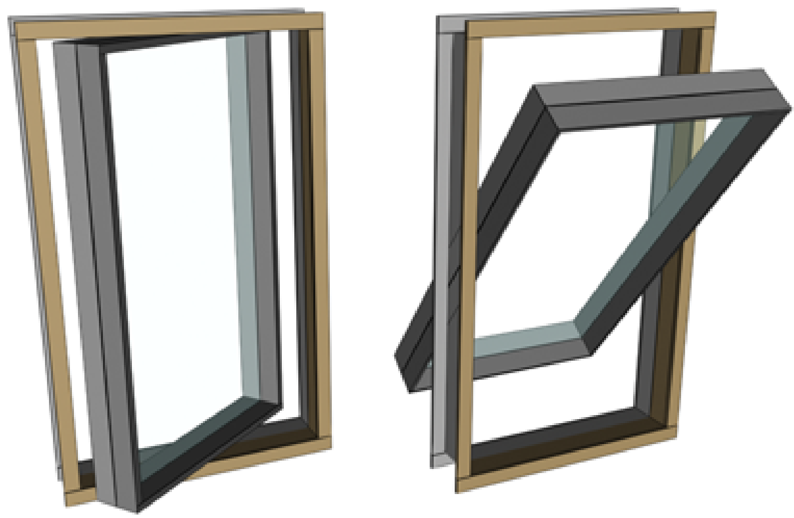
Chamfered Profile Chamfers the rails between the glazed panels, as in a conventional wooden window. When Chamfered Profile is off, the rails have a rectangular profile.

Pivoted Window

Create panel > Geometry > Windows > Pivoted button

Create menu > AEC Objects > Pivoted Window

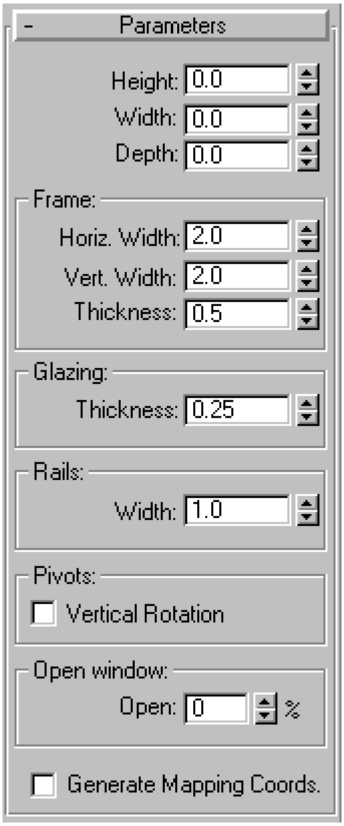
A pivoted window has one sash only, hinged midway through the side of the sash. It can swing open either vertically or horizontally.



Pivoted windows

Interface

Parameters rollout



The topic for each kind of Window describes its unique controls and behavior. Most Window parameters are common to all kinds of Windows; see Windows on page 577.

Rails group

Width Sets the width of the rails in the sash.

Pivots group

Vertical Rotation Switches the pivot axis from horizontal to vertical.

Open Window group

Open Specifies the percent that the window is open. This control is animatable.

Projected Window

Create panel > Geometry > Windows > Projected button

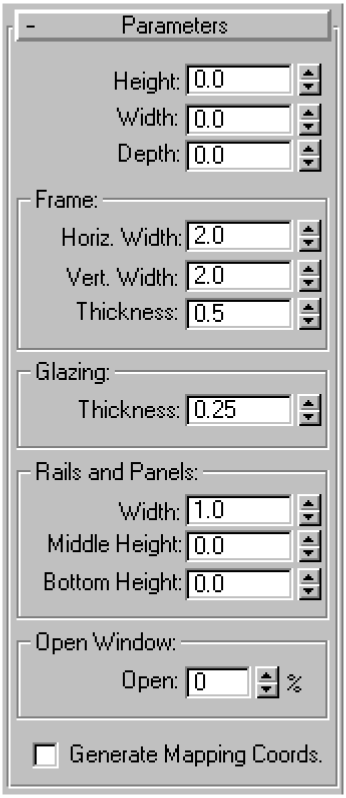
Create menu > AEC Objects > Projected Window

Projected windows have three sashes: The top sash doesn't move, while the bottom two sashes swing open like awning windows, but in opposite directions.

Projected window

Interface

Parameters rollout



The topic for each kind of window describes its unique controls and behavior. Some window parameters are common to all kinds of windows; see Windows on page 577.

Rails and Panels group

Width Sets the width (depth) of the rails in the sashes.

Middle Height Sets the height of the middle sash, relative to the window's frame.

Bottom Height Sets the height of the bottom sash, relative to the window's frame.

Open Window group

Open Specifies the percent that the two movable sashes are open. This control is animatable.

Sliding Window

Create panel > Geometry > Windows > Sliding button

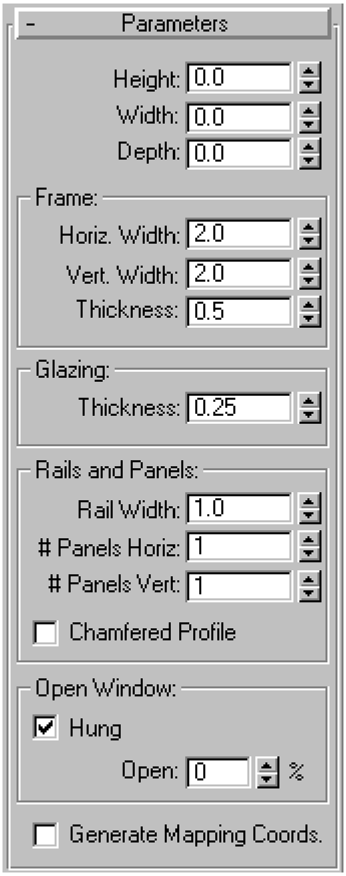
Create menu > AEC Objects > Sliding Window

Sliding windows have two sashes: one fixed, one movable. The sliding part can move either vertically or horizontally.

Sliding windows

Interface

Parameters rollout



The topic for each kind of Window describes its unique controls and behavior. Most Window parameters are common to all kinds of Windows; see Windows on page 577.

Rails and Panels group

Rail Width Sets the width of the rails in the sash.

 # Panels Horiz Sets the number of horizontal divisions in each sash.

 # Panels Vert Sets the number of vertical divisions in each sash.

Chamfered Profile Chamfers the rails between the glazed panels, as in a conventional wooden window. When Chamfered Profile is off, the rails have a rectangular profile.

Open Window group

Hung When on, the window slides vertically. When off, the window slides horizontally.

Open Specifies the percent that the window is open. This control is animatable.

mental ray Object

The mental ray object category contains one object type: the mr Proxy object. Use this object to lighten the rendering load with geometry-heavy scenes.

mr Proxy Object

Set mental ray as the current renderer. > Create panel > Geometry button > mental ray > Object Type rollout > mr Proxy button

The mr Proxy object is intended for use in large scenes to be rendered with mental ray. This object type is useful when scenes contain many instances of an object, such as an auditorium with hundreds or thousands of instances of a seat model. It is also particularly useful for objects with extremely high polygon counts, in that it obviates both the conversion to mental ray format and the presence of the source object at render time, thus saving time and freeing up a great deal of memory for rendering. The only drawbacks are the reduced fidelity of the proxy object in the viewports and the inability to edit proxies directly.

The basic usage of the mr Proxy object is that you create and optionally animate the “source” object, apply a material, and then save a copy to disk for possible future modification. Next, you convert the object to mr Proxy format and write the proxy to a disk file or series of files, after which it appears in the viewports as a point cloud (a set of vertices showing the object size and approximate shape). Finally, you delete the source object, as its presence in the scene is no longer necessary. You can then use the mr Proxy object like any object in the scene, instancing it, transforming it, and so on. When you render, mental ray loads the disk files and uses the geometry defined therein.

However, if you need to modify it (such as changing a modifier parameter, or editing a sub-object), you need to load the source object, modify it, reconnect it to the proxy object, and rewrite the files.

NOTE Using Select Similar on page 250 with an mr Proxy object selects all mr Proxy objects, even if they use different source objects.

Procedures

To use the mr Proxy object:

1. Make sure mental ray is the active renderer.
2. Create or load the object that is to serve as the source object. Apply any necessary modifiers and material. Be sure to save a copy for future reference.
3. Add an mr Proxy object.

   Until you define the source object, the mr Proxy object appears in the scene as a wireframe cube.
4. Go to the Modify panel.

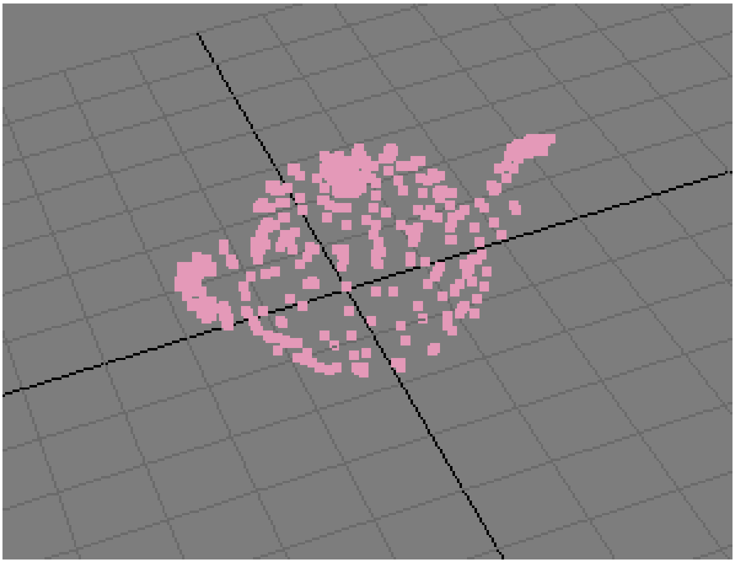
   The mr Proxy object requires that you specify the source object from the Modify panel.
5. Click the source object button, which currently reads “None,” and then select the source object.

   The name of the source object appears on the button.

   TIP If you plan to modify the source object, do so before converting it to mr Proxy format. Because the mr Proxy geometry is loaded by mental ray at render time, the renderer ignores any modifiers that change the geometry. The one exception is the Skew modifier on page 1668, which affects the object’s transformation matrix rather than its sub-object geometry, and thus can modify a proxy.
6. Click the Write Object To File button, enter a file name, and click Save.

   This opens the mr Proxy Creation dialog, which lets you set parameters for the proxy object file, including animation frames and preview settings. Change settings as necessary and then click OK to continue.

   The file is saved in the MIB format and its path and file name are placed in the Proxy File field. After you save the file, the Display group shows the proxy geometry and the viewport shows the object, by default, as a *point cloud*: a group of vertices that roughly defines the object's shape. You can change the number of points in the cloud and use a bounding box to represent the object, with or without the point cloud. (When the point cloud is on, you can toggle the bounding box, but when you turn off the point cloud, the bounding box always displays.)



mr Proxy object represented as a point cloud

If you like, you can now delete the original object.

TIP To select an mr Proxy object when represented as a point cloud, click one of the vertices.

---

7 If the object is difficult to visualize in the viewports, increase the Display group > Viewport Verts value.

You can now use this object as any other object in 3ds Max, applying materials, copying it, animating with it, and so on.

To use materials with the mr Proxy object:

When you convert an object to mr Proxy format, the proxy does not inherit the object's material. An efficient way to handle this is with the XRef material.

1 Create or load the object that is to serve as the source object. Apply any necessary modifiers and material, and then save a copy.

2 Create the mr Proxy object and then convert the source object, as described in the preceding procedure. Delete the source object.

3 Apply an XRef material on page 5765 to the proxy object.

4 Set the material to use the material from the source object file you saved in step 1.

   Then, to modify the material on the proxy, load the source object, edit its material, and save the file. Because the material on the proxy object is externally referenced, it updates automatically.

Example: To use an animated source object:

The mr Proxy object supports vertex-level animation as well as topological changes in the source object. For example, you could create a fluid simulation with a particle system to which BlobMesh on page 745 is applied, and then bake it out to a series of mr Proxy files. This procedure gives an example of how to use this feature.

1 Create a source object and add vertex-level animation. For example, you could apply the Bend modifier and then set keys for different values of the Angle parameter.

2 Add an mr Proxy object.

3 Go to the Modify panel, click the source object (“None”) button and then select the object from step 1.

4 Click the Write Object To File button, enter a file name, and click Save.

   On the mr Proxy Creation dialog, choose either animation option (Active Time Segment or Custom Range) and, if you like, toggle the Preview Generation switches. Click OK to continue.

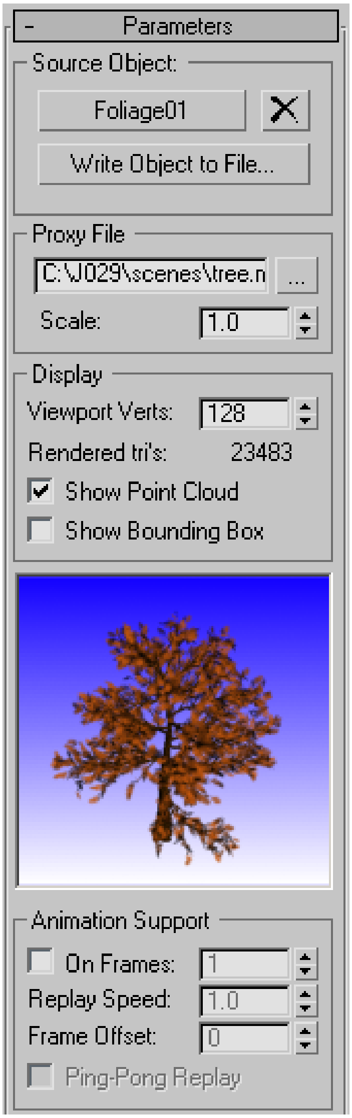
   The software writes a pair of files (geometry and thumbnail) for each animation frame.

5 You control animation playback in the Proxy object with the Animation Support group settings. When you save or load an animated Proxy, the software automatically enables the On check box in this group and sets Frames to the number of frames in the animation. You can change the value to the number of frames you want to use from the animation. Also, you can adjust the rate at which the animation appears in the proxy object by adjusting the Replay Speed value, and change the frame at which the playback begins with the Frame Offset parameter. Last, to have the animation play back and forth, turn on Ping-Pong Replay.

Interface

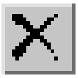
NOTE Before using the mr Proxy object, make sure mental ray is the active renderer.

Parameters rollout



Source Object group

[source object button] Shows the name of the source object, or, if you haven't assigned one, the text "None." To assign an object, click the button and then select the source object. The name of the source object then appears on the button, unless you delete the source object subsequently. In that case, the button label returns to "None," although the mr Proxy object still functions normally, assuming you've saved the object to an MIB file.

 **Clear source object slot** Restores the source object button label to "None," but doesn't otherwise affect the proxy object.

Write Object to File Lets you save the object as an MIB file, which you can then load into another mr Proxy object using the Proxy File controls.

Clicking the button opens a File Name dialog, where you can navigate to the desired folder and enter a file name. Then, when you click Save, the software opens the mr Proxy Creation dialog on page 604. Set parameters as desired and then click OK.

This saves a file named *[file name].mib*, containing the geometry data, and another named *[file name].mib.bmp*, which contains the thumbnail image that appears on the Parameters rollout.

NOTE The MIB file contains only geometry, not materials. You can apply different materials to each instance or copy of an mr Proxy object.

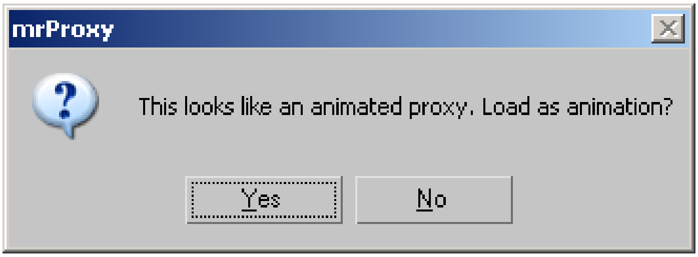
If Animation Support is on, the software writes a sequence of both types of file, with a four-digit frame number appended to the base file name for each file. For example, if you use the file name *Test* and set Animation Support group > Frames to 10, then the first pair of files are named *Test0000.mib* and *Test0000.mib.bmp*, the second pair are named *Test0001.mib* and *Test0001.mib.bmp*, and so on.

Proxy File group

Proxy File This editable field shows the location and name of the base MIB file stored with the Write Object To File command. To use a different file you can edit the field manually or click the [...] button and use the File Name dialog to choose a new file.

... [browse] Click this button to choose an MIB file to load into the proxy object. You can use this button to load an existing MIB file into a new proxy object, allowing you to easily transfer objects between different scenes.

NOTE If you load an MIB file that's part of an animated sequence, you're given the opportunity to load the entire sequence. Confirming automatically turns on animation support and sets the animation parameters to appropriate values. For details, see Animation Support group on page 603.



Scale Adjusts the size of the proxy object. Alternatively, you can use the Scale tool to resize the object.

Display group

Viewport Verts The number of vertices displayed in the point cloud for the proxy object. For best performance, display only enough vertices to make the object recognizable.

Show Point Cloud When on, the proxy object appears as a point cloud (group of vertices) in the viewports. When on, you can combine this with Show Bounding Box (see following).

NOTE When the object is selected, the point cloud always displays.

Show Bounding Box When on, the proxy object appears as a bounding box in the viewports. Available only when Show Point Cloud is on. When Show Point Cloud is off, the bounding box always displays.

[preview window] Shows the thumbnail image stored for the current frame of the MIB file. The thumbnail is generated when you click Write Object To File. The image includes the current scene background. Also, if you turn off Exclude Other Objects on the Preview Generation Options rollout, it includes other objects in the scene.

Animation Support group

These settings control animation playback; animation recording is controlled by the mr Proxy Creation settings (see following) when you save the Proxy object.

On When enabled, plays animation in the proxy object if the current MIB file is part of an animation sequence. When off, the proxy object remains in the state of the last animation frame.

Saving or loading an animated Proxy object automatically enables animation playback.

The remaining Animation Support settings are available only when animation is enabled.

Frames The number of frames to use from the animation, starting at the first frame of the animation + the Frame Offset value.

After the last frame plays, the animation repeats, either from the first frame forward again, or, if Ping-Pong Replay is on, back and forth.

Saving or loading an animated Proxy object automatically sets Frames to the correct value.

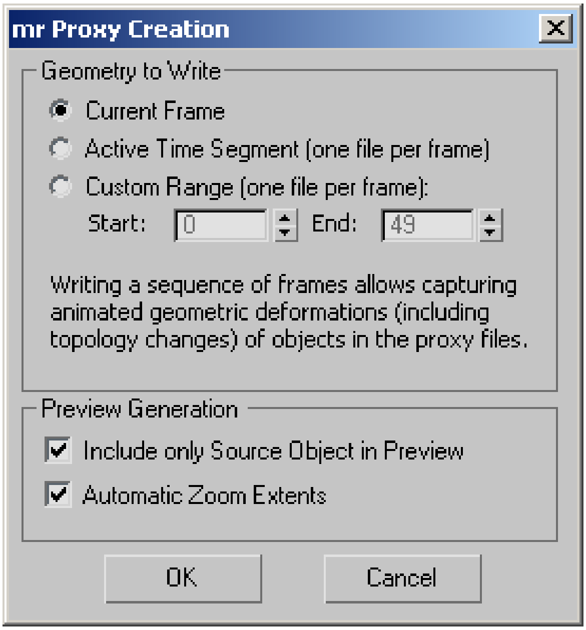
Replay Speed Lets you adjust the playback speed as a multiplier. For example, if you load a 100-frame animation and then set Replay Speed to 0.5 (half speed), the software plays back each frame twice, so the animation takes 200 frames to play back fully. Default=1.0 (full speed).

Frame Offset Lets you start the animation playback at a frame other than the first. Add this value to the start frame (see Geometry to Write on page 605) to determine at which frame the animation starts playing back.

Ping-Pong Replay When on, the animation plays forward, then in reverse, then forward again and so on. When off, it plays forward only.

mr Proxy Creation dialog

This dialog opens when you click Write Object To File on page 602 and then enter a file name and click Save. They determine whether and how animation is saved to the proxy file, and how to display the preview.



Geometry to Write Choose the option that specifies how to save deformation animation, such as that produced by modifiers, in the proxy file. Transform animation, such as rotation of the entire object, is not saved.

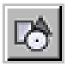
- **Current Frame** Saves no animation; only the state of the object in the current frame.
- **Active Time Segment** Saves all animation in the active time segment on page 7898.
- **Custom Range** Saves animation only in the frame range you specify with the Start and End values.

Preview Generation group

These settings determine how to display the object preview on page 603, and are saved with the proxy file.

Include only Source Object in Preview When on, the preview image contains only the geometry in the proxy object. When off, the preview shows any additional geometry in the scene within the extent of the image zoom (see Automatic Zoom Extents, following). Default=on.

Automatic Zoom Extents When on, zooms the preview image to the bounding box of the proxy object before rendering the image. Default=on.

Shapes 

Create panel > Shapes

Create menu > Shapes

A shape is an object made from one or more curved or straight lines.

3ds Max includes the following shape types:

Splines and Extended Splines on page 611

NURBS Curves on page 2284

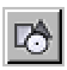
Using Shapes

Shapes are 2D and 3D lines and groups of lines that you typically use as components of other objects. Most of the default shapes are made from splines. You use these spline shapes to do the following:

- Generate planar and thin 3D surfaces
- Define loft components such as paths, shapes, and fit curves
- Generate surfaces of revolution
- Generate extrusions
- Define motion paths

The program supplies 11 basic spline shape objects, two types of NURBS curves, and five extended splines. You can quickly create these shapes using mouse or keyboard entry and combine them to form compound shapes. For information about the methods and parameters used to create these shapes, see Splines and Extended Splines on page 611 .

Creating Shapes

 To access the shape-creation tools, go to the Create panel and click the Shapes button. You'll find the standard shapes under Splines in the category

list, Point Curve and CV Curve under NURBS curves, and WRectangle, Channel, Angle, Tee, and Wide Flange under Extended Splines.

As you add plug-ins, other shape categories might appear in this list.

The Object Type rollout contains the spline creation buttons. You can combine one or more of these spline types into a single shape.

Create Shape from Edges

You can create shapes from edge selections in mesh objects. In Edit/Editable Mesh objects, at the Edge selection level, in the Edit Geometry rollout, is a button called Create Shape from Edges that creates a spline shape based on selected edges. See Editable Mesh (Edge) on page 2092. Similarly, with Editable Poly objects, you can use the Create Shape button at the Edge selection level. See Editable Poly (Edge) on page 2151

Editable Splines

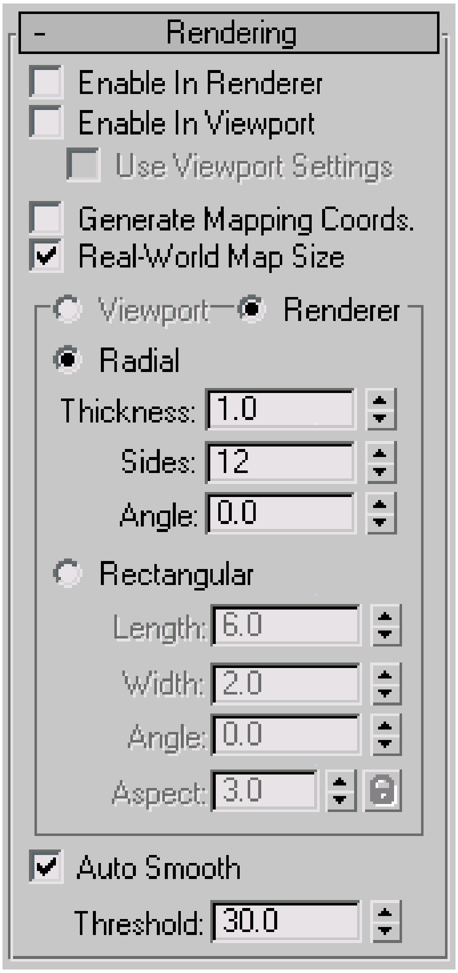
You can convert a basic spline to an editable spline object on page 659. The editable spline has a variety of controls that let you directly manipulate it and its sub-objects. For example, at the Vertex sub-object level you can move vertices or adjust their Bezier handles. Editable splines let you create shapes that are less regular, more free-form than the basic spline options.

When you convert a spline to an editable spline, you lose the ability to adjust or animate its creation parameters.

Renderable Shapes

When you use a shape to create a 3D object by lofting, extruding, or other means, the shape becomes a renderable 3D object. However, you can make a shape render without making it into a 3D object. There are three basic steps to rendering a shape:

1. On the Rendering rollout of the shape's creation parameters, turn on Enable In Renderer.
2. Specify the thickness for the spline using the Thickness spinner in the Rendering rollout.



3 If you plan to assign a mapped material to the spline, turn on Generate Mapping Coords.

When Enable in Renderer is on, the shape is rendered using a circle as a cross section. Mapping coordinates are generated with U mapped once around the perimeter, and V mapped once along the length.

The software provides more control over renderable shapes; viewports, including wireframe viewports, can display the geometry of renderable shapes. The rendering parameters for shapes appear in their own rollout.

The Steps settings affect the number of cross sections in the renderable shape.

Please observe the following:

- When you apply a modifier that converts a shape into a mesh (such as Extrude on page 1448 or Lathe on page 1501), the object automatically becomes renderable, regardless of the state of the Enable in Renderer check box. You need to turn on the Enable in Renderer check box only when you want to render an unmodified spline shape in the scene.
- As with all objects, a shape's layer must be on for the shape to render. See Layer Properties on page 7441.
- The Object Properties dialog on page 305 also has a Renderable check box, which is turned on by default. Both this check box and the General rollout > Renderable check box must be turned on in order to render a shape.

Shapes as Planar Objects

A straightforward usage for shapes is 2D cutouts or planar objects. Examples include ground planes, text for signs, and cutout billboards. You create a planar object by applying an Edit Mesh modifier on page 1353 to a closed shape, or by converting it to an editable mesh object on page 2075.

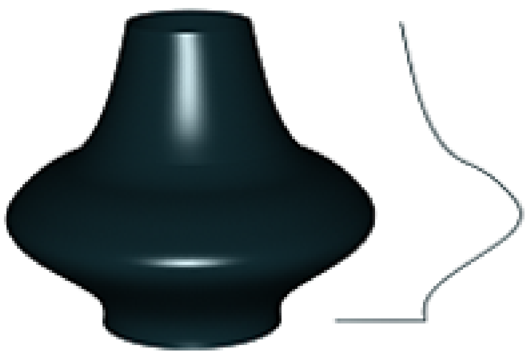
2D objects

You can also apply an Edit Mesh modifier to a 3D shape (for example, a shape whose vertices have been moved vertically away from the construction plane by different amounts) to create a curved surface. The resulting 3D surface often requires manual editing of faces and edges to smooth surface ridges.

Extruded and Lathed Shapes

You can apply modifiers to a shape to create a 3D object. Two of these modifiers are Extrude and Lathe. Extrude on page 1448 creates a 3D object by adding height to a shape. Lathe on page 1501 creates a 3D object by rotating a shape about an axis.

Initial text shape with extruded shape below



Lathed object with initial shape on right

Lofting Shapes

You create Lofts on page 786 by combining two or more splines in special ways. Shapes form the lofting path, loft cross-sections, and loft fit curves.

Shapes as Animation Paths

You can use shapes to define the position of an animated object. You create a shape and use it to define a path that some other object follows.

Some possible ways for a shape to control animated position are:

- You can use a Path constraint on page 3297 to use a shape to control object motion.
- You can convert a shape into position keys using the Motion panel > Trajectories > Convert From function (see Trajectories on page 3121).

See also:

- Edit Modifiers and Editable Objects on page 1097
- Modifying at the Sub-Object Level on page 1098
- Modifier Stack Controls on page 7635

Splines and Extended Splines

Create panel > Shapes > Splines

Create menu > Shapes

Create panel > Shapes > Extended Splines

Splines include the following object types:

Line on page 620

Rectangle on page 624

Circle on page 626

Ellipse on page 628

Arc on page 629

Donut on page 633

Extended Splines include the following object types:

This topic covers aspects of spline and extended spline creation that are common to all spline object types, including the parameters available in the General rollout. For parameters unique to a particular spline or extended spline type, see its section by clicking the appropriate link above.

Procedures

To control starting a new shape manually:

1. On the Create panel, turn off the check box next to the Start New Shape button.
2. Click the Start New Shape button.
3. Begin creating splines.

   Each spline is added to the compound shape. You can tell you are creating a compound shape because all the splines remain selected.
4. Click Start New Shape to complete the current shape and prepare to start another.

Issues to remember about creating shapes:

- You can go back and change the parameters of a shape containing a single spline after the shape is created.
- You can create a compound shape by adding splines to a shape: Select the shape, turn off Start New Shape, and then create more splines.

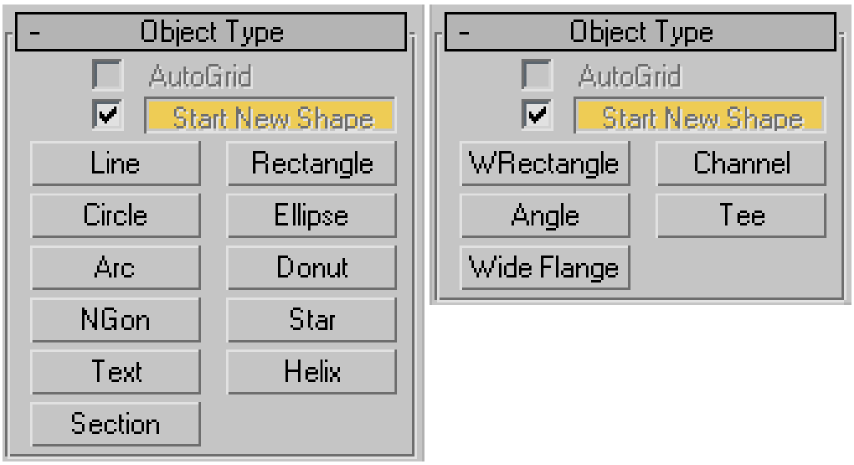
- You cannot change the parameters of a compound shape. For example, create a compound shape by creating a circle and then adding an arc. Once you create the arc, you cannot change the circle parameters.

To create a spline using keyboard entry:

1. Click a spline creation button.
2. Expand the Keyboard Entry rollout.
3. Enter X, Y, and Z values for the first point.
4. Enter values in any remaining parameter fields.
5. Click Create.

Interface

Object Type rollout (Splines and Extended Splines)



AutoGrid Lets you automatically create objects on the surface of other objects by generating and activating a temporary construction plane based on normals of the face that you click.

For more information, see AutoGrid on page 2597.

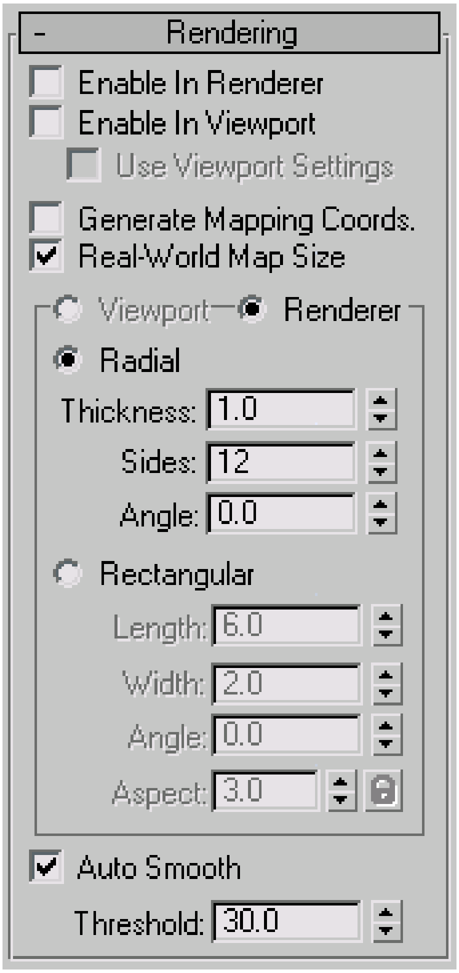
Start New Shape A shape can contain a single spline or it can be a compound shape containing multiple splines. You control how many splines are in a shape using the Start New Shape button and check box on the Object Type rollout. The check box next to the Start New Shape button determines when new shapes are created. When the box is on, the program creates a new shape object for every spline you create. When the box is off, splines are added to the current shape until you click the Start New Shape button.

Shape Selection buttons Lets you specify the type of shape to create.

Name and Color rollout

Lets you name an object and assign it a viewport color. For details, see Object Name and Wireframe Color on page 7631.

Rendering rollout



Lets you turn on and off the renderability of a spline or NURBS curve, specify its thickness in the rendered scene, and apply mapping coordinates.

You can animate render parameters, such as the number of sides, but you cannot animate the Viewport settings.

You can convert the displayed mesh into a mesh object by applying an Edit Mesh or Edit Poly modifier or converting to an editable mesh or editable poly object. If Enable In Viewport is off when converting, closed shapes will be “filled in” and open shapes will contain only vertices; no edges or faces. If

Enable In Viewport is on when converting, the system will use the Viewport settings for this mesh conversion. This gives maximum flexibility, and will always give the conversion of the mesh displayed in the viewports.

Enable In Renderer When on, the shape is rendered as a 3D mesh using the Radial or Rectangular parameters set for Renderer. In previous versions of the program, the Renderable switch performed the same operation.

Enable In Viewport When on, the shape is displayed in the viewport as a 3D mesh using the Radial or Rectangular parameters set for Renderer. In previous versions of the program, the Display Render Mesh performed the same operation.

Use Viewport settings Lets you set different rendering parameters, and displays the mesh generated by the Viewport settings. Available only when Enable in Viewport is turned on.

Generate Mapping Coords Turn this on to apply mapping coordinates. Default=off.

3ds Max generates the mapping coordinates in the U and V dimensions. The U coordinate wraps once around the spline; the V coordinate is mapped once along its length. Tiling is achieved using the Tiling parameters in the applied material. For more information, see Mapping Coordinates on page 5279.

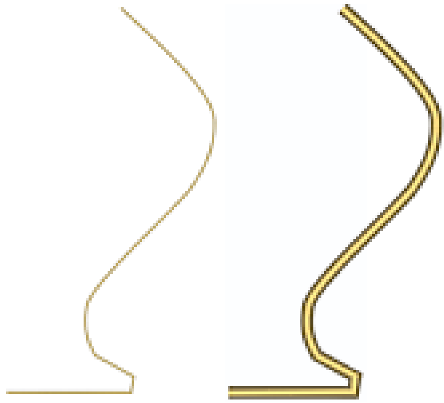
Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Viewport Choose this to specify Radial or Rectangular parameters for the shape as it will display in the viewports when Enable In Viewport is on.

Renderer Choose this on to specify Radial or Rectangular parameters for the shape as it will display when rendered or viewed in the viewport when Enable in Viewport is turned on.

Radial Displays the 3D mesh as a cylindrical object.

Thickness Specifies the diameter of the viewport or rendered spline mesh. Default=1.0. Range=0.0 to 100,000,000.0.



Splines rendered at thickness of 1.0 and 5.0, respectively

Sides Sets the number of sides (or facets) for the spline mesh n the viewport or renderer. For example, a value of 4 results in a square cross section.

Angle Adjusts the rotational position of the cross-section in the viewport or renderer. For example, if the spline mesh has a square cross section you can use Angle to position a "flat" side down.

Rectangular Displays the spline's mesh shape as a rectangle.

Length Specifies the size of the cross–section along the local *Y* axis.

Width Specifies the size of the cross–section along the local *X* axis.

Angle Adjusts the rotational position of the cross-section in the viewport or renderer. For example, if you have a square cross-section you can use Angle to position a "flat" side down.

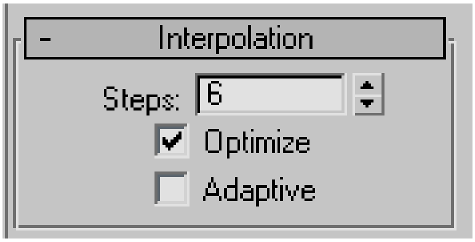
Aspect Sets the aspect ratio for rectangular cross-sections. The Lock check box lets you lock the aspect ratio. When Lock is turned on, Width is locked to Length that results in a constant ratio of Width to Length.

Auto Smooth If Auto Smooth is turned on, the spline is auto-smoothed using the threshold specified by the Threshold setting below it. Auto Smooth sets the smoothing based on the angle between spline segments. Any two adjacent segments are put in the same smoothing group if the angle between them is less than the threshold angle.

NOTE Turning Auto Smooth on for every situation does not always give you the best smoothing quality. Altering the Threshold angle may be necessary or turning Auto Smooth off may produce the best results.

Threshold Specifies the threshold angle in degrees. Any two adjacent spline segments are put in the same smoothing group if the angle between them is less than the threshold angle.

Interpolation rollout



These settings control how a spline is generated. All spline curves are divided into small straight lines that approximate the true curve. The number of divisions between each vertex on the spline are called steps. The more steps used, the smoother the curve appears.

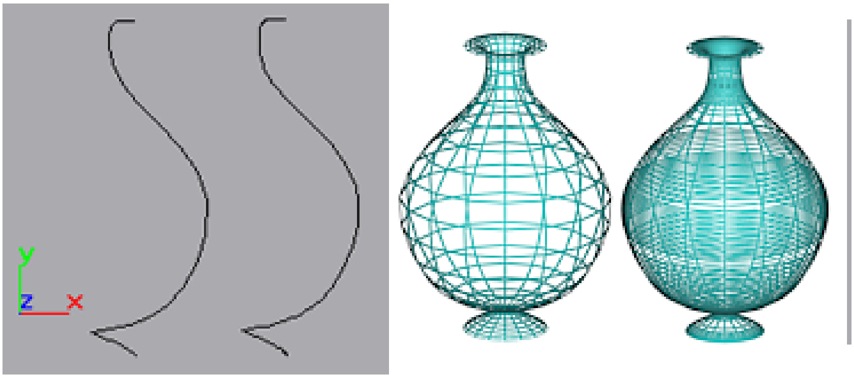
Steps Spline steps can be either adaptive (that is, set automatically by turning on Adaptive) or specified manually.

When Adaptive is off, use the Steps field/spinner to set the number of divisions between each vertex. Splines with tight curves require many steps to look smooth while gentle curves require fewer steps. Range=0 to 100.

Optimize When on, removes unneeded steps from straight segments in the spline. Optimize is not available when Adaptive is on. Default=on.

Adaptive When off, enables manual interpolation control using Optimize and Steps. Default=off.

When on, Adaptive sets the number of steps for each spline to produce a smooth curve. Straight segments always receive 0 steps.



Optimized spline left and adaptive spline right. Resulting wireframe view of each, respectively, on the right.

The main use for manual interpolation of splines is in morphing or other operations where you must have exact control over the number of vertices created.

Creation Method rollout

Many spline tools use the Creation Methods rollout. On this rollout you choose to define splines by either their center point or their diagonal.

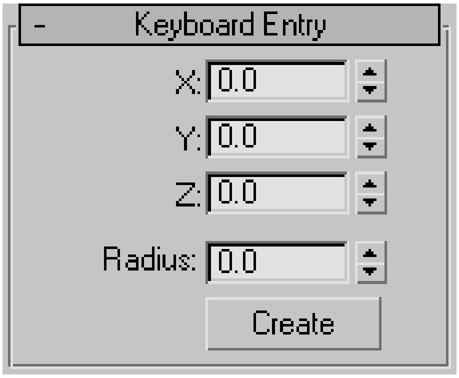
Edge Your first click defines a point on the side or at a corner of the shape and you drag a diameter or the diagonal corner.

Center Your first click defines the center of the shape and you drag a radius or corner point.

Text on page 638 and Star on page 636 do not have a Creation Methods rollout.

Line on page 620 and Arc on page 629 have unique Creation Methods rollouts that are discussed in their respective topics.

Keyboard Entry rollout



You can create most splines using keyboard entry. The process is generally the same for all splines and the parameters are found under the Keyboard Entry rollout. Keyboard entry varies primarily in the number of optional parameters. The image above shows a sample Keyboard Entry rollout for the Circle shape.

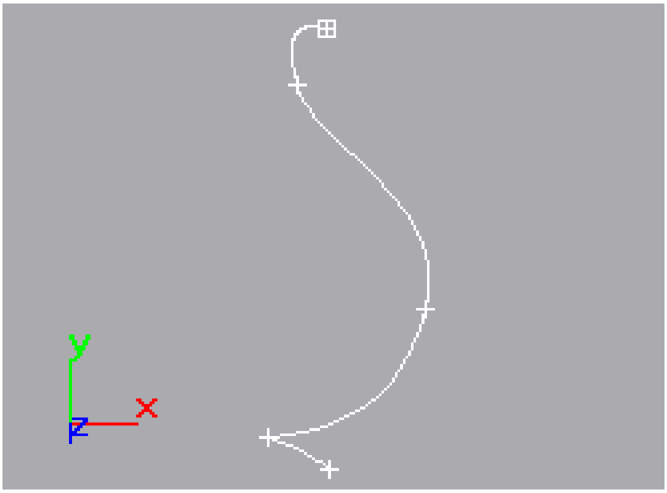
The Keyboard Entry rollout contains three fields for the X, Y, and Z coordinates of the initial creation point, plus a variable number of parameters to complete the spline. Enter values in each field and click the Create button to create the spline.

Line Spline

Create panel > Shapes > Splines > Object Type rollout > Line

Create menu > Shapes > Line

Use Line to create a free-form spline made of multiple segments.



Example of line

Procedures

To create a line:

1  Go to the Create panel and choose Shapes.

2 On the Object Type rollout, click the Line button.

3 Choose a creation method.

4 Click or drag the start point.

Clicking creates a corner vertex; dragging creates a Bezier vertex.

5 Click or drag additional points.

Clicking creates a corner vertex; dragging creates a Bezier vertex.

6 Do one of the following:

- Right-click to create an open spline.
- Click the first vertex and click Yes in the "Close spline?" dialog to create a closed spline.

To create a line using rectilinear and angle-snap options:

These two options aid in creating regular shapes:

- While creating a spline with the mouse, press and hold Shift to constrain new points to 90-degree-angle increments from previous points. Use the default Initial type setting of Corner and click all subsequent points to create fully rectilinear shapes.
- While creating a spline with the mouse, press and hold Ctrl to constrain new points to angle increments determined by the current Angle Snap setting on page 2607. To set this angle, go to Customize menu > Grid and Snap Settings, click the Options tab on page 2671 in the Grid and Snap Settings dialog, and change the value in the Angle (deg) field.

The angle for each new segment relates to the previous segment, so the angle snap works only after you've placed the first two spline vertices (that is, the first segment). Angle Snap need not be enabled for this feature to work.

To create a line from the keyboard:

1. Enter values in the X, Y, and Z fields to specify a vertex coordinate.
2. Click Add Point to add a vertex to the current line at the specified coordinate.
3. Repeat steps 1 and 2 for each additional vertex.
4. Do one of the following:
   - Click Finish to create an open spline.
   - Click Close to connect the current vertex to the first vertex and create a closed spline.

Interface

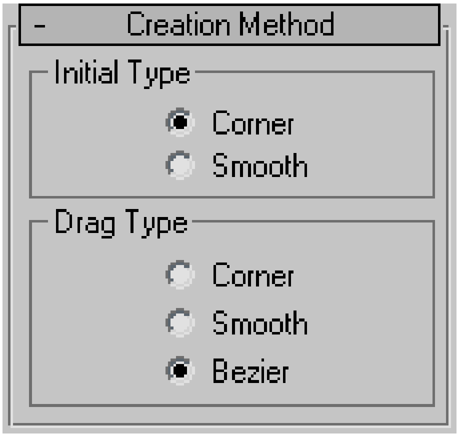
Automatic Conversion to an Editable Spline

Because the Line object has no dimension parameters to be carried over to the Modify panel, it converts to an editable spline on page 659 when you move from the Create panel to the Modify panel. While you are creating the line, the Create panel displays the original controls, such as Interpolation, Rendering, Creation Method, and Keyboard Entry. After creating the line, when you go to the Modify panel you have immediate access to the Selection and Geometry rollouts to edit the vertices or any part of the shape.

Rendering and Interpolation rollouts

All spline-based shapes share these parameters. See Splines on page 611 for an explanation of these parameters.

Creation Method rollout



Creation method options for lines are different from other spline tools. You choose options to control the type of vertex created when you click or drag vertices.

You can preset the default types of spline vertices during line creation with these settings:

Initial Type group

Sets the type of vertex you create when you click a vertex location.

Corner Produces a sharp point. The spline is linear to either side of the vertex.

Smooth Produces a smooth, nonadjustable curve through the vertex. The amount of curvature is set by the spacing of the vertices.

Drag Type group

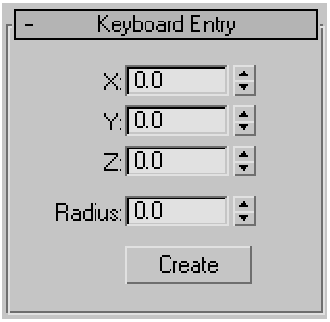
Sets the type of vertex you create when you drag a vertex location. The vertex is located at the cursor position where you first press the mouse button. The direction and distance that you drag are used only when creating Bezier vertices.

Corner Produces a sharp point. The spline is linear to either side of the vertex.

Smooth Produces a smooth, nonadjustable curve through the vertex. The amount of curvature are set by the spacing of the vertices.

Bezier Produces a smooth, adjustable curve through the vertex. The amount of curvature and direction of the curve are set by dragging the mouse at each vertex.

Keyboard Entry rollout



Keyboard entry for lines is different from keyboard entry for other splines. Entering keyboard values continues to add vertices to the existing line until you click Close or Finish.

Add Point Adds a new point to the line at the current X/Y/Z coordinates.

Close Closes the shape, adding a final spline segment between the most recent vertex and the first.

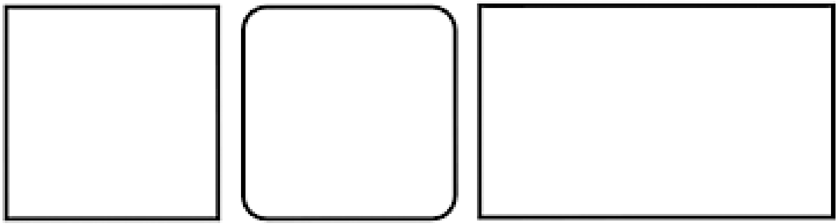
Finish Finishes the spline without closing it.

Rectangle Spline

Create panel > Shapes > Splines > Object Type rollout > Rectangle

Create menu > Shapes > Rectangle

Use Rectangle to create square and rectangular splines.



Examples of rectangles

Procedures

To create a rectangle:

1.  Go to the Create panel and choose Shapes.
2. Click Rectangle.
3. Choose a creation method.
4. Drag in a viewport to create a rectangle.
   Optionally, press Ctrl while dragging to constrain the spline to a square.

Interface

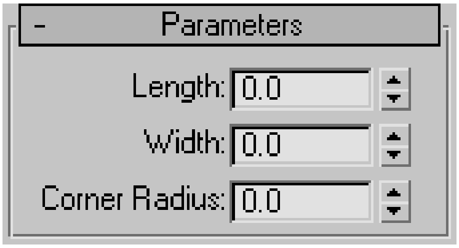
Rendering and Interpolation rollouts

All spline-based shapes share these parameters. See Splines on page 611 for an explanation of these parameters.

Creation Method rollout

The Rectangle shape uses the standard creation methods of Center or Edge. Most spline-based shapes share the same Creation Method parameters. See Splines on page 611 for an explanation of these parameters.

Parameters rollout



Once you have created a rectangle, you can make changes using the following parameters:

Length Specifies the size of the rectangle along the local Y axis.

Width Specifies the size of the rectangle along the local X axis.

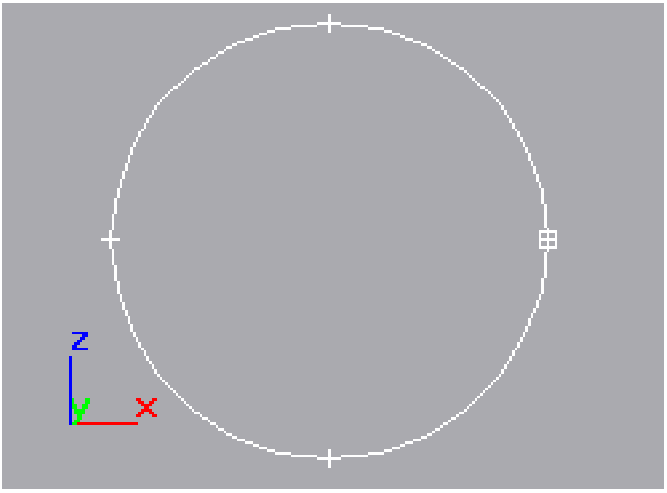
Corner Radius Creates rounded corners. When set to 0, the rectangle contains 90-degree corners.

Circle Spline

Create panel > Shapes > Splines > Object Type rollout > Circle

Create menu > Shapes > Circle

Use Circle to create closed circular splines made of four vertices.



Example of circle

Procedures

To create a circle:

1  Go to the Create panel and choose Shapes.

2 Click Circle.

3 Choose a creation method.

4 Drag in a viewport to draw the circle.

Interface

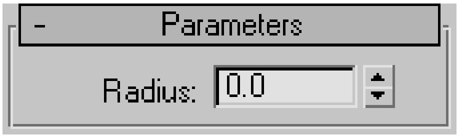
Rendering and Interpolation rollouts

All spline-based shapes share these parameters. See Splines on page 611 for explanations of these parameters.

Creation Method rollout

The Circle shape uses the standard creation methods of Center or Edge. Most spline-based shapes share the same Creation Method parameters. See Splines on page 611 for an explanation of these parameters.

Parameters rollout



Once you have created a circle, you can make changes using the following parameter:

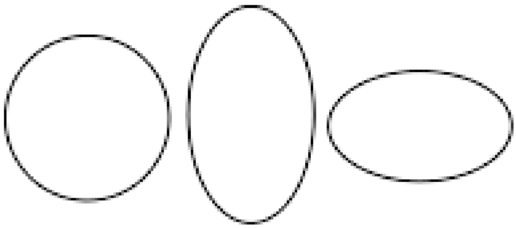
Radius Specifies the radius of the circle.

Ellipse Spline

Create panel > Shapes > Splines > Object Type rollout > Ellipse

Create menu > Shapes > Ellipse

Use Ellipse to create elliptical and circular splines.



Examples of ellipses

Procedures

To create an ellipse:

1.  Go to the Create panel and choose Shapes.
2. Click Ellipse.
3. Choose a creation method.
4. Drag in a viewport to draw the ellipse.
   Optionally, press Ctrl while dragging to constrain the spline to a circle.

Interface

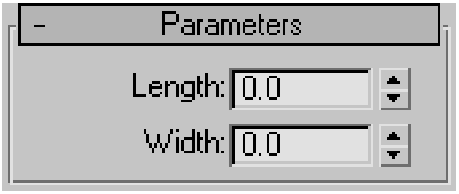
Rendering and Interpolation rollouts

All spline-based shapes share these parameters. See Splines on page 611 for explanations of these parameters.

Creation Method rollout

The Ellipse shape uses the standard creation methods of Center or Edge. Most spline-based shapes share the same Creation Method parameters. See Splines on page 611 for an explanation of these parameters.

Parameters rollout



Once you have created an Ellipse, you can make changes using the following parameters:

Length Specifies the size of the Ellipse along the local Y axis.

Width Specifies the size of the Ellipse local X axis.

Arc Spline

Create panel > Shapes > Splines > Object Type rollout > Arc

Create menu > Shapes > Arc

Use Arc to create open and closed circular arcs made of four vertices.

Procedures

To create an arc using the end-end-middle method:

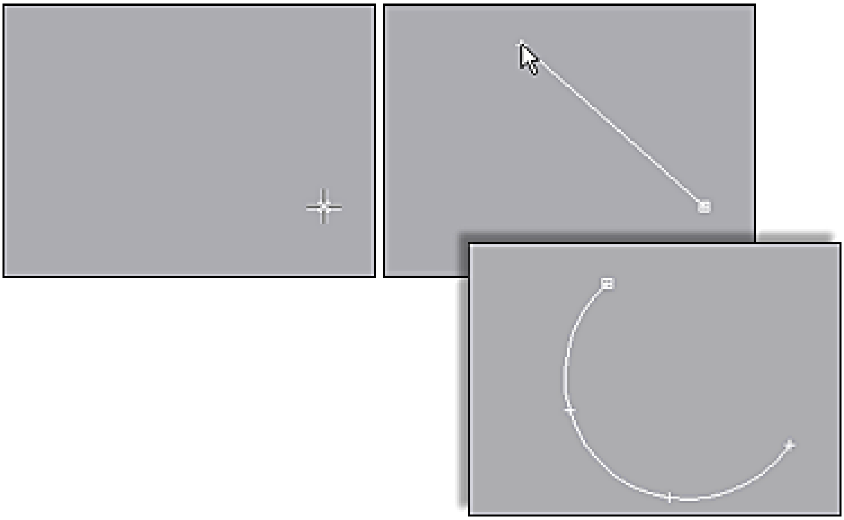
1  Go to the Create panel and choose Shapes.

2 Click Arc.

3 Choose the End-End-Middle creation method.

4 Drag in a viewport to set the two ends of the arc.

5 Release the mouse button, then move the mouse and click to specify a third point on an arc between the two endpoints.



Creating an arc using the End-End-Middle creation method

To create an arc using the center-end-end method:

1  Go to the Create panel and choose Shapes.

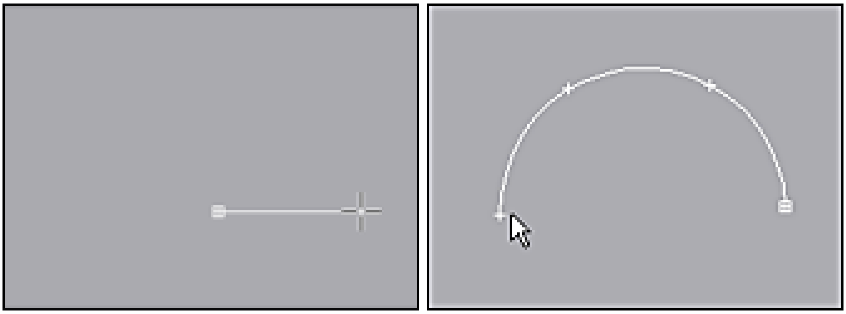
2 Click Arc.

3 Choose the Center-End-End creation method.

4 Press the mouse button to define the center of the arc.

5 Drag and release the mouse button to specify the start point of the arc.

6 Move the mouse and click to specify the other end of the arc.



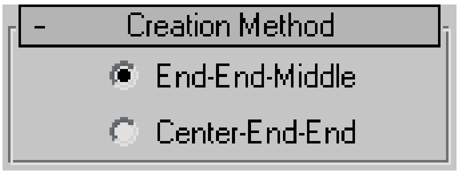
Creating an arc using the Center-End-End creation method

Interface

Rendering and Interpolation rollouts

All spline-based shapes share these parameters. See Splines on page 611 for an explanation of these parameters.

Creation Method rollout

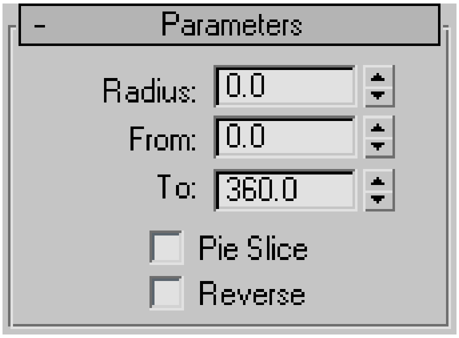


These options determine the sequence of mouse clicks involved in the creation of the arc.

End-End-Middle Drag and release to set the two endpoints of the arc, and then click to specify the third point between the two endpoints.

Center-End-End Press the mouse button to specify the center point of the arc, drag and release to specify one endpoint of the arc, and click to specify the other endpoint of the arc.

Parameters rollout



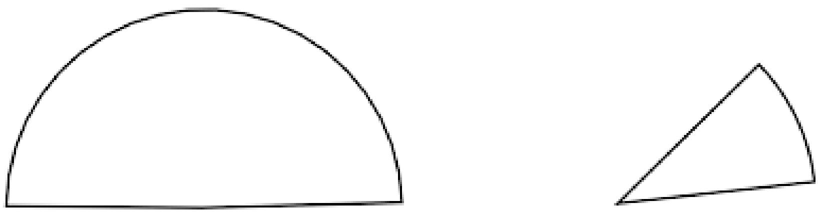
Once you have created an arc, you can make changes using the following parameters:

Radius Specifies the arc radius.

From Specifies the location of the start point as an angle measured from the local positive X axis.

To Specifies the location of the end point as an angle measured from the local positive X axis.

Pie Slice When on, creates a closed spline in the form of a pie. The start point and end point are connected to the center with straight segments.



Closed pie slice arcs

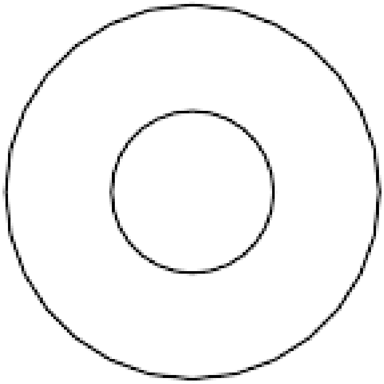
Reverse When on, the direction of the arc spline is reversed, and the first vertex is placed at the opposite end of an open arc. As long as the shape remains an original shape (and not an editable spline), you can switch its direction by toggling Reverse. Once the arc is converted to an editable spline, you can use Reverse at the Spline sub-object level to reverse direction.

Donut Spline

Create panel > Shapes > Splines > Object Type rollout > Donut

Create menu > Shapes > Donut

Use Donut to create closed shapes from two concentric circles. Each circle is made of four vertices.



Example of donut

Procedures

To create a donut:

1.   Go to the Create panel and choose Shapes.
2. Click Donut.
3. Choose a creation method.
4. Drag and release the mouse button to define the first donut circle.
5. Move the mouse and then click to define the radius of the second concentric donut circle.

   The second circle can be larger or smaller than the first.

Interface

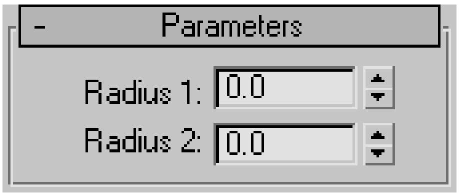
Rendering and Interpolation rollouts

All spline-based shapes share these parameters. See Splines on page 611 for explanations of these parameters.

Creation Method rollout

The Donut shape uses the standard creation methods of Center or Edge. Most spline-based shapes share the same Creation Method parameters. See Splines on page 611 for an explanation of these parameters.

Parameters rollout



Once you have created a donut, you can make changes using the following parameters:

Radius 1 Sets the radius of the first circle.

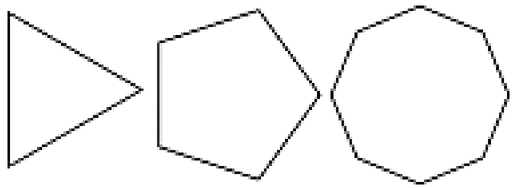
Radius 2 Sets the radius of the second circle.

NGon Spline

Create panel > Shapes > Splines > Object Type rollout > NGon

Create menu > Shapes > NGon

Use NGon to create closed flat-sided or circular splines with any number (N) of sides or vertices.



Examples of NGons

Procedures

To create an NGon:

1  Go to the Create panel and choose Shapes.

2 Click NGon.

3 Choose a creation method.

4 Drag and release the mouse button in a viewport to draw the NGon.

Interface

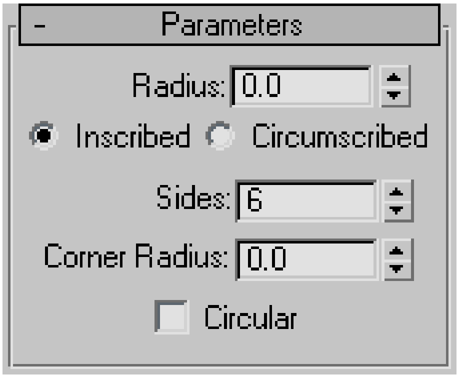
Rendering and Interpolation rollouts

All spline-based shapes share these parameters. See Splines on page 611 for an explanation of these parameters.

Creation Method rollout

The NGon shape uses the standard creation methods of Center or Edge. Most spline-based shapes share the same Creation Method parameters. See Splines on page 611 for an explanation of these parameters.

Parameters rollout



Once you have created an NGon, you can make changes using the following parameters:

Radius Specifies the NGon radius. You can use either of two methods to specify the radius:

- **Inscribed** The radius from the center to the corners of the NGon
- **Circumscribed** The radius from the center to the sides of the NGon.

Sides Specifies the number of sides and vertices used by the NGon. Range=3 to 100.

Corner Radius Specifies the degree of rounding to apply to the corners of the NGon. A setting of 0 specifies a standard unrounded corner.

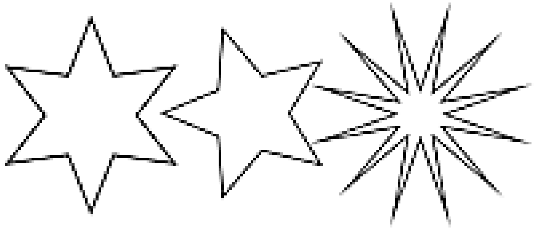
Circular When on, specifies a circular NGon.

Star Spline

Create panel > Shapes > Splines > Object Type rollout > Star

Create menu > Shapes > Star

Use Star to create closed star-shaped splines with any number of points. Star splines use two radiuses to set the distance between the outer points and inner valleys.



Examples of stars

Procedures

To create a star:

1  Go to the Create panel and choose Shapes.

2 Click Star.

3 Drag and release the mouse button to define the first star radius.

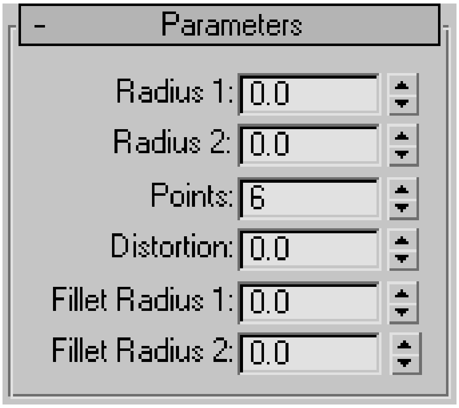
4 Move the mouse and then click to define the second star radius.

Interface

Rendering and Interpolation rollouts

All spline-based shapes share these parameters. See Splines on page 611 for explanations of these parameters.

Parameters rollout



Once you have created a star, you can make changes using the following parameters:

Radius 1 Specifies the radius of the inner vertices (the valley) of the star.

Radius 2 Specifies the radius of the outer vertices (the points) of the star.

Points Specifies the number of points on the star. Range=3 to 100.

A star has twice as many vertices as the specified number of points. Half the vertices lie on one radius, forming points, and the remaining vertices lie on the other radius, forming valleys.

Distortion Rotates the outer vertices (the points) about the center of the star. This produces a sawtooth affect.

Fillet Radius 1 Rounds the inner vertices (the valleys) of the star.

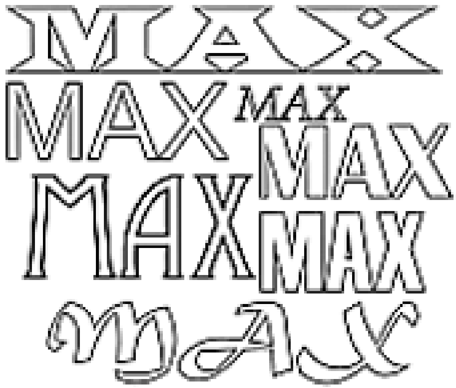
Fillet Radius 2 Rounds the outer vertices (the points) of the star.

Text Spline

Create panel > Shapes > Splines > Object Type rollout > Text

Create menu > Shapes > Text

Use Text to create splines in the shape of text. The text can use any Windows font installed on your system, or a Type 1 PostScript font installed in the directory pointed to by the Fonts path on the Configure System Paths dialog on page 7732. Because fonts are loaded only at first use, changing the font path later in the program has no effect. The program must be restarted before the new path is used, if the font manager has been used by the program.



Examples of text

You can edit the text in the Create panel, or later in the Modify panel.

Using Text Shapes

Text shapes maintain the text as an editable parameter. You can change the text at any time. If the font used by your text is deleted from the system, 3ds Max still properly displays the text shape. However, to edit the text string in the edit box you must choose an available font.

The text in your scene is just a shape where each letter and, in some cases, pieces of each letter are individual splines. You can apply modifiers like Edit Spline on page 1447, Bend on page 1208, and Extrude on page 1448 to edit Text shapes just like any other shape.

Procedures

To create text:

1  Go to the Create panel and choose Shapes.

2 Click Text.

3 Enter text in the Text box.

4 Do either of the following to define an insertion point:

- Click in a viewport to place the text in the scene.
- Drag the text into position and release the mouse button.

To enter a special Windows character:

1 Hold down the Alt key.

2 Enter the character's numeric value on the numeric keypad.

   You *must* use the numeric keypad, not the row of numbers above the alphabetic keys.

   For some characters, you must enter a leading zero. For example, 0233 to enter an e with an acute accent.

3 Release the Alt key.

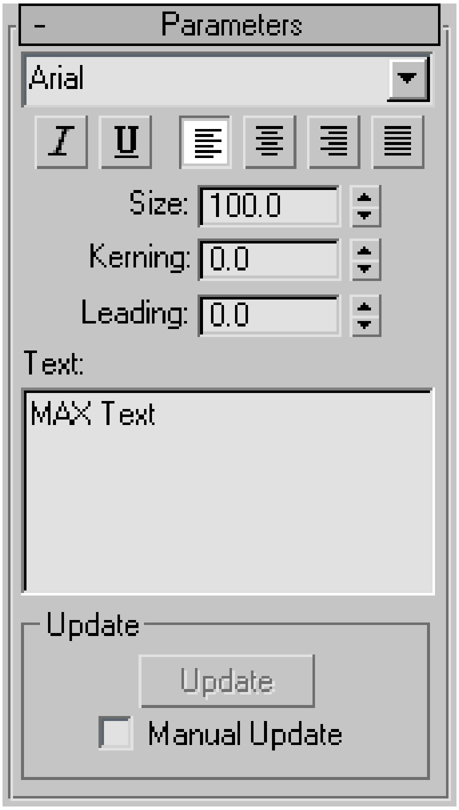
Interface

Settings available for text include kerning, leading, justification, multiple lines, and a manual update option.

Rendering and Interpolation rollouts

All spline-based shapes share these parameters. See Splines on page 611 for an explanation of these parameters.

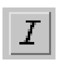
Parameters rollout

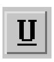


Once you have created text, you can make changes using the following parameters:

Font list Choose from a list of all available fonts. Available fonts include:

- Fonts installed in Windows.
- Type 1 PostScript fonts located in the directory pointed to by the Fonts path on the Configure System Paths dialog on page 7732.

 **Italic style button** Toggles italicized text.

 **Underline style button** Toggles underlined text.

 **Align Left** Aligns text to the left side of its bounding box.

 **Center** Aligns text to the center of its bounding box.

 **Align Right** Aligns text to the right side of its bounding box.

 **Justify** Spaces all lines of text to fill the extents of the bounding box.

NOTE The four text-alignment buttons require multiple lines of text for effect because they act on the text in relation to its bounding box. If there's only one line of text, it's the same size as its bounding box.

Size Sets the text height where the height measuring method is defined by the active font. The first time you enter text, the default size is 100 units.

Kerning Adjusts the kerning (the distance between letters).

Leading Adjusts the leading (the distance between lines). This has an effect only when multiple lines of text are included in the shape.

Text edit box Allows for multiple lines of text. Press Enter after each line of text to start the next line.

- The initial session default is "VIZ Text."
- The initial session default is "MAX Text."
- The edit box does not support word wrap.
- You can cut and paste single- and multi-line text from the Clipboard.

Update group

These options let you select a manual update option for situations where the complexity of the text shape is too high for automatic updates.

Update Updates the text in the viewport to match the current settings in the edit box. This button is available only when Manual Update is on.

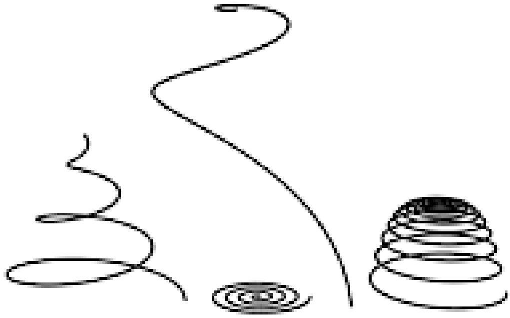
Manual Update When on, the text that you type into the edit box is not shown in the viewport until you click the Update button.

Helix Spline

Create panel > Shapes > Splines > Object Type rollout > Helix

Create menu > Shapes > Helix

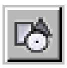
Use Helix to create open flat or 3D spirals.



Examples of helixes

Procedures

To create a helix:

1.   Go to the Create panel and choose Shapes.
2. Click Helix.
3. Choose a creation method.
4. Press the mouse button to define the first point of the Helix start circle.
5. Drag and release the mouse button to define the second point of the Helix start circle.
6. Move the mouse and then click to define the height of the Helix.
7. Move the mouse and then click to define the radius of the Helix end.

Interface

Rendering rollout

All spline-based shapes share these parameters. See Splines on page 611 for explanations of these parameters.

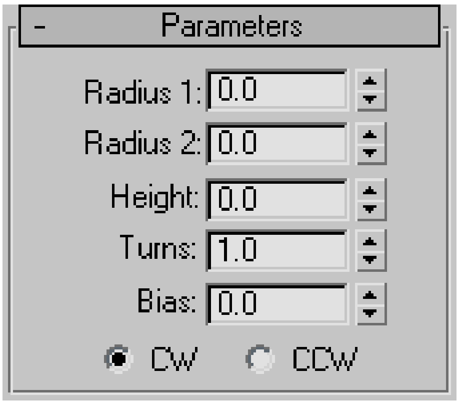
Interpolation

The helix differs from other spline-based shapes in that it always uses *adaptive interpolation*: the number of vertices in a helix is determined by the number of turns.

Creation Method rollout

The Helix shape uses the standard creation methods of Center or Edge. Most spline-based shapes share the same Creation Method parameters. See Splines on page 611 for an explanation of these parameters.

Parameters rollout



Once you have created a helix, you can make changes using the following parameters:

Radius 1 Specifies the radius for the Helix start.

Radius 2 Specifies the radius for the Helix end.

Height Specifies the height of the Helix.

Turns Specifies the number of turns the Helix makes between its start and end points.

Bias Forces the turns to accumulate at one end of the helix. Bias has no visible affect when the height is 0.0.

Helical spline varied only by bias settings

- A bias of –1.0 forces the turns toward the start of the helix.
- A bias of 0.0 evenly distributes the turns between the ends.
- A bias of 1.0 forces the turns toward the end of the helix.

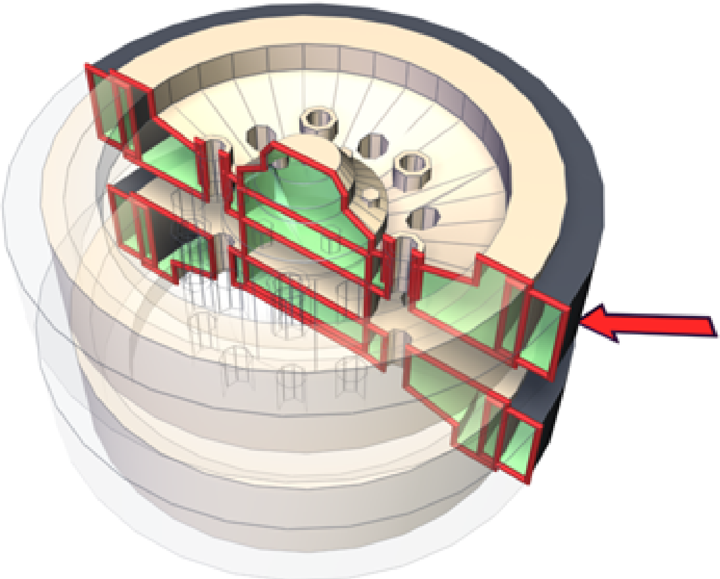
CW/CCW The direction buttons set whether the Helix turns clockwise (CW) or counterclockwise (CCW).

Section Spline

Create panel > Shapes > Splines > Object Type rollout > Section

Create menu > Shapes > Section

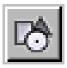
This is a special type of object that generates other shapes based on a cross-sectional slice through mesh objects. The Section object appears as a bisected rectangle. You simply move and rotate it to slice through one or more mesh objects, and then click the Create Shape button to generate a shape based on the 2D intersection.



Red line shows the section shape based on the structure.

Procedures

To create and use a section shape:

1   Go to the Create panel and choose Shapes.

2 Click Section.

3 Drag a rectangle in the viewport in which you want to orient the plane. (For example, create it in the Top viewport to place the Section object parallel with the XY home grid.)

   The Section object appears as a simple rectangle with crossed lines indicating its center. With the default settings, the rectangle is for display purposes only, because the effect of the Section object extends along its plane to the full extents of the scene.

4 Move and rotate the section so that its plane intersects mesh objects in the scene.

   Yellow lines are displayed where the sectional plane intersects objects.

5 On the Create panel, click Create Shape, enter a name in the resulting dialog, and click OK.

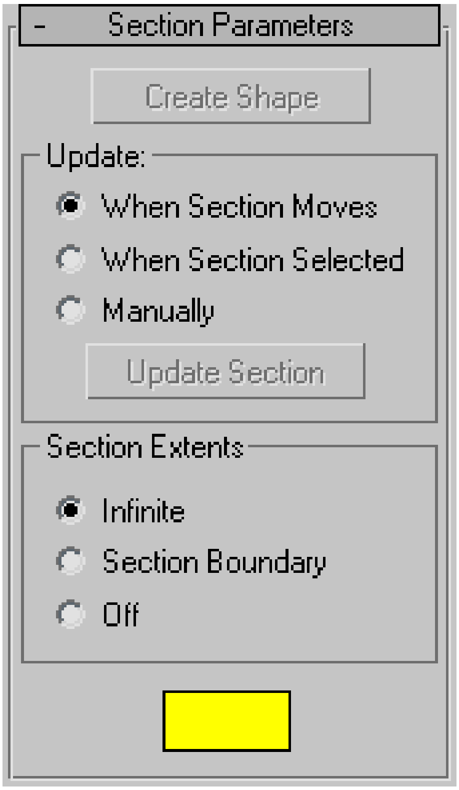
An editable spline on page 659 is created, based on the displayed cross sections.

Interface

Rendering and Interpolation rollouts

All spline-based shapes share these parameters. See Splines on page 611 for an explanation of these parameters.

Section Parameters rollout



Create Shape Creates a shape based on the currently displayed intersection lines. A dialog is displayed in which you can name the new object. The resulting shape is an editable spline consisting of curve segments and corner vertices, based on all intersected meshes in the scene.

Update group

Provides options for specifying when the intersection line is updated.

When Section Moves Updates the intersection line when you move or resize the Section shape.

When Section Selected Updates the intersection line when you select the section shape, but not while you move it. Click the Update Section button to update the intersection.

Manually Updates the intersection line only when you click the Update Section button.

Update Section Updates the intersection to match the current placement of the Section object when using When Section Selected or Manually option.

NOTE When using When Section Selected or Manually, you can offset the generated cross section from the position of the intersected geometry. As you move the section object, the yellow cross-section lines move with it, leaving the geometry behind. When you click Create Shape, the new shape is generated at the displayed cross-section lines in the offset position.

Section Extents group

Choose one of these options to specify the extents of the cross-section generated by the section object.

Infinite The section plane is infinite in all directions, resulting in a cross section at any mesh geometry in its plane.

Section Boundary The cross-section is generated only in objects that are within or touched by the boundary of the section shape.

Off No cross section is displayed or generated. The Create Shape button is disabled.

Color swatch Click this to set the display color of the intersection.

Section Size rollout

Provides spinners that let you adjust the length and width of the displayed section rectangle.

Length/Width Adjust the length and width of the displayed section rectangle.

NOTE If you convert the section grid to an editable spline, it's converted to a shape based on the current cross section.

Extended Splines

WRectangle Spline

Create panel > Shapes > Extended Splines > Object Type rollout > WRectangle

Create menu > Shapes > WRectangle

Use WRectangle to create closed shapes from two concentric rectangles. Each rectangle is made of four vertices. The WRectangle is similar to the Donut tool except it uses rectangles instead of circles.

WRectangle stands for “walled rectangle”.

Example of WRectangle

Procedures

To create a wrectangle:

1  Go to the Create panel and choose Shapes.

2 Open the Shapes List and choose Extended Splines.

3 Click WRectangle.

4 Drag and release the mouse button to define the outer rectangle.

5 Move the mouse and then click to define the inner rectangle.

Interface

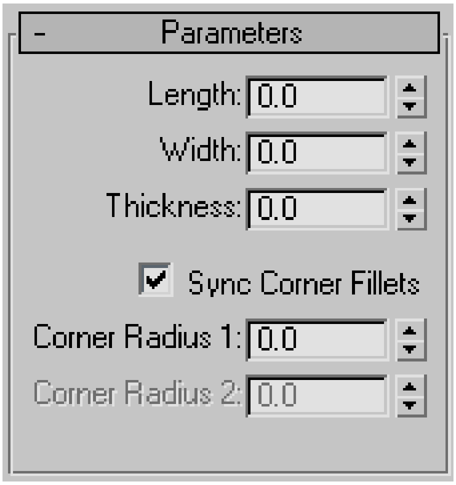
Rendering and Interpolation rollouts

All spline-based shapes share these parameters. For explanations, see Splines and Extended Splines on page 611.

Creation Method rollout

The WRectangle shape uses the standard creation methods of Center or Edge. Most spline-based shapes share the same Creation Method parameters. For explanations, see Splines and Extended Splines on page 611.

Parameters rollout



Length Controls the height of the wrectangle section.

Width Controls the width of the wrectangle section.

Thickness Controls the thickness of the walls of the wrectangle.

Sync Corner Fillets When turned on, Corner Radius 1 controls the radius of both the interior and exterior corners of the wrectangle. It also maintains the thickness of the section. Default=on.

Corner Radius 1 Controls the radius of all four interior and exterior corners of the section.

If Sync Corner Fillets is turned off, Corner Radius 1 controls the radius of the four exterior corners of the wrectangle.

Corner Radius 2 Controls the radius of the four interior corners of the wrectangle.

Corner Radius 2 is only available when Sync Corner Fillets is turned off.

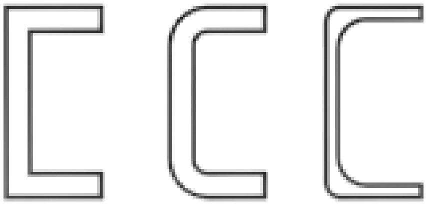
NOTE Take care when adjusting these settings. There are no constraining relationships between them. Therefore, it's possible to set an inside radius (Corner Radius 2) that is greater than the length and width of the sides.

Channel Spline

Create panel > Shapes > Extended Splines > Object Type rollout > Channel

Create menu > Shapes > Channel

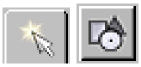
Use Channel to create a closed "C" shaped spline. You have the option to specify the interior and exterior corners between the vertical web and horizontal legs of the section.



Example of Channel

Procedures

To create a channel:

1.  Go to the Create panel and choose Shapes.
2. Open the Shapes List and select Extended Splines.
3. Click Channel.
4. Drag and release the mouse button to define the outer perimeter of the channel.
5. Move the mouse and then click to define the thickness of the walls of the channel.

Interface

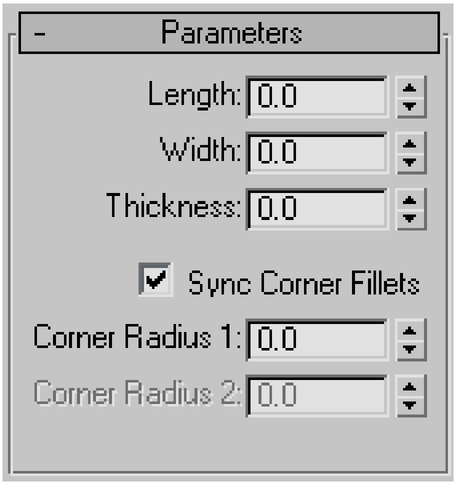
Rendering and Interpolation rollouts

All spline-based shapes share these parameters. For explanations, see Splines and Extended Splines on page 611.

Creation Method rollout

The Channel shape uses the standard creation methods of Center or Edge. Most spline-based shapes share the same Creation Method parameters. For explanations, see Splines and Extended Splines on page 611.

Parameters rollout



Length Controls the height of the vertical web of the channel.

Width Controls the width of the top and bottom horizontal legs of the channel.

Thickness Controls the thickness of both legs of the angle.

Sync Corner Fillets When turned on, Corner Radius 1 controls the radius of both the interior and exterior corners between the vertical web and horizontal legs. It also maintains the thickness of the channel. Default=on.

Corner Radius 1 Controls the exterior radius between the vertical web and horizontal legs of the channel.

Corner Radius 2 Controls the interior radius between the vertical web and horizontal legs of the channel.

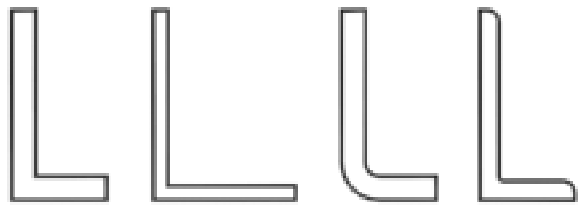
NOTE Take care when adjusting these settings. There are no constraining relationships between them. Therefore, it's possible to set an inside radius (Corner Radius 2) that is greater than the length of the web or width of the legs.

Angle Spline

Create panel > Shapes > Extended Splines > Object Type rollout > Angle

Create menu > Shapes > Angle

Use Angle to create a closed “L” shaped spline. You have the option to specify the radii of the corners between the vertical and horizontal legs of the section.



Example of Angle

Procedures

To create an Angle spline:

1  Go to the Create panel and choose Shapes.

2 Open the Shapes List and select Extended Splines.

3 Click Angle.

4 Drag and release the mouse button to define the initial size of the angle.

5 Move the mouse and then click to define the thickness of the walls of the angle.

Interface

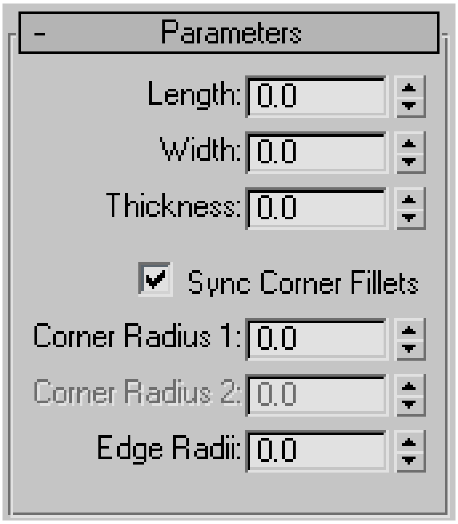
Rendering and Interpolation rollouts

All spline-based shapes share these parameters. For explanations, see Splines and Extended Splines on page 611.

Creation Method rollout

The Angle shape uses the standard creation methods of Center or Edge. Most spline-based shapes share the same Creation Method parameters. For explanations, see Splines and Extended Splines on page 611.

Parameters rollout



Length Controls the height of the vertical leg of the angle.

Width Controls the width of the horizontal leg of the angle.

Thickness Controls the thickness of both legs of the angle.

Sync Corner Fillets When turned on, Corner Radius 1 controls the radius of both the interior and exterior corners between the vertical and horizontal legs. It also maintains the thickness of the section. Default=on.

Corner Radius 1 Controls the exterior radius between the vertical and horizontal legs of the angle.

Corner Radius 2 Controls the interior radius between the vertical and horizontal legs of the angle.

Edge Radii Controls the interior radius at the outermost edges of the vertical and horizontal legs.

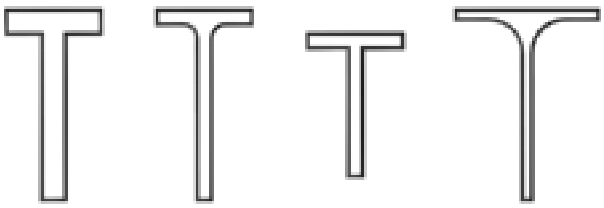
NOTE Take care when adjusting these settings. There are no constraining relationships between them. Therefore, it's possible to set an inside radius (Corner Radius 2) that is greater than the length or width of the legs of the angle.

Tee Spline

Create panel > Shapes > Extended Splines > Object Type rollout > Tee

Create menu > Shapes > Tee

Use Tee to create a closed T-shaped spline. You can specify the radius of the two interior corners between the vertical web and horizontal flange of the section.



Example of Tee

Procedures

To create a Tee spline:

1.  Go to the Create panel and choose Shapes.
2. Open the Shapes List and select Extended Splines.
3. Click Tee.
4. Drag and release the mouse button to define the initial size of the tee.
5. Move the mouse and then click to define the thickness of the walls of the tee.

Interface

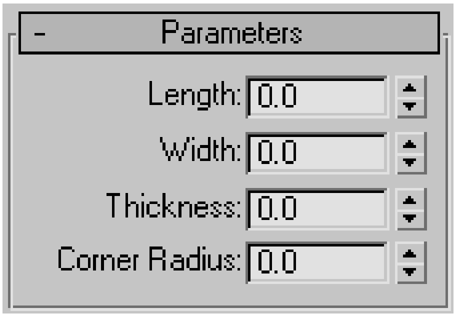
Rendering and Interpolation rollouts

All spline-based shapes share these parameters. For explanations, see Splines and Extended Splines on page 611.

Creation Method rollout

The Tee shape uses the standard creation methods of Center or Edge. Most spline-based shapes share the same Creation Method parameters. For explanations, see Splines and Extended Splines on page 611.

Parameters rollout



Length Controls the height of the vertical web of the tee.

Width Controls the width of the flange crossing the tee.

Thickness Controls the thickness of the web and flange.

Corner Radius Controls the radius of the two interior corners between the vertical web and horizontal flange of the section.

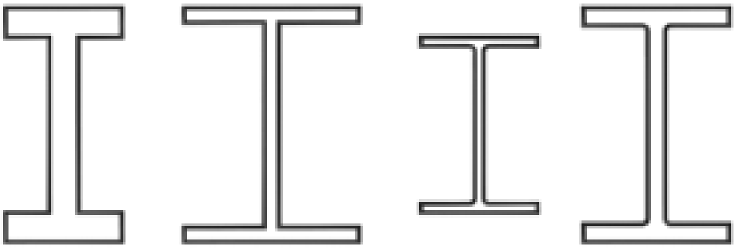
NOTE Take care when adjusting these settings. There are no constraining relationships between them. Therefore, it's possible to set a radius (Corner Radius) that is greater than the length of the web or width of the flange.

Wide Flange Spline

Create panel > Shapes > Extended Splines > Object Type rollout > Wide Flange

Create menu > Shapes > Wide Flange

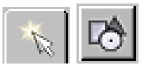
Use Wide Flange to create a closed spline shaped like a capital letter I. You can specify the interior corners between the vertical web and horizontal flanges of the section.



Example of Wide Flange

Procedures

To create a Wide Flange spline:

1.  Go to the Create panel and choose Shapes.
2. Open the Shapes List and select Extended Splines.
3. Click Wide Flange.
4. Drag and release the mouse button to define the initial size of the wide flange.
5. Move the mouse and then click to define the thickness of the walls of the wide flange.

Interface

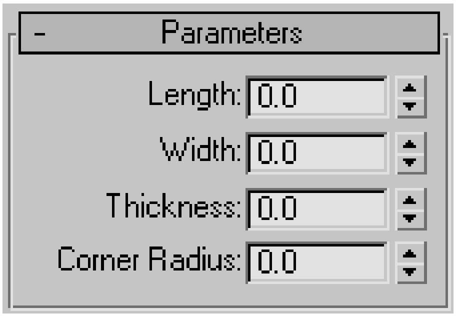
Rendering and Interpolation rollouts

All spline-based shapes share these parameters. For explanations, see Splines and Extended Splines on page 611.

Creation Method rollout

The Wide Flange shape uses the standard creation methods of Center or Edge. Most spline-based shapes share the same Creation Method parameters. For explanations, see Splines and Extended Splines on page 611.

Parameters rollout



Length Controls the height of the vertical web of the wide flange.

Width Controls the width of the horizontal flanges crossing the wide flange.

Thickness Controls the thickness of the web and flanges.

Corner Radius Controls the radius of the four interior corners between the vertical web and horizontal flanges.

NOTE Take care when adjusting these settings. There are no constraining relationships between them. Therefore, it's possible to set a radius (Corner Radius) that is greater than the length of the web or width of the flanges.

Editable Spline

Create or select a spline > Modify panel > Right-click spline entry in the stack display > Convert To: Editable Spline

Create a line > Modify panel

Create or select a spline > Right-click the spline > Transform (lower-right) quadrant of the quad menu > Convert To: > Convert to Editable Spline

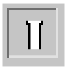
Editable Spline provides controls for manipulating an object as a spline object and at three sub-object levels: vertex, segment, and spline.

The functions in Editable Spline are the same as those in the Edit Spline modifier on page 1447. The exception is that when you convert an existing spline shape to an editable spline, the creation parameters are no longer accessible or animatable. However, the spline's interpolation settings (step settings) remain available in the editable spline.

When a spline-editing operation (typically, moving a segment or vertex) causes end vertices to overlap, you can use the Weld on page 681command to weld the overlapping vertices together or the Fuse on page 682 command if you want the two overlapping vertices to occupy the same point in space but remain separate vertices.

NOTE Welding coincident vertices is controlled by the End Point Auto-Welding feature on page ?.

Show End Result

 If you have several modifiers higher in the modifier stack, and want to see the results of edits in an Edit Spline modifier or Editable Spline object, then turn on Show End Result on the Modify panel. As you edit the spline network, you'll be able to see the result of modifiers above the Editable Spline object. This is useful for Surface Tools work where you add a Surface modifier above an Editable Spline object in the modifier stack.

See also:

- Edit Modifiers and Editable Objects on page 1097
- Modifying at the Sub-Object Level on page 1098
- Modifier Stack Controls on page 7635

Procedures

To produce an editable spline object, first select the shape, and then do one of the following:

1. Right-click the shape entry in the stack display and choose Convert To: Editable Spline.
2. In a viewport, right-click the object and choose Convert To: > Convert to Editable Spline from the Transform (lower-right) quadrant of the quad menu.
3. Create a shape with two or more splines by first turning off Start New Shape (on the Create panel). Any shape made up of two or more splines is automatically an editable spline.

4 Apply an Edit Spline modifier to a shape, and then collapse the stack. If you use the Collapse utility on page 2016 to collapse the stack, be sure to choose Output Type > Modifier Stack Result.

5 Import a *.shp* file.

6 Merge a shape from a 3ds Max file.

To select shape sub-objects:

1 Expand the object's hierarchy in the stack display and choose a sub-object level, or click one of the sub-object buttons at the top of the Selection rollout.

   You can also right-click the object in the viewports and choose a sub-object level from the quad menu: Tools 1 (upper-left) quadrant > Sub-objects > Choose the sub-object level.

2 Click a selection or transform tool, and then select sub-objects using standard click or region-selection techniques.

   Because sub-object selections can be complex, you might consider using one of the following techniques to prevent clearing the sub-object selection by accident:

   - Use Lock Selection on page 7539.
   - Name the sub-object selection (see Named Selection Sets List on page 206).

To clone sub-object selections:

- Hold down the Shift key while transforming the sub-objects.
  You can clone segment and spline sub-objects, but not vertices.

To draw a spline cage:

1 Select a segment sub-object on a spline.

2 On the Geometry rollout in the Connect Copy group, turn on Connect.

3 Hold down the Shift key and transform the selected segment. You can move, rotate or scale using the transform gizmo to control the direction.

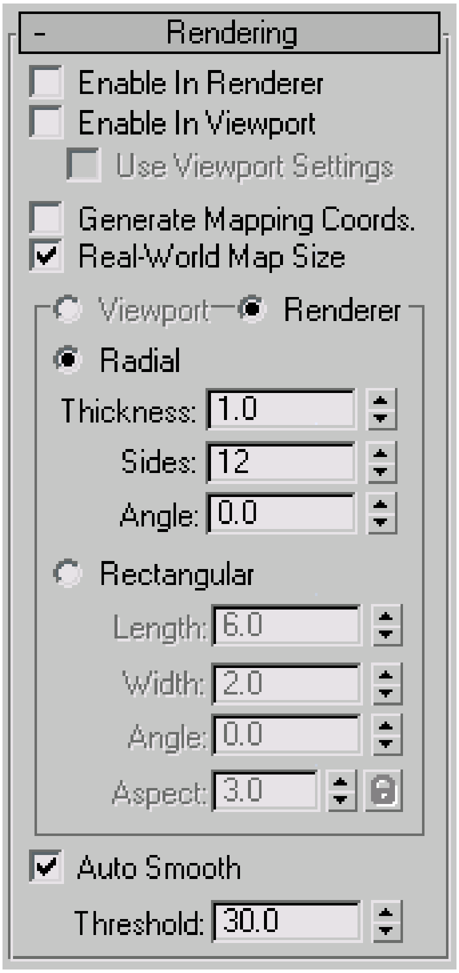
   Notice that with Connect Copy on, new splines are drawn between the locations of the segment and its clone.

TIP Use Area Selection or Fuse before selecting and moving these vertices. They will not move together as they do with the Cross-Section modifier. Or use Fuse to keep the vertices together.

Interface

The following controls are available at the object (top) level and at all sub-object levels.

Rendering and Interpolation rollouts



These creation parameters appear in these rollouts for editable splines. For splines to which the Edit Spline modifier has been applied, creation parameters are available by selecting the object type entry (for example, Circle or NGon) at the bottom of the modifier stack on page 7635.

Rendering rollout

Controls here let you turn on and off the renderability of the shape, specify its thickness in the rendered scene, and apply mapping coordinates. The spline mesh can be viewed in the viewports. You can animate the render parameters, such as the number of sides. Viewport settings cannot be animated.

You can also convert the displayed mesh into a mesh object by applying an Edit Mesh modifier or converting to an Editable Mesh. The system will use the Viewport settings for this mesh conversion if Use Viewport Settings is turned on; otherwise it will use the Renderer settings. This gives maximum flexibility, and will always give the conversion of the mesh displayed in the viewports.

Enable In Renderer When on, the shape is rendered as a 3D mesh using the Radial or Rectangular parameters set for Renderer. In previous versions of the program, the Renderable switch performed the same operation.

Enable In Viewport When on, the shape is displayed in the viewport as a 3D mesh using the Radial or Rectangular parameters set for Renderer. In previous versions of the program, the Display Render Mesh performed the same operation.

Use Viewport settings Lets you set different rendering parameters, and displays the mesh generated by the Viewport settings. Available only when Enable in Viewport is turned on.

Generate Mapping Coords Turn this on to apply mapping coordinates. Default=off.

The U coordinate wraps once around the thickness of the spline; the V coordinate is mapped once along the length of the spline. Tiling is achieved using the Tiling parameters in the material itself.

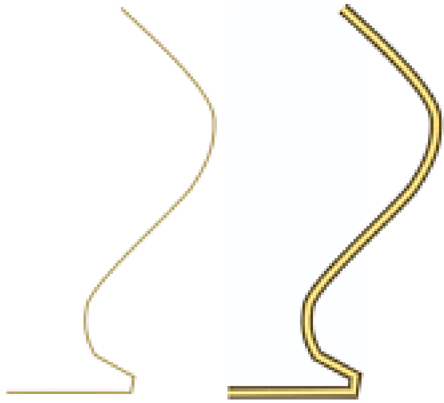
Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Viewport Turn this on to specify Radial or Rectangular parameters for the shape as it will display in the viewport when Enable in Viewport is turned on.

Renderer Turn this on to specify Radial or Rectangular parameters for the shape as it will display when rendered or viewed in the viewport when Enable in Viewport is turned on.

Radial Displays the 3D mesh as a cylindrical object.

- **Thickness** Specifies the diameter of the viewport or rendered spline mesh. Default=1.0. Range=0.0 to 100,000,000.0.



Splines rendered at thickness of 1.0 and 5.0, respectively

- **Sides** Sets the number of sides (or facets) for the spline mesh n the viewport or renderer. For example, a value of 4 results in a square cross section.
- **Angle** Adjusts the rotational position of the cross-section in the viewport or renderer. For example, if the spline mesh has a square cross section you can use Angle to position a "flat" side down.

Rectangular Displays the spline's mesh shape as rectangular.

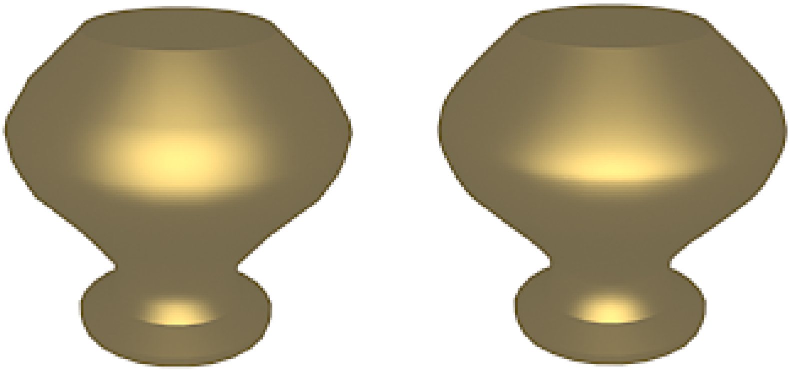
- **Aspect** Sets the aspect ratio for rectangular cross-sections. The Lock check box lets you lock the aspect ratio. When Lock is turned on, Width is locked to Depth that results in a constant ratio of Width to Depth.
- **Length** Specifies the size of the cross–section along the local *Y* axis.
- **Width** Specifies the size of the cross–section along the local *X* axis.
- **Angle** Adjusts the rotational position of the cross-section in the viewport or renderer. For example, if you have a square cross-section you can use Angle to position a "flat" side down.
- **Aspect** The ratio of length to width. This control is linked to the Length setting; when Aspect is unlocked, changing Length changes Aspect and vice-versa. When Aspect is locked, the control is unavailable, and changing Length or Width automatically changes the Width or Length (respectively) to maintain the current aspect ratio.

Auto Smooth When on, the spline is auto-smoothed using the threshold specified by the Threshold setting. Auto Smooth sets the smoothing based on the angle between spline segments. Any two adjacent segments are put in the same smoothing group if the angle between them is less than the threshold angle.

Threshold Specifies the threshold angle in degrees. Any two adjacent spline segments are put in the same smoothing group if the angle between them is less than the threshold angle.

Interpolation rollout

The Interpolation controls set how the program generates a spline. All spline curves are divided into small straight lines that approximate the true curve. The number of divisions between each vertex on the spline is called *steps*. The more steps used, the smoother the curve appears.



Splines used in above lathed objects contained two steps (left) and 20 steps (right)

Steps Use the Steps field to set the number of divisions, or steps, the program uses between each vertex. Splines with tight curves require many steps to look smooth while gentle curves require fewer steps. Range=0 to 100.

Spline steps can be either adaptive or manually specified. The method used is set by the state of the Adaptive check box. The main use for manual interpolation is to create splines for morphing or other operations where you must have exact control over the number of vertices created.

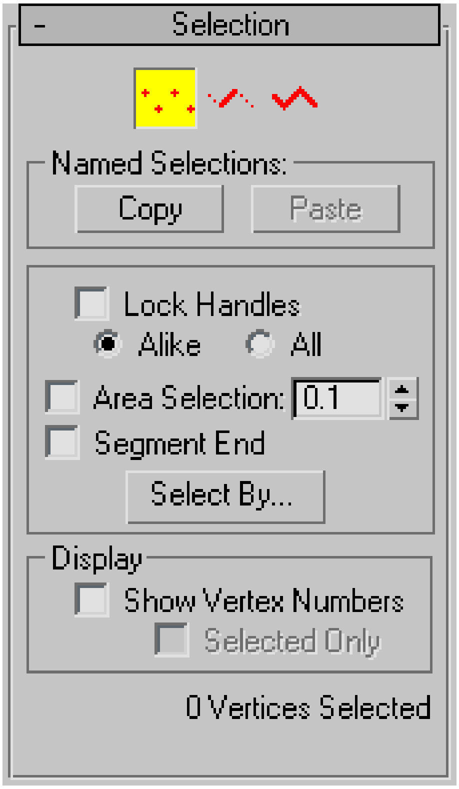
Optimize When on, removes unneeded steps from straight segments in the spline. Default=on.

NOTE Optimize is not available when Adaptive is on.

Optimize was used to create spline in this lathed object.

Adaptive When on, automatically sets the number of steps for each spline to produce a smooth curve. Straight segments always receive 0 steps. When off, enables manual interpolation control using Optimize and Steps. Default=off.

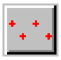
Selection rollout

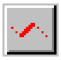


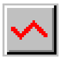
Provides controls for turning different sub-object modes on and off, working with named selections and handles, display settings, and information about selected entities.

When you first access the Modify panel with an editable spline selected, you're at the Object level, with access to several functions available as described in Editable Spline (Object) on page 671. You can toggle the sub-object modes and access relevant functions by clicking sub-object buttons at the top of the Selection rollout.

You can work with parts of shapes and splines using shape sub-object selection of the Editable Spline object. Clicking a button here is the same as selecting a sub-object type in the Modifier List. Click the button again to turn it off and return to object selection level.

 **Vertices** Define points and curve tangents.

 **Segments** Connect vertices.

 **Splines** Are a combination of one or more connected segments.

Named Selections group

Copy Places a named selection into the copy buffer.

Paste Pastes a named selection from the copy buffer.

Lock Handles Normally you can transform the tangent handles of only one vertex at a time, even when multiple vertices are selected. Use the Lock Handles controls to transform multiple Bezier and Bezier Corner handles simultaneously.

Alike As you drag the handle of an incoming vector, all incoming vectors of the selected vertices move simultaneously. Likewise, moving the outgoing tangent handle on one vertex moves the outgoing tangent handle for all selected vertices.

All Any handle you move affects all handles in the selection, regardless of whether they're broken. This option is also useful when working with a single Bezier Corner vertex when you want to move both handles.

Shift+click a handle to "break" the tangent and move each handle independently. The Alike option must be chosen to break the tangent.

Area Selection Lets you select automatically all vertices within a specific radius of the vertex you click. At the Vertex sub-object level, turn on Area Selection, and then set the radius with the spinner to the right of the Area Selection check box. This is useful when moving vertices that have been created using Connect Copy or Cross Section button.

Segment End Select a vertex by clicking a segment. In Vertex sub-object, turn on and select a segment close to the vertex that you want selected. Use this when there are a number of coincident vertices and you want to select a vertex on a specific segment. The cursor changes to a cross when it is over a segment. By holding down the Ctrl key you can add to the selection.

Select By Selects vertices on the selected spline or segment. First select a spline or segment in sub-object spline or segment, then turn on vertex sub-object and click Select By and choose Spline or Segment. All the vertices on the selected spline or segment are selected. You can then edit the vertices.

Display group

Show Vertex Numbers When on, the program displays vertex numbers next to the selected spline's vertices at any sub-object level.

Selected Only When on, the vertex number or numbers appear only next to selected vertices.

Soft Selection

For information on the Soft Selection rollout settings, see Soft Selection Rollout on page 2008.

Selection Info

At the bottom of the Selection rollout is a text display giving information about the current selection. If 0 or more than one sub-object is selected, the text gives the number selected.

At the Vertex and Segment sub-object levels, if one sub-object is selected, the text gives the identification numbers of the current spline (with respect to the current object) and of the current selected sub-object. Each spline object contains a spline number 1; if it contains more than one spline, the subsequent splines are numbered consecutively higher.

When a single spline is selected at the Spline sub-object level, the first line displays the identification number of the selected spline and whether it's open or closed, and the second line displays the number of vertices it contains. When more than one spline is selected, the number of splines selected is displayed on the first line, and the total number of vertices they contain is displayed on the second line.

Geometry rollout

The Geometry rollout provides functions for editing a spline object and sub-objects. The functions available at the spline object level (when no sub-object level is active; see Editable Spline (Object) on page 671) are also available at all sub-object levels, and work exactly the same at each level. Other functions are also available, depending on which sub-object level is active. Those that apply to other sub-object levels are unavailable.

For specific information, see these topics:

Editable Spline (Object)

Select an editable spline > Modify panel > Editable spline (not a sub-object level) selected in the modifier stack

Select an editable spline > Right-click the spline > Tools 1 (upper-left) quadrant of the quad menu > Sub-objects > Top-level

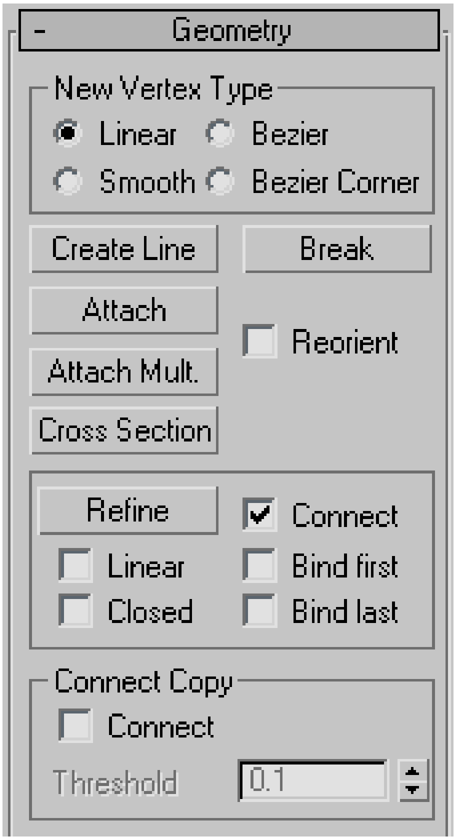
The functions available at the editable spline object level (that is, when no sub-object level is active) are also available at all sub-object levels, and work exactly the same at each level.

Interface

Rendering, Interpolation, and Selection rollouts

See the Editable Spline topic for information on the Rendering and Interpolation rollouts on page 663, and Selection rollout on page 668 settings.

Geometry rollout



New Vertex Type group The radio buttons in this group let you determine the tangency of the new vertices created when you Shift+Clone segments or splines. If you later use Connect Copy, vertices on the splines that connect the original segment or spline to the new one will have the type specified in this group.

This setting has no effect on the tangency of vertices created using tools such as the Create Line button, Refine, and so on.

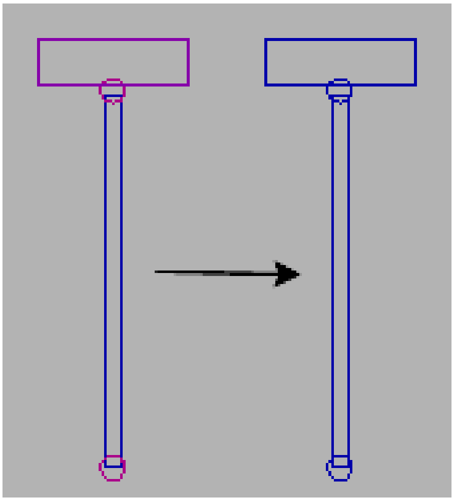
- **Linear** New vertices will have linear tangency.
- **Smooth** New vertices will have smooth tangency.
  When this option is chosen, new vertices that overlap are automatically welded.

- **Bezier** New vertices will have bezier tangency.
- **Bezier Corner** New vertices will have bezier corner tangency.

Create Line Adds more splines to the selected spline. These lines are separate spline sub-objects; create them in the same way as the line spline on page 620. To exit line creation, right-click or click to turn off Create Line.

Break Splits a spline at the selected vertex or vertices. Select one or more vertices and then click Break to create the split. There are now two superimposed non-connected vertices for every previous one, allowing the once-joined segment ends to be moved away from each other.

Attach Lets you attach another spline in the scene to the selected spline. Click the object you want to attach to the currently selected spline object. The object you're attaching to must also be a spline.



Unattached splines (left) and attached splines (right)

When you attach an object, the materials of the two objects are combined in the following way:

- If the object being attached does not have a material assigned, it inherits the material of the object it is being attached to.
- Likewise if the object you're attaching to doesn't have a material, it inherits the material of the object being attached.

- If both objects have materials, the resulting new material is a multi/sub-object material on page 5720 that encompasses the input materials. A dialog appears offering three methods of combining the objects' materials and material IDs. For more information, see Attach Options Dialog on page 2116.
  Attached shapes lose their identity as individual shapes, with the following results:

- The attached shape loses all access to its creation parameters. For example, once you attach a circle to a square you cannot go back and change the radius parameter of the circle.

- The modifier stack of the attached shape is collapsed.
  Any edits, modifiers, and animation applied to the attached shape are frozen at the current frame.

Reorient When on, rotates the attached spline so that its creation local coordinate system is aligned with the creation local coordinate system of the selected spline.

Attach Mult. Click this button to display the Attach Multiple dialog, which contains a list of all other shapes in the scene. Select the shapes you want to attach to the current editable spline, then click OK.

Cross Section Creates a spline cage out of cross-sectional shapes. Click Cross Section, select one shape then a second shape, splines are created joining the first shape with the second. Continue clicking shapes to add them to the cage. This functionality is similar to the Cross Section modifier, but here you can determine the order of the cross sections. Spline cage tangency can be defined by choosing Linear, Bezier, Bezier Corner or Smooth in New Vertex Type group.

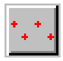
End Point Auto-Welding group

- **Automatic Welding** When Automatic Welding is turned on, an end point vertex that is placed or moved within the threshold distance of another end point of the same spline is automatically welded. This feature is available at the object and all sub-object levels.

- **Threshold** A proximity setting that controls how close vertices can be to one another before they are automatically welded. Default=6.0.

Insert Inserts one or more vertices, creating additional segments. Click anywhere in a segment to insert a vertex and attach the vertex to the mouse. Optionally move the mouse and then click to place the new vertex. Continue moving the mouse and clicking to add vertices. A single click inserts a corner vertex, while a drag creates a Bezier (smooth) vertex.

Right-click to complete the operation and release the mouse. At this point, you're still in Insert mode, and can begin inserting vertices in a different segment. Otherwise, right-click again or click Insert to exit Insert mode.

Editable Spline (Vertex) 

Select an editable spline > Modify panel > Expand the editable spline in the stack display > Vertex sub-object level

Select an editable spline > Modify panel > Selection rollout > Vertex button

Select an editable spline > Right-click the spline > Tools 1 (upper-left) quadrant of the quad menu > Sub-objects > Vertex

While at the Editable Spline (Vertex) level, you can select single and multiple vertices and move them using standard methods. If the vertex is of the Bezier or Bezier Corner type, you can also move and rotate handles, thus affecting the shapes of any segments joined at the vertex. You can copy and paste the handles between vertices using tangent copy/paste. You can reset them or switch between types using the quad menu. The tangent types are always available on the quad menu when a vertex is selected; your cursor doesn't have to be directly over them in the viewport.

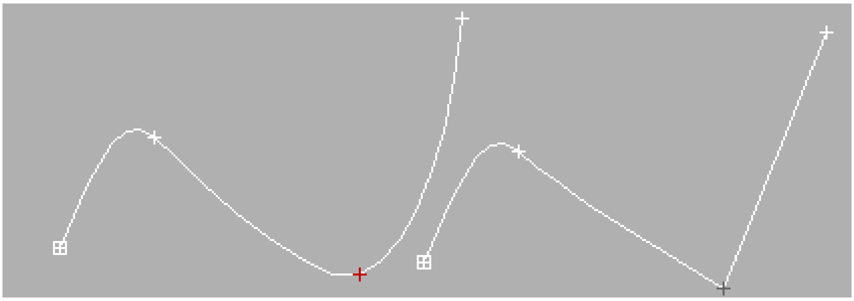
Procedures

To set a vertex type:

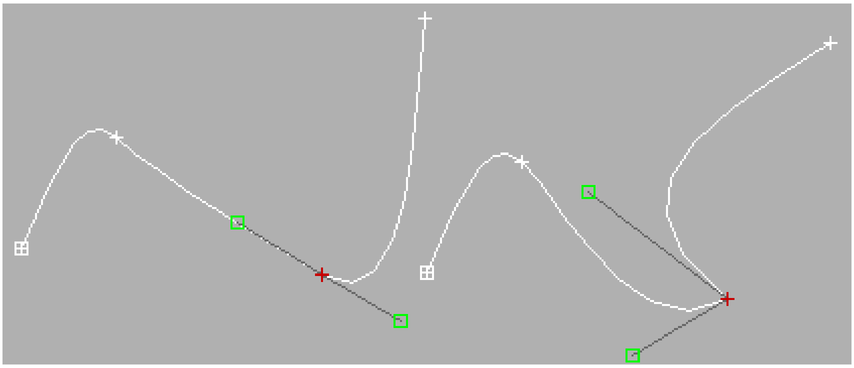
1 Right-click any vertex in a selection.

2 Choose a type from the shortcut menu. Each vertex in a shape can be one of four types:

- **Smooth:** Nonadjustable vertices that create smooth continuous curves. The curvature at a smooth vertex is determined by the spacing of adjacent vertices.
- **Corner:** Nonadjustable vertices that create sharp corners.
- **Bezier:** Adjustable vertex with locked continuous tangent handles that create a smooth curve. The curvature at the vertex is set by the direction and magnitude of the tangent handles.
- **Bezier Corner:** Adjustable vertex with discontinuous tangent handles that create a sharp corner. The curvature of the segment as it leaves the corner is set by the direction and magnitude of the tangent handles.

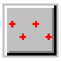


Smooth vertex (left) and Corner vertex (right)



Bezier vertex (left) and Bezier Corner vertex (right)

To copy and paste vertex tangent handles:

1  Turn on Vertex Selection, then Select the vertex you want to copy from.

2 On the Geometry rollout scroll down to the Tangent group and click Copy.

3 Move your cursor over the vertices in the viewport. The cursor changes to a copy cursor. Click the handle you wish to copy.

4 On the Geometry rollout scroll down to the Tangent group and click Paste.

5 Move your cursor over the vertices in the viewport. The cursor changes to a paste cursor. Click the handle you wish to paste to.

   The vertex tangency changes in the viewport.

To reset vertex handle tangency:

It is easy to make the handles very small and coincident with the vertex, which makes them hard to select and edit. Reset the vertex handle tangency to redraw your handles

1 Select the vertex that is problematic.

2 Right-click and choose Reset Tangents.

   Any vertex handle editing you have done is discarded and the handles are reset.

Interface

Soft Selection rollout

For information on the Soft Selection rollout settings, see Soft Selection Rollout on page 2008.

Geometry rollout

New Vertex Type group

The radio buttons in this group let you determine the tangency of the new vertices created when you Shift+Clone segments or splines. If you later use Connect Copy, vertices on the splines that connect the original segment or spline to the new one will have the type specified in this group.

This setting has no effect on the tangency of vertices created using tools such as the Create Line button, Refine, and so on.

- **Linear** New vertices will have linear tangency.
- **Smooth** New vertices will have smooth tangency.
  When this option is chosen, new vertices that overlap are automatically welded.
- **Bezier** New vertices will have bezier tangency.
- **Bezier Corner** New vertices will have bezier corner tangency.

Create Line Adds more splines to the selected object. These lines are separate spline sub-objects; create them in the same way as the line spline on page 620. To exit line creation, right-click or click to turn off Create Line.

Break Splits a spline at the selected vertex or vertices. Select one or more vertices and then click Break to create the split. There are now two superimposed non-connected vertices for every previous one, allowing the once-joined segment ends to be moved away from each other.

Attach Attaches another spline in the scene to the selected spline. Click the object you want to attach to the currently selected spline object. The object you're attaching must also be a spline.

For further details, see Attach on page 673.

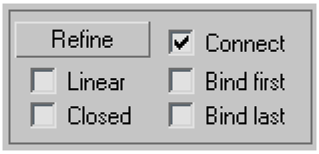
Attach Mult. Click this button to display the Attach Multiple dialog, which contains a list of all other shapes in the scene. Select the shapes you want to attach to the current editable spline, then click OK.

- **Reorient** When on, reorients attached splines so that each spline's creation local coordinate system is aligned with the creation local coordinate system of the selected spline.

Cross Section Creates a spline cage out of cross-sectional shapes. Click Cross Section, select one shape then a second shape, splines are created joining the first shape with the second. Continue clicking shapes to add them to the cage. This functionality is similar to the Cross Section modifier, but here you can determine the order of the cross sections. Spline cage tangency can be defined by choosing Linear, Bezier, Bezier Corner or Smooth in New Vertex Type group.

TIP When you edit the spline cage, use Area Selection before selecting your vertices. This will keep their positions together as you transform them.

Refine group



The Refine group includes a number of functions useful for building spline networks for use with the Surface modifier on page 1766.

Refine Lets you add vertices without altering the curvature values of the spline. Click Refine, and then select any number of spline segments to add a vertex each time you click (the mouse cursor changes to a "connect" symbol when over an eligible segment). To finish adding vertices, click Refine again, or right-click in the viewport.

You can also click existing vertices during a refine operation, in which case 3ds Max displays a dialog asking if you want to **Refine** or **Connect Only** to the vertex. If you choose Connect Only, 3ds Max will not create a vertex: it simply connects to the existing vertex.

The Refine operation creates a different type of vertex depending on the types of vertices on the endpoints of the segment being refined.

- If the bordering vertices are both Smooth types, the Refine operation creates a Smooth type vertex.
- If the bordering vertices are both Corner types, the Refine operation creates a Corner type vertex.
- If either of the bordering vertices is a Corner or Bezier Corner, the Refine operation creates a Bezier Corner type.
- Otherwise, the operation creates a Bezier type vertex.

Connect When on, creates a new spline sub-object by connecting the new vertices. When you finish adding vertices with Refine, Connect makes a separate copy of each new vertex and then connects all of the copies with a new spline.

NOTE For Connect to work, you must turn it on before you click Refine.

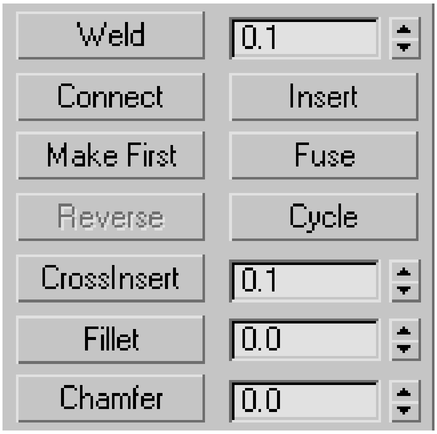
After turning on Connect and before beginning the refinement process, turn on any combination of these options:

- **Linear** When on, makes all segments in the new spline straight lines by using Corner vertices. When Linear is off, the vertices used to create the new spline are of the Smooth type.
- **Bind First** Causes the first vertex created in a refinement operation to be bound to the center of the selected segment. See Bound Vertex on page 7931.
- **Closed** When on, connects the first and last vertices in the new spline to create a closed spline. When Closed is off, Connect always creates an open spline.
- **Bind Last** Causes the last vertex created in a refinement operation to be bound to the center of the selected segment. See Bound Vertex on page 7931.

End Point Auto-Welding group

Automatic Welding When Automatic Welding is turned on, an end point vertex that is placed or moved within the threshold distance of another end point of the same spline is automatically welded. This feature is available at the object and all sub-object levels.

Threshold The threshold distance spinner is a proximity setting that controls how close vertices can be to one another before they are automatically welded. Default=6.0.



Weld Converts two end vertices, or two adjacent vertices within the same spline, into a single vertex. Move either two end vertices or two adjacent vertices near each other, select both vertices, and then click Weld. If the vertices are within the unit distance set by the Weld Threshold spinner (to the right of the button), they're converted into a single vertex. You can weld a selection set of vertices, as long as each pair of vertices is within the threshold.

Connect Connects any two end vertices, resulting in a linear segment, regardless of the tangent values of the end vertices. Click the Connect button, point the mouse over an end vertex until the cursor changes to a cross, and then drag from one end vertex to another end vertex.

Insert Inserts one or more vertices, creating additional segments. Click anywhere in a segment to insert a vertex and attach the mouse to the spline. Then optionally move the mouse and click to place the new vertex. Continue moving the mouse and clicking to add vertices. A single click inserts a corner vertex, while a drag creates a Bezier (smooth) vertex.

Right-click to complete the operation and release the mouse. At this point, you're still in Insert mode, and can begin inserting vertices in a different segment. Otherwise, right-click again or click Insert to exit Insert mode.

Make First Specifies which vertex in the selected shape is the first vertex. The first vertex of a spline is indicated as a vertex with a small box around it. Select one vertex on each spline within the currently edited shape that you want to change and click the Make First button.

On open splines, the first vertex must be the endpoint that is not already the first vertex. On closed splines, it can be any point that isn't already the first vertex. Click the Make First button, and the first vertices will be set.

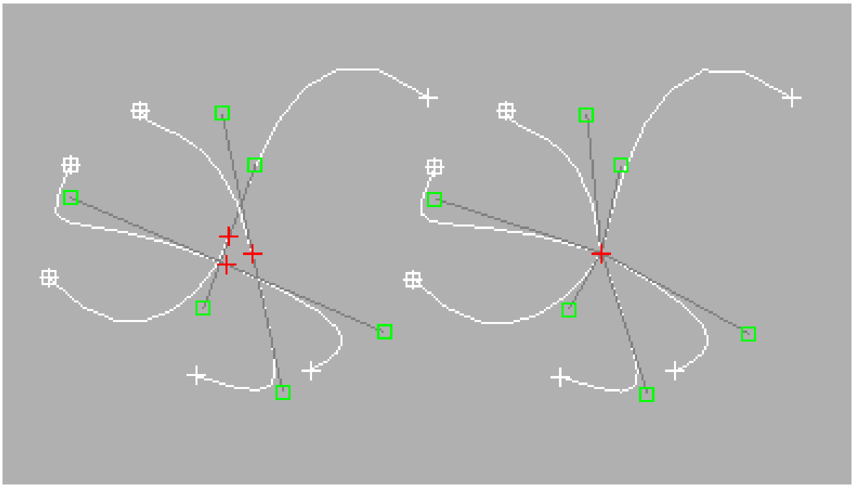
The first vertex on a spline has special significance. The following table defines how the first vertex is used.

Shape Use	First Vertex Meaning
Loft Path	Start of the path. Level 0.
Loft Shape	Initial skin alignment.
Path Constraint	Start of the motion path. 0% location on the path.
Trajectory	First position key.

Fuse Moves all selected vertices to their averaged center.

Fuse is useful for making vertices coincide when building a spline network for use with the Surface modifier on page 1766.

NOTE Fuse doesn't join the vertices; it simply moves them to the same location.



Three selected vertices (left); fused vertices (right)

Cycle Selects successive coincident vertices. Select one of two or more vertices that share the exact same location in 3D space, and then click Cycle repeatedly until the vertex you want is selected.

Cycle is useful for selecting a specific vertex from a group of coincident vertices at a spline intersection when building a spline network for use with the Surface modifier on page 1766.

TIP Watch the info display at the bottom of the Selection rollout to see which vertex is selected.

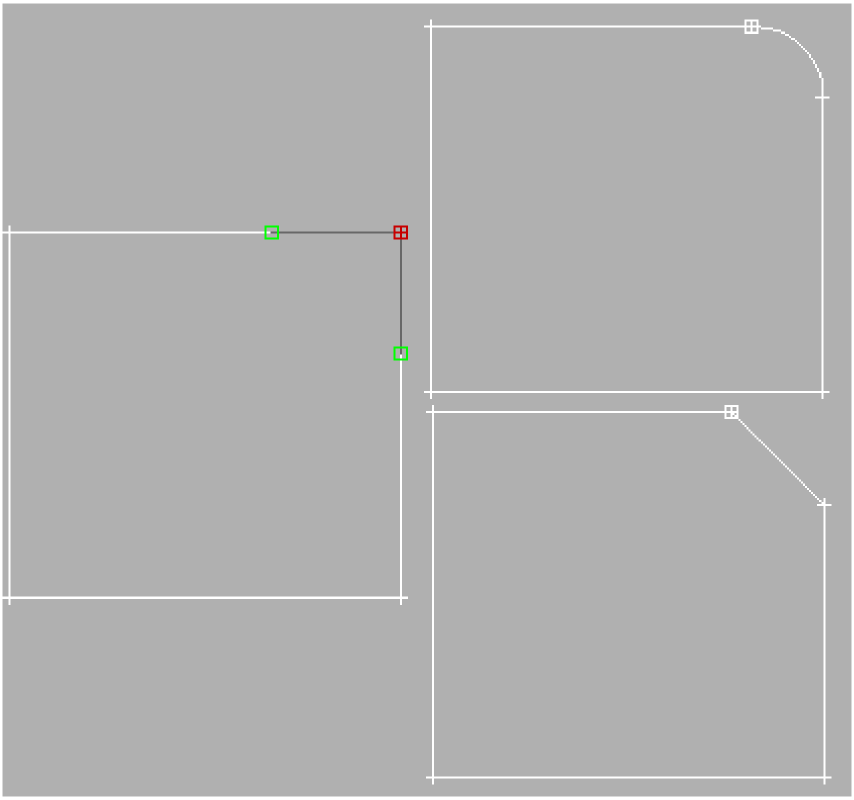
CrossInsert Adds vertices at the intersection of two splines belonging to the same spline object. Click CrossInsert, and then click the point of intersection between the two splines. If the distance between the splines is within the unit distance set by the CrossInsert Threshold spinner (to the right of the button), the vertices are added to both splines.

You can continue using CrossInsert by clicking different spline intersections. To finish, right-click in the active viewport or click the CrossInsert button again.

CrossInsert is useful for creating vertices at spline intersections when building a spline network for use with the Surface modifier on page 1766.

NOTE CrossInsert doesn't join the two splines, but simply adds vertices where they cross.

Fillet Lets you round corners where segments meet, adding new control vertices. You can apply this effect interactively (by dragging vertices) or numerically (using the Fillet spinner). Click the Fillet button, and then drag vertices in the active object. The Fillet spinner updates to indicate the fillet amount as you drag.



Original rectangle (left), after applying Fillet (top right), and after applying Chamfer (bottom right)

If you drag one or more selected vertices, all selected vertices are filleted identically. If you drag an unselected vertex, any selected vertices are first deselected.

You can continue using Fillet by dragging on different vertices. To finish, right-click in an active viewport or click the Fillet button again.

A fillet creates a new segment connecting new points on both segments leading to the original vertex. These new points are exactly <fillet amount> distance from the original vertex along both segments. New fillet segments are created with the material ID of one of the neighboring segments (picked at random).

For example, if you fillet one corner of a rectangle, the single corner vertex is replaced by two vertices moving along the two segments that lead to the corner, and a new rounded segment is created at the corner.

NOTE Unlike the Fillet/Chamfer modifier, you can apply the Fillet function to *any* type of vertex, not just Corner and Bezier Corner vertices. Similarly, adjoining segments need not be linear.

- **Fillet Amount** Adjust this spinner (to the right of the Fillet button) to apply a fillet effect to selected vertices.

Chamfer Lets you bevel shape corners using a chamfer function. You can apply this effect interactively (by dragging vertices) or numerically (using the Chamfer spinner). Click the Chamfer button, and then drag vertices in the active object. The Chamfer spinner updates to indicate the chamfer amount as you drag.

If you drag one or more selected vertices, all selected vertices are chamfered identically. If you drag an unselected vertex, any selected vertices are first deselected.

You can continue using Chamfer by dragging on different vertices. To finish, right-click in an active viewport or click the Chamfer button again.

A chamfer "chops off" the selected vertices, creating a new segment connecting new points on both segments leading to the original vertex. These new points are exactly <chamfer amount> distance from the original vertex along both segments. New chamfer segments are created with the material ID of one of the neighboring segments (picked at random).

For example, if you chamfer one corner of a rectangle, the single corner vertex is replaced by two vertices moving along the two segments that lead to the corner, and a new segment is created at the corner.

NOTE Unlike the Fillet/Chamfer modifier, you can apply the Chamfer function to *any* type of vertex, not just Corner and Bezier Corner vertices. Similarly, adjoining segments need not be linear.

- **Chamfer Amount** Adjust this spinner (to the right of the Chamfer button) to apply a chamfer effect to selected vertices.

Tangent group

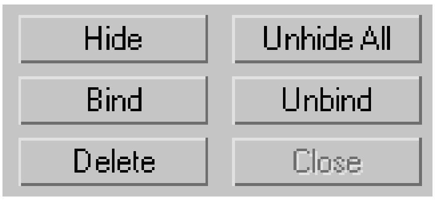
Tools in this group let you copy and paste vertex handles from one vertex to another.

Copy Turn this on, then choose a handle. This action copies the selected handle tangent into a buffer.

Paste Turn this on, then click a handle. This pastes the handle tangent onto the selected vertex.

Paste Length When this is on, the handle length is also copied. When this is off, only the handle angle is considered, the handle length is unchanged.

Hide and Bind group



Hide Hides selected vertices and any connected segments. Select one or more vertices, and then click Hide.

Unhide All Displays any hidden sub-objects.

Bind Lets you create bound vertices on page 7931. Click Bind, and then drag from any end vertex in the current selection to any segment in the current selection except the one connected to the vertex. Before dragging, when the cursor is over an eligible vertex, it changes to a + cursor. While dragging, a dashed line connects the vertex and the current mouse position, and when the mouse cursor is over an eligible segment, it changes to a "connect" symbol. When you release over an eligible segment, the vertex jumps to the center of the segment and is bound to it.

Bind is useful for connecting splines when building a spline network for use with the Surface modifier on page 1766.

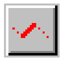
Unbind Lets you disconnect bound vertices on page 7931 from the segments to which they're attached. Select one or more bound vertices, and the click the Unbind button.

Delete Deletes the selected vertex or vertices, along with one attached segment per deleted vertex.

Display group

Show selected segs When on, any selected segments are highlighted in red at the Vertex sub-object level. When off (the default), selected segments are highlighted only at the Segment sub-object level.

This feature is useful for comparing complex curves against each other.

Editable Spline (Segment) 

Select an editable spline > Modify panel > Expand the editable spline in the stack display > Segment sub-object level

Select an editable spline > Modify panel > Selection rollout > Segment button

Select an editable spline > Right-click the spline > Tools 1 (upper-left) quadrant of the quad menu > Sub-objects > Segment

A segment is the portion of a spline curve between two of its vertices. While at the Editable Spline (Segment) level, you can select single and multiple segments and move, rotate, scale or clone them using standard methods.

Procedures

To change segment properties:

1. Select an editable spline segment, and then right-click.
2. On the Tools 1 (upper-left) quadrant of the quad menu, choose Line or Curve.

The effect of changing segment properties varies according to the type of vertices at the segment end.

- Corner vertices always result in line segments regardless of the segment property.
- Smooth vertices can support both line or curve segment properties.
- Bezier and Bezier Corner vertices apply their tangent handles only to curve segments. Tangent handles are ignored by line segments.
- A tangent handle associated with a line segment displays an X at the end of the handle. You can still transform the handle, but it has no effect until the segment is converted to a curve segment.

TIP If you have problems transforming the handles, display the axis constraints toolbar and change the transform axis there.

Interface

Rendering, Interpolation, and Selection rollouts

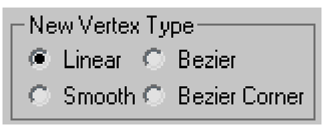
For information on the Rendering, Interpolation on page 663 and Selection rollout on page 668 settings, see Editable Spline on page 659.

Soft Selection rollout

For information on the Soft Selection rollout settings, see Soft Selection Rollout on page 2008.

Geometry rollout

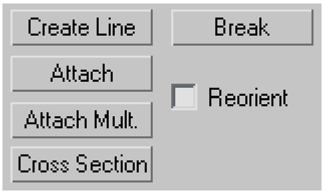
New Vertex Type group



The radio buttons in this group let you determine the tangency of the new vertices created when you Shift+Clone segments or splines. If you later use Connect Copy, vertices on the splines that connect the original segment or spline to the new one will have the type specified in this group.

This setting has no effect on the tangency of vertices created using tools such as the Create Line button, Refine, and so on.

- **Linear** New vertices will have linear tangency.
- **Smooth** New vertices will have smooth tangency.
  When this option is chosen, new vertices that overlap are automatically welded.
- **Bezier** New vertices will have bezier tangency.
- **Bezier Corner** New vertices will have bezier corner tangency.



Create Line Adds more splines to the selected spline. These lines are separate spline sub-objects; create them in the same way as the line spline on page 620. To exit line creation, right-click or click to turn off Create Line.

Break Lets you specify a break point at any segment in the shape (you do not have to first select a segment). When on, the mouse icon changes to a Break icon. You can now click any spot on a segment. The clicked spot becomes two coincident vertices, and the segment is split into two parts.

Attach Attaches another spline in the scene to the selected spline. Click the object you want to attach to the currently selected spline object. The object you're attaching to must also be a spline.

For further details, see Attach on page 673.

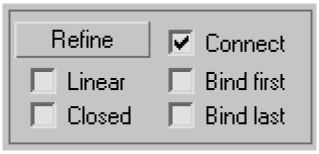
Reorient Reorients the attached spline so that its creation local coordinate system is aligned with the creation local coordinate system of the selected spline.

Attach Mult. Click this button to display the Attach Multiple dialog, which contains a list of all other shapes in the scene. Select the shapes you want to attach to the current editable spline, then click OK.

Cross Section Creates a spline cage out of cross–sectional shapes. Click Cross Section, select one segment then another sub-object segment, splines are created joining the first shape with the second. Continue clicking segments to add them to the cage. All segments must be part of the same object to build cross sections. This functionality is similar to the Cross Section modifier, but here you can determine the order of the cross sections. Spline cage tangency can be defined by choosing Linear, Bezier, Bezier Corner or Smooth in New Vertex Type group.

TIP When you want to move these vertices, turn on Area Selection before you select them. When you transform them, the vertices will stay together.

Refine group



The Refine group includes a number of functions useful for building spline networks for use with the Surface modifier on page 1766.

Refine Lets you add vertices without altering the curvature values of the spline. Click Refine, and then select any number of spline segments to add a vertex each time you click (the mouse cursor changes to a "connect" symbol when over an eligible segment). To finish adding vertices, click Refine again, or right-click in the viewport.

You can also click existing vertices during a refine operation, in which case 3ds Max displays a dialog asking if you want to **Refine** or **Connect** to the vertex. If you choose Connect, 3ds Max will not create a vertex: it simply connects to the existing vertex.

The Refine operation creates a different type of vertex depending on the types of vertices on the endpoints of the segment being refined.

- If the bordering vertices are both Smooth types, the Refine operation creates a Smooth type vertex.
- If the bordering vertices are both Corner types, the Refine operation creates a Corner type vertex.
- If either of the bordering vertices is a Corner or Bezier Corner, the Refine operation creates a Bezier Corner type.
- Otherwise, the operation creates a Bezier type vertex.

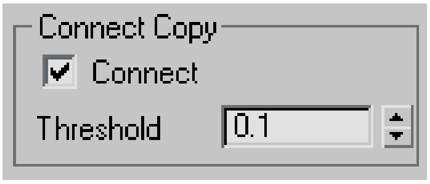
Connect When on, creates a new spline sub-object by connecting the new vertices. When you finish adding vertices with Refine, Connect makes a separate copy of each new vertex and then connects all of the copies with a new spline.

NOTE For Connect to work, you must turn it on before you click Refine.

After turning on Connect and before beginning the refinement process, turn on any combination of these options:

- **Linear** When on, makes all segments in the new spline linear by using Corner vertices. When Linear is off, the vertices used to create the new spline are of the Smooth type.
- **Bind First** Causes the first vertex created in a refinement operation to be bound to the center of the selected segment.
  For more information, see Bound Vertex on page 7931.
- **Closed** When on, connects the first and last vertices in the new spline to create a closed spline. When Closed is off, Connect always creates an open spline.
- **Bind Last** Causes the last vertex created in a refinement operation to be bound to the center of the selected segment.
  For more information, see Bound Vertex on page 7931.

Connect Copy group



Connect Copy When on, Shift+cloning a segment creates a new spline sub-object with additional splines that connect the new segment's vertices to the vertices of the original segment. It is analogous to Shift+cloning edges in Editable Mesh and Editable Poly objects.

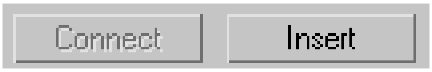
NOTE For Connect Copy to work, you must turn it on before you Shift+Clone.

Threshold Determines the distance soft selection will use when Connect Copy is on. A higher threshold results in more splines being created; a lower threshold results in fewer splines.

End Point Auto-Welding group

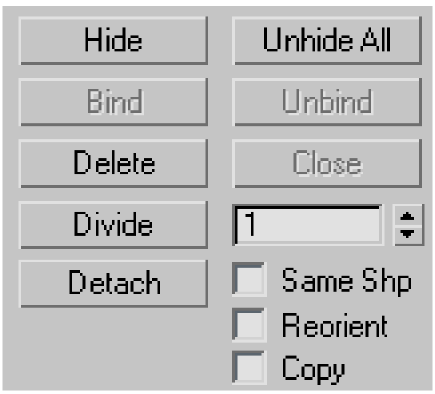
Automatic Welding When Automatic Welding is turned on, an end point vertex that is placed or moved within the threshold distance of another end point of the same spline is automatically welded. This feature is available at the object and all sub-object levels.

Threshold The threshold distance spinner is a proximity setting that controls how close vertices can be to one another before they are automatically welded. Default=6.0.



Insert Inserts one or more vertices, creating additional segments. Click anywhere in a segment to insert a vertex and attach the mouse to the spline. Then optionally move the mouse and click to place the new vertex. Continue moving the mouse and clicking to add vertices. A single click inserts a corner vertex, while a drag creates a Bezier (smooth) vertex.

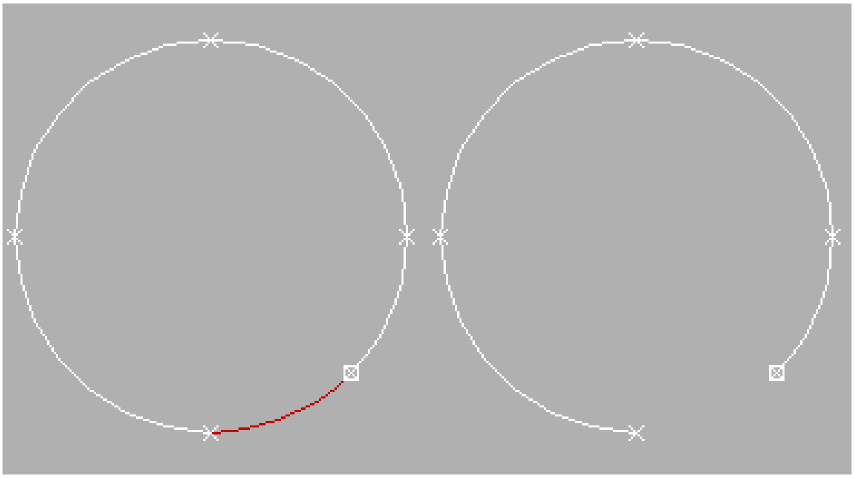
Right-click to complete the operation and release the mouse. At this point, you're still in Insert mode, and can begin inserting vertices in a different segment. Otherwise, right-click again or click Insert to exit Insert mode.



Hide Hides selected segments. Select one or more segments, and then click Hide.

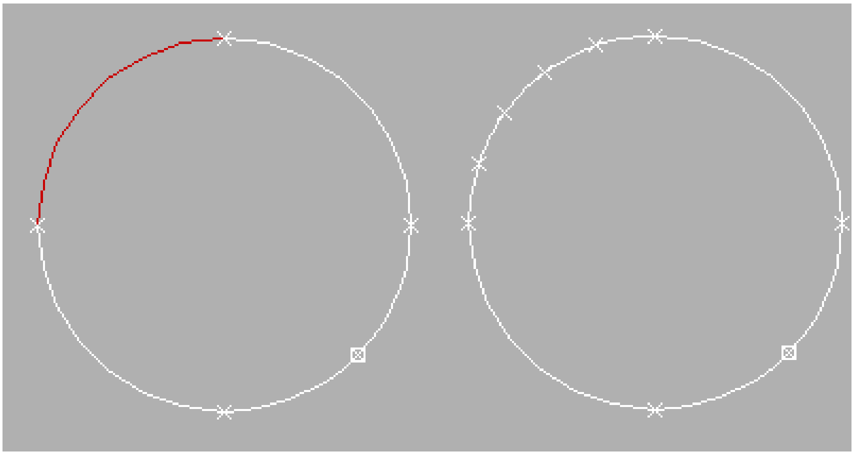
Unhide All Displays any hidden sub-objects.

Delete Deletes any selected segments in the current shape.



Selected and deleted segment

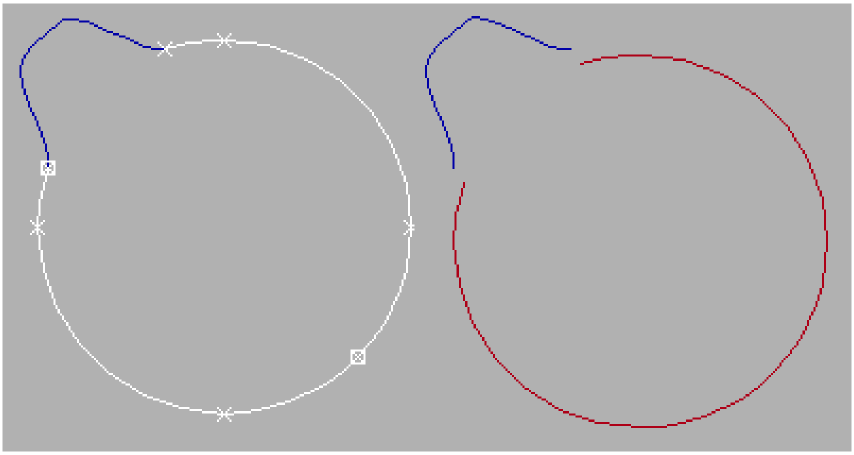
Divide Subdivides the selected segment or segments by adding the number of vertices specified by the spinner. Select one or more segments, set the Divisions spinner (to the button's right), and then click Divide. Each selected segment is divided by the number of vertices specified in the Divisions spinner. The distance between the vertices depends on the segment's relative curvature, with areas of greater curvature receiving more vertices.



Selected and divided segment

Detach Lets you select several segments in various splines and then detach them (or copy them) to form a new shape. Three options are available:

- **Same Shp** (Same Shape) When on, Reorient is disabled, and a Detach operation keeps the detached segment as part of the shape (rather than producing a new shape). If Copy is also on, you end up with a detached copy of the segment in the same location.
- **Reorient** The detached segment copies the position and orientation of the source object's creation Local coordinate system. The new detached object is moved and rotated so that its Local coordinate system is positioned and aligned with the origin of the current active grid.
- **Copy** Copies the detached segment rather than moving it.



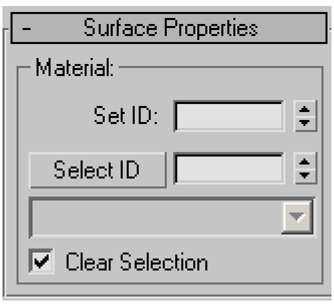
Original and detached splines

Display group

Show selected segs When on, any selected segments are highlighted in red at the Vertex sub-object level. When off (the default), selected segments are highlighted only at the Segment sub-object level.

This feature is useful for comparing complex curves against each other.

Surface Properties rollout



Material group

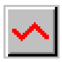
You can apply different material IDs to spline segments (see Material ID on page 8038). You can then assign a multi/sub-object material on page 5720 to such splines, which appears when the spline is renderable, or when used for lathing or extrusion. Be sure to turn on Generate Material IDs and Use Shape IDs when lofting, lathing or extruding.

Set ID Lets you assign a particular material ID number to selected segments for use with multi/sub-object materials and other applications. Use the spinner or enter the number from the keyboard. The total number of available IDs is 65,535.

Select ID Selects the segments or splines corresponding to the Material ID specified in the adjacent ID field. Type or use the spinner to specify an ID, then click the Select ID button.

Select By Name This drop-down list shows the names of sub-materials if an object has a Multi/Sub-object material assigned to it. Click the drop arrow and select a material from the list. The segments or splines that are assigned that material are selected. If a shape does not have a Multi/Sub-Object material assigned to it, the name list will be unavailable. Likewise, if multiple shapes are selected that have an Edit Spline modifier applied to them, the name list is inactive.

Clear Selection When turned on, selecting a new ID or material name forces a deselection of any previously selected segments or splines. When turned off, selections are cumulative so new ID or material name selections add to a previous selection set of segments or splines. Default=on.

Editable Spline (Spline) 

Select an editable spline > Modify panel > Expand the editable spline in the stack display > Spline sub-object level

Select an editable spline > Modify panel > Selection rollout > Spline button

Select an editable spline > Right-click the spline > Tools 1 (upper-left) quadrant of the quad menu > Sub-objects > Spline

While at the Editable Spline (Spline) level, you can select single and multiple splines within a single spline object and move, rotate, and scale them using standard methods.

Procedures

To change spline properties:

- You change the properties of a spline from Line to Curve by right-clicking and choosing Line or Curve from the Tools 1 (upper-left) quadrant of the quad menu.
  Changing the spline property also changes the property of all vertices in the spline:
  - Choosing Line converts vertices to Corners.
  - Choosing Curve converts vertices to Beziers.

Interface

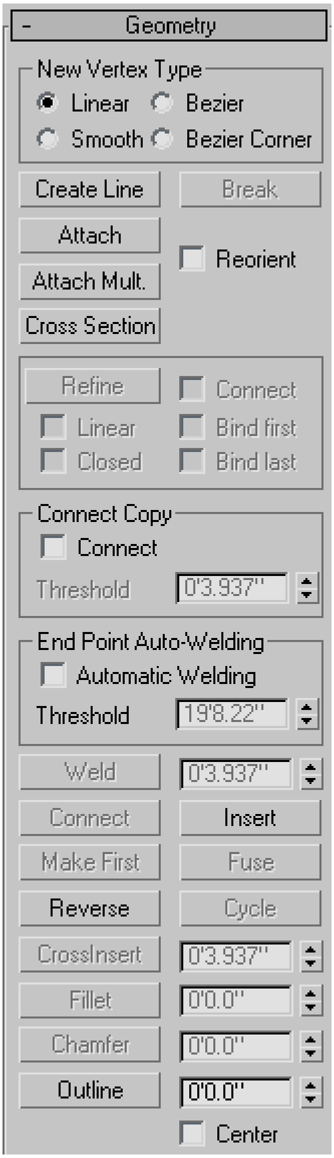
Rendering, Interpolation and Selection rollouts

For information on the Rendering, Interpolation on page 663 and Selection rollout on page 668 settings, see Editable Spline on page 659.

Soft Selection rollout

See Soft Selection Rollout on page 2008 for information on the Soft Selection rollout settings.

Geometry rollout



New Vertex Type group

The radio buttons in this group let you determine the tangency of the new vertices created when you Shift+Clone segments or splines. If you later use Connect Copy, vertices on the splines that connect the original segment or spline to the new one will have the type specified in this group.

This setting has no effect on the tangency of vertices created using tools such as the Create Line button, Refine, and so on.

- **Linear** New vertices will have linear tangency.
- **Smooth** New vertices will have smooth tangency.
  When this option is chosen, new vertices that overlap are automatically welded.
- **Bezier** New vertices will have bezier tangency.
- **Bezier Corner** New vertices will have bezier corner tangency.

Create Line Adds more splines to the selected spline. These lines are separate spline sub-objects; create them in the same way as the line spline on page 620. To exit line creation, right-click or click to turn off Create Line.

Attach Attaches another spline in the scene to the selected spline. Click the object you want to attach to the currently selected spline object. The object you're attaching to must also be a spline.

For further details, see Attach on page 673.

Reorient Reorients the attached spline so that its creation local coordinate system is aligned with the creation local coordinate system of the selected spline.

Attach Mult. Click this button to display the Attach Multiple dialog, which contains a list of all other shapes in the scene. Select the shapes you want to attach to the current editable spline, then click OK.

Cross Section Creates a spline cage out of cross–sectional shapes. Click Cross Section, select one shape then a second shape, splines are created joining the first shape with the second. Continue clicking shapes to add them to the cage. This functionality is similar to the Cross Section modifier, but here you can determine the order of the cross sections. Spline cage tangency can be defined in the New Vertex Type group.

TIP When you edit the spline cage, use Area Selection before selecting your vertices. This will keep their positions together as you transform them.

Connect Copy group

Connect Copy When on, Shift+Cloning a spline creates a new spline sub-object with additional splines that connect the new spline's vertices to the vertices of the original segment. It is analogous to Shift+Cloning edges in Editable Mesh and Editable Poly objects.

NOTE For Connect Copy to work, you must turn it on before you Shift+Clone.

Threshold Determines the distance soft selection uses when Connect Copy is on. A higher value results in more splines being created, a lower value results in fewer splines.

End Point Auto-Welding group

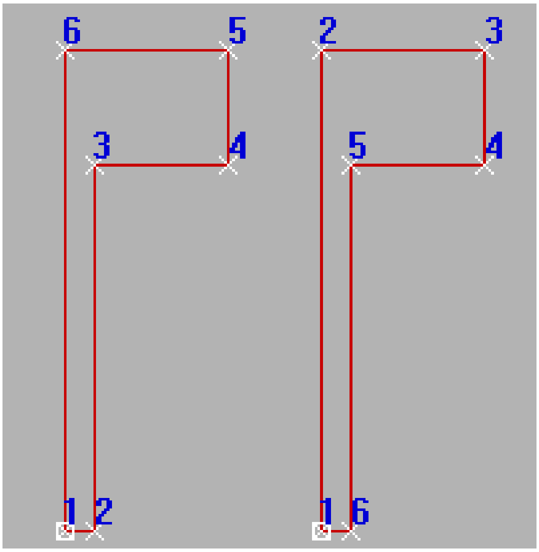
Automatic Welding When Automatic Welding is turned on, an endpoint vertex that is placed or moved within the threshold distance of another endpoint of the same spline is automatically welded. This feature is available at the object and all sub-object levels.

Threshold A proximity setting that controls how close vertices can be to one another before they are automatically welded. Default=6.0.

Insert Inserts one or more vertices, creating additional segments. Click anywhere in a segment to insert a vertex and attach the mouse to the spline. Then optionally move the mouse and click to place the new vertex. Continue moving the mouse and clicking to add vertices. A single click inserts a corner vertex, while a drag creates a Bezier (smooth) vertex.

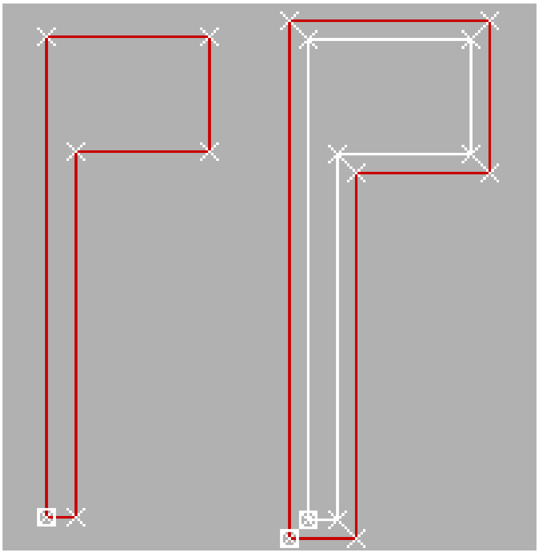
Right-click to complete the operation and release the mouse. At this point, you're still in Insert mode, and can begin inserting vertices in a different segment. Otherwise, right-click again or click Insert to exit Insert mode.

Reverse Reverses the direction of the selected spline. If the spline is open, the first vertex will be switched to the opposite end of the spline. Reversing the direction of a spline is usually done in order to reverse the effect of using the Insert tool at vertex selection level.



Original and reversed splines

Outline Makes a copy of the spline, offset on all sides to the distance specified by the Outline Width spinner (to the right of the Outline button). Select one or more splines and then adjust the outline position dynamically with the spinner, or click Outline and then drag a spline. If the spline is open, the resulting spline and its outline will make a single closed spline.

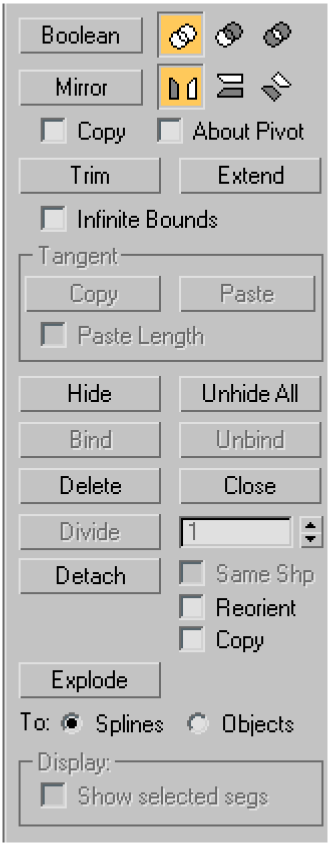


Original and outlined splines

NOTE Normally, if using the spinner, you must first select a spline before using Outline. If, however, the spline object contains only one spline, it is automatically selected for the outlining process.

Center When off (default), the original spline remains stationary and the outline is offset on one side only to the distance specified by Outline Width.

When Center is on, the original spline and the outline move away from an invisible center line to the distance specified by Outline Width.



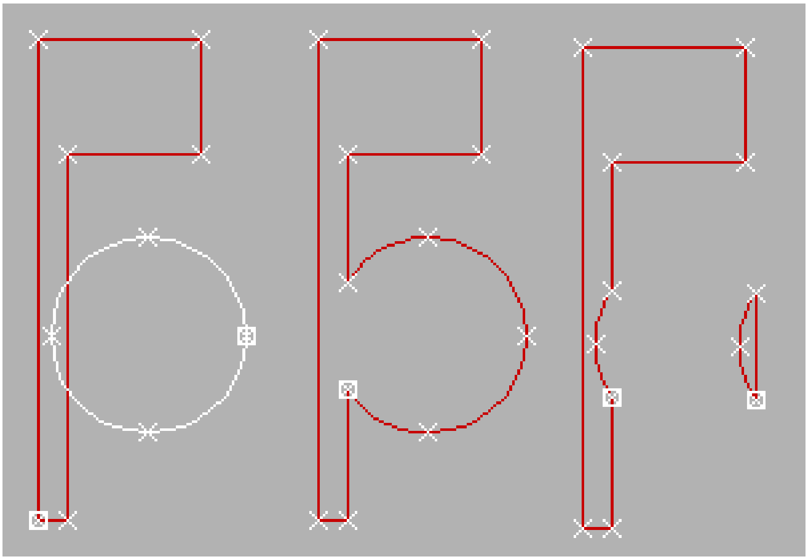
Boolean Combines two closed polygons by performing a 2D Boolean operation that alters the first spline you select, and deletes the second one. Select the first spline, then click the Boolean button and the desired operation, and then select the second spline.

NOTE 2D Booleans only work on 2D splines that are in the same plane.

There are three Boolean operations:

- **Union** Combines two overlapping splines into a single spline, in which the overlapping portion is removed, leaving non-overlapping portions of the two splines as a single spline.

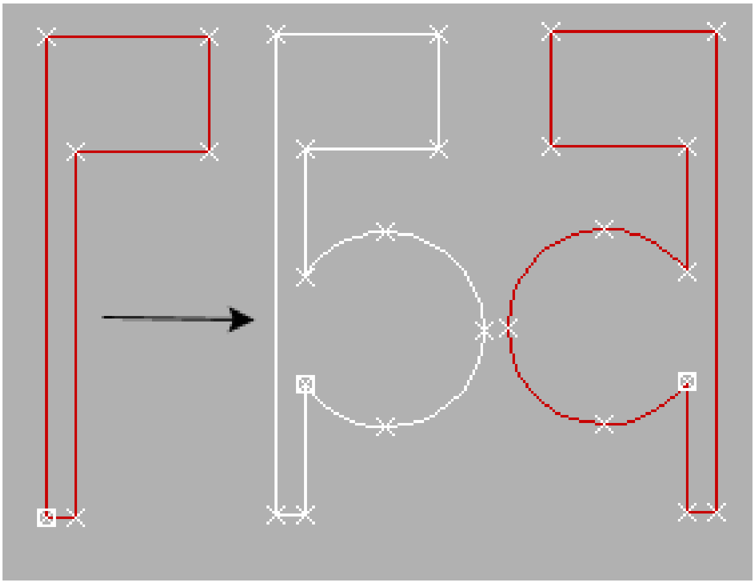
- **Subtraction** Subtracts the overlapping portion of the second spline from the first spline, and deletes the remainder of the second spline.
- **Intersection** Leaves only the overlapping portions of the two splines, deleting the non-overlapping portion of both.



Original splines (left), Boolean Union, Boolean Subtraction, and Boolean Intersection, respectively

Mirror Mirrors splines along the length, width, or diagonally. Click the direction you want to mirror first so it is active, then click Mirror.

- **Copy** When selected, copies rather than moves the spline as it is mirrored.
- **About Pivot** When on, mirrors the spline about the spline object's pivot point (see Pivot on page 3475). When off, mirrors the spline about its geometric center.



Mirrored splines

Trim Use Trim to clean up overlapping segments in a shape so that ends meet at a single point.

To trim, you need intersecting splines. Click the portion of the spline you want to remove. The spline is searched in both directions along its length until it hits an intersecting spline, and deleted up to the intersection. If the section intersects at two points, the entire section is deleted up to the two intersections. If the section is open on one end and intersects at the other, the entire section is deleted up to the intersection and the open end. If the section is not intersected, or if the spline is closed and only one intersection is found, nothing happens.

Extend Use Extend to clean up open segments in a shape so that ends meet at a single point.

To extend, you need an open spline. The end of the spline nearest the picked point is extended until it reaches an intersecting spline. If there is no intersecting spline, nothing happens. Curved splines extend in a direction tangent to the end of the spline. If the end of a spline lies directly on a boundary (an intersecting spline), then it looks for an intersection further along.

Infinite Bounds For the purposes of calculating intersections, turn this on to treat open splines as infinite in length. For example, this lets you trim one linear spline against the extended length of another line that it doesn't actually intersect.

Hide Hides selected splines. Select one or more splines, and then click Hide.

Unhide All Displays any hidden sub-objects.

Delete Deletes the selected spline.

Close Closes the selected spline by joining its end vertices with a new segment.

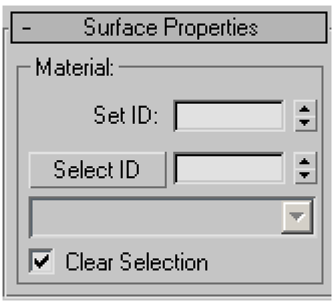
Detach Copies selected splines to a new spline object, and deletes them from the currently selected spline if Copy is clear.

- **Reorient** The spline being detached is moved and rotated so that its creation local coordinate system is aligned with the creation local coordinate system of the selected spline.
- **Copy** When selected, copies rather than moves the spline as it is detached.

Explode Breaks up any selected splines by converting each segment to a separate spline or object. This is a time-saving equivalent of using Detach on each segment in the spline in succession.

You can choose to explode to splines or objects. If you choose Object, you're prompted for a name; each successive new spline object uses that name appended with an incremented two-digit number.

Surface Properties rollout



Material group

You can apply different material IDs (see material ID on page 8038) to splines in shapes containing multiple splines. You can then assign a multi/sub-object material on page 5720 to such shapes, which appears when the spline is renderable, or when used for lathing or extrusion.

Set ID Lets you assign a particular material ID number to selected segments for use with multi/sub-object materials and other applications. Use the spinner or enter the number from the keyboard. The total number of available IDs is 65,535.

Select ID Selects the segments or splines corresponding to the Material ID specified in the adjacent ID field. Type or use the spinner to specify an ID, then click the Select ID button.

Select By Name This drop-down list shows the names of sub-materials if an object has a Multi/Sub-object material assigned to it. Click the drop arrow and select a material from the list. The segments or splines that are assigned that material are selected. If a shape does not have a Multi/Sub-Object material assigned to it, the name list will be unavailable. Likewise, if multiple shapes are selected that have an Edit Spline modifier applied to them, the name list is inactive.

Clear Selection When turned on, selecting a new ID or material name, forces a deselection of any previously selected segments or splines. If turned off, selections are cumulative so new ID or material name selections add to a previous selection set of segments or splines. Default=on.

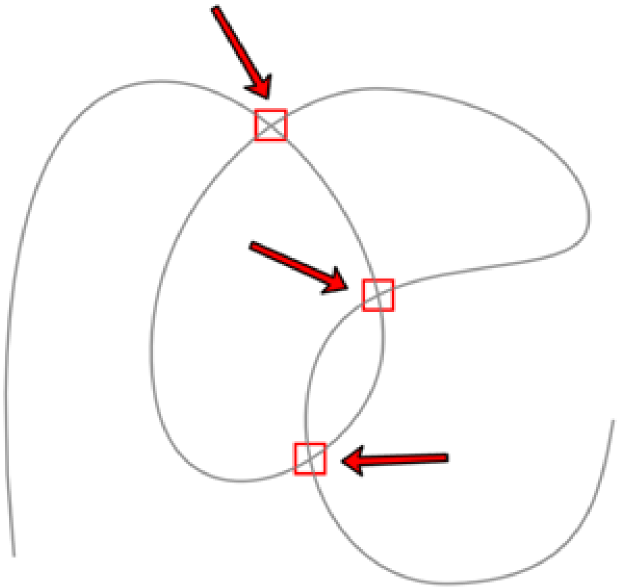
Shape Check Utility

Utilities panel > Utilities rollout > More button Utilities dialog > Shape Check

The Shape Check utility tests spline and NURBS-based shapes and curves for self-intersection and graphically displays any instances of intersecting segments. Self-intersecting shapes used to produce lathed, extruded, lofted, or other 3D objects can result in rendering errors.

The utility is "sticky" in that once you've picked a shape object for it to check, you can pan/zoom viewports and it will continually display the locations of intersecting curves in the shape you pick.

If a shape is animated, moving the time slider will recheck the shape on each frame of the animation, allowing for easy checking of these changing shapes.



Intersection points highlighted by Shape Check

Interface

Pick Object Click this button, and then click the shape for the utility to check. You can pick only spline- and NURBS-based shapes and curves. Points of intersection discovered by the utility are highlighted with red boxes. The text below the button indicates whether any points of intersection occur.

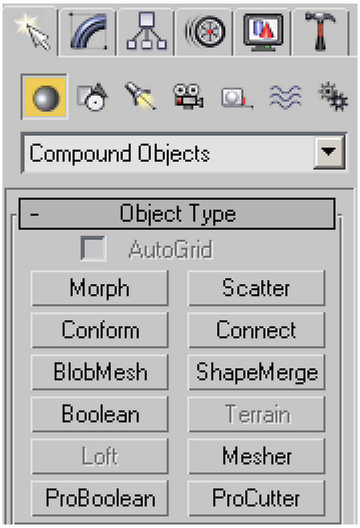
Close Closes the utility.

Compound Objects

Create panel > Geometry > Compound Objects

Create menu > Compound

Compound objects generally combine two or more existing objects into a single object.



Compound objects include the following object types:

Morph on page 709

Scatter on page 717

Conform on page 731

Connect on page 739

BlobMesh on page 745

ShapeMerge on page 752

Boolean on page 757

Terrain on page 774

Loft on page 786

Mesher on page 837

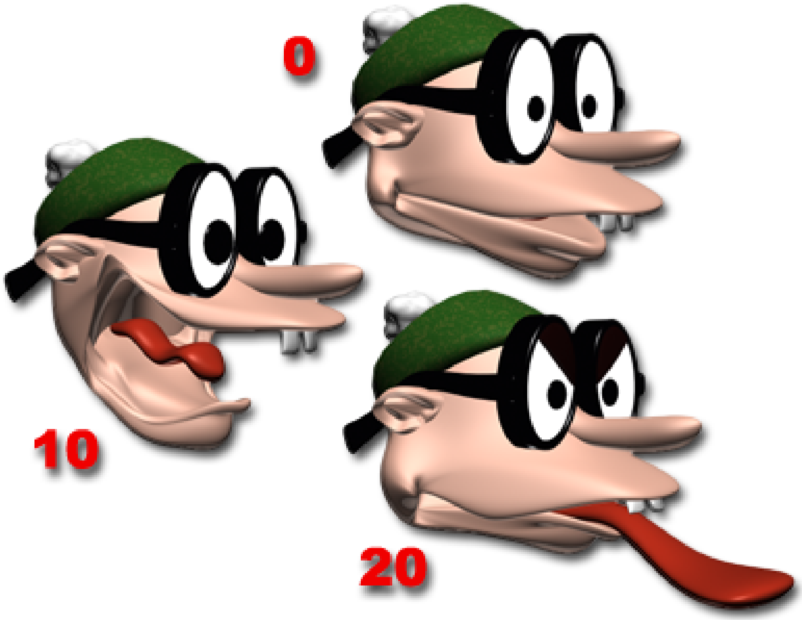
ProBoolean on page 844

ProCutter on page 862

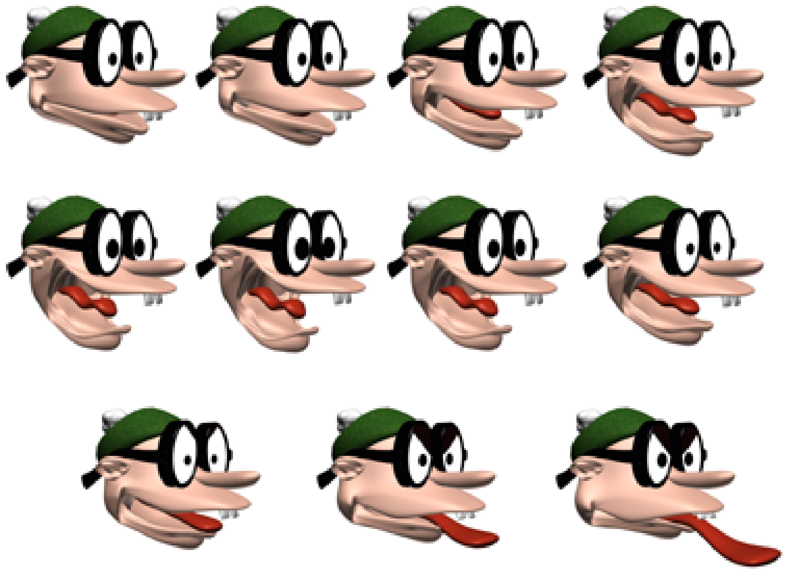
Morph Compound Object

Select an object. > Create panel > Geometry > Compound Objects > Object Type rollout > Morph

Select an object. > Create menu > Compound > Morph



Seed or base object, and the target objects at specific frames



The resulting animation

Morphing is an animation technique similar to tweening in 2D animation. A Morph object combines two or more objects by interpolating the vertices of the first object to match the vertex positions of another object. When this interpolation occurs over time, a morphing animation results.

The original object is known as the *seed* or *base* object. The object into which the seed object morphs is known as the *target* object.

You can morph one seed into multiple targets; the seed object's form changes successively to match the forms of the target objects as the animation plays.

Before you can create a morph, the seed and target objects must meet these conditions:

- Both objects must be mesh, patch, or poly objects.
- Both objects must have an equal number of vertices.

If these conditions don't apply, the Morph button is unavailable.

You can use any kind of object as a morph target, including an animated object or another morph object, as long as the target is a mesh that has the same number of vertices as the seed object.

Creating a morph involves the following steps:

- Model the base object and target objects.
- Select the base object.
- Click Create panel > Geometry > Compound Objects > Morph.
- Add the target objects.
- Animate.

Setting Up the Morph Geometry

Make sure that the objects you want to use as the seed and targets have the same number of vertices.

TIP When you create Loft objects that you want to use as morph seeds and targets, make sure that Morph Capping is on and Adaptive Path Steps and Optimize are turned off. All shapes in the Loft object must have the same number of vertices.

You should also turn off Adaptive and Optimize for other shape-based objects that you want to use with Morph, such as those with Extrude or Lathe modifiers.

WARNING The selected object is permanently converted to a morph object as soon as you click Morph, whether or not you proceed to select a target object. The only way to restore the original object is to undo the Morph click.

Morph Object and Morpher Modifier

There are two ways to set up morphing animations: the Morph compound object and the Morpher modifier.

The Morpher modifier on page 1545 is more flexible because you can add it multiple times at any place in an object's modifier stack display. This flexibility lets you animate the base object or the morph targets before reaching the Morpher modifier, for example with a noise modifier. The Morpher modifier works hand in hand with the Morpher material. The Morpher modifier is the ideal way to morph characters.

The Barycentric Morph controller can be simpler to use in Track View. The Track View display for Compound Morph has only one animation track regardless of the number of targets. Each key on the track represents a morph result based on a percentage of all the targets. For basic morphing needs, Compound Morph may be preferable to the Morpher modifier.

Lastly, you can add the Morpher modifier to the stack of a Compound Morph object.

Procedures

Example: To create a basic morph:

1. On the Create panel > Geometry > Patch Grids > Object Type rollout, click Quad Patch.
2. In the Top viewport, click and drag to create a patch on the left side of the viewport.
3. Right-click the modifier stack display in the Modify panel and select Convert To Editable Patch from the pop-up menu.
4. Right-click the patch, and then click Move in the Transform quadrant of the quad menu.
5. In the Top viewport, hold Shift and drag with the patch to create a copy on the right side of the viewport.
6. On the Modify panel > Selection rollout, go to the Vertex sub-object level.
7. In the Front viewport, select and move vertices on the selected patch to alter its shape.
8. On the Modify panel, in the stack display, click Editable Patch again to return to the top level.
9. Select the original patch in the viewports.
10. On the Create panel > Geometry > Compound Objects > Objects Type rollout, click Morph.
11. On the Pick Targets rollout, click Pick Target.
12. In the viewports, click the second patch object.

    Both patch objects are listed in the Morph Targets list.
13. Click Modify panel.

    Morph displays above the Editable Patch in the modifier stack.

14 Move the time slider to frame 10.

15 In the Morph Targets list, click M_QuadPatch01.

16 On the Current Targets rollout, click Create Morph Key.

On the track bar, a key is displayed at frame 10.

17 On the track bar, right-click the key at frame 10 and click QuadPatch01:Morph in the menu.

A Key Info dialog displays.

18 On the Key Info dialog, select M_QuadPatch01 from the list.

19 On the Key Info dialog, drag the percentage spinner.

The base object changes shape.

20 Close the Key Info dialog and drag the time slider back and forth. The patch morphs its shape.

To select the targets for a morph:

1 Select the seed object.

2 On the Create panel > Geometry > Compound Objects, click Morph.

The name of the seed object is displayed at the top of the Morph Targets list on the Current Targets rollout.

3 On the Pick Targets rollout, choose the method for creating targets: Reference, Move, Copy, or Instance.

4 Click Pick Target.

5 Select one or more target objects in the viewports.

As you select each target, its name is added to the Morph Targets list. If an object can't be a target (for example, if it has a different number of vertices than the morph seed), you can't select it.

If you select a target object while you are not at frame 0, creating the target also creates a morph key. You can create additional morph keys from targets you've already selected, as described in the following procedure.

To create morph keys from existing targets:

1 Drag the time slider to the frame where you want to place the morph key.

NOTE The Auto Key button does not need to be on to set morph keys.

2 Highlight the name of a target object on the Morph Targets list.

The Create Morph Key button is available only when a target object name is selected.

3 Click Create Morph Key.

3ds Max places a morph key at the active frame.

4 To preview the effect of the morph, drag the time slider back and forth. You can view and edit the morph keys in Track View, which also lets you view the morph's target object parameters.

Interface

Pick Targets rollout

When you pick target objects, you designate each target as a Reference, Move (the object itself), Copy, or Instance. Base your selection on how you want to use the scene geometry after you create the morph.

Pick Target Use this button to designate the target object or objects.

Reference/Copy/Move/Instance Lets you specify how the target is transferred to the compound object. It can be transferred either as a reference on page 8106, a copy, an instance on page 8014, or it can be moved, in which case the original shape is not left behind.

- Use Copy when you want to reuse the target geometry for other purposes in the scene.
- Use Instance to synchronize morphing with animated changes to the original target object.

- Use Move if you've created the target geometry to be only a morph target, and have no other use for it.

You can use an animated object or another morph as the target of a morph.

Current Targets rollout

Morph Targets Displays a list of the current morph targets.

Morph Target Name Use this field to change the name of the selected morph target in the Morph Targets list.

Create Morph Key Adds a morph key for the selected target at the current frame.

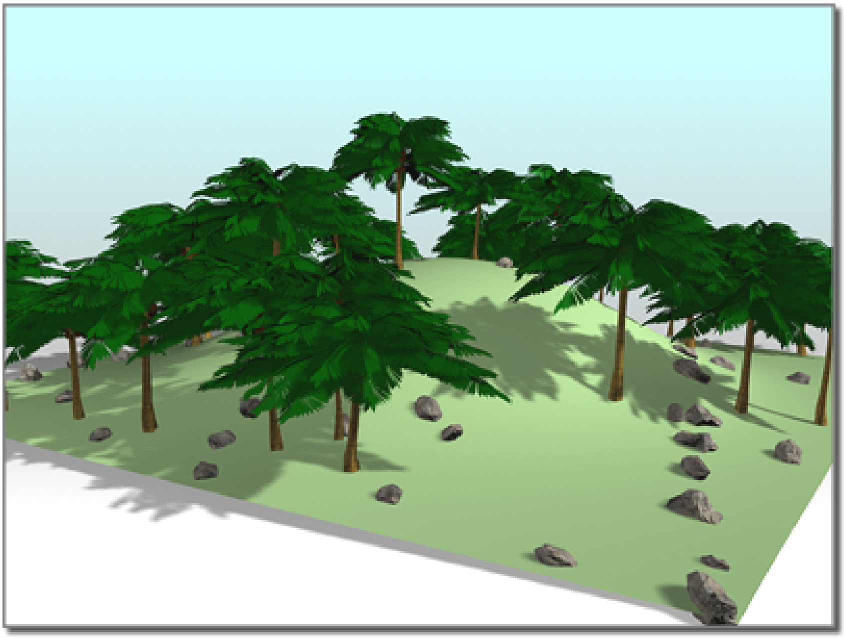
Delete Morph Target Deletes the currently highlighted morph target. If morph keys reference the deleted target, then those keys are deleted as well.

Scatter Compound Object

Select an object. > Create panel > Geometry > Compound Objects > Object Type rollout > Scatter

Select an object. > Create menu > Compound > Scatter

Scatter is a form of compound object that randomly scatters the selected *source* object either as an array, or over the surface of a *distribution* object.



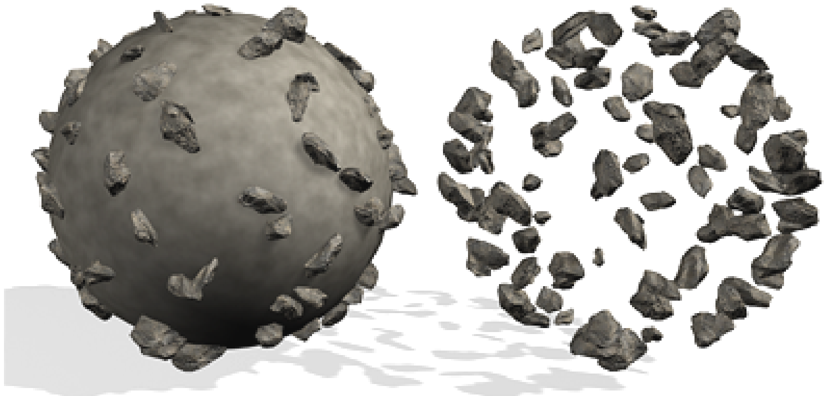
The plane of the hill is used to scatter the trees and two different sets of rocks.

Procedures

To create a Scatter object:

1. Create an object to be used as a source object.
2. Optionally, create an object to be used as a distribution object.
3. Select the source object, and then click Scatter in the Compound Objects panel.

NOTE The source object must be either a mesh object or an object that can be converted to a mesh object. If the currently selected object is invalid, the Scatter button is unavailable.



Results of scattering source object with distribution object visible (above) and hidden (below)

You now have two choices. You can either scatter the source object as an array without using a distribution object, or use a distribution object to scatter the object. See the following procedures.

To scatter the source object without a distribution object:

1. Choose Use Transforms Only in the Scatter Objects rollout > Distribution group.
2. Set the Duplicates spinner to specify the desired total number of duplicates of the source object.
3. Adjust the spinners on the Transforms rollout to set random transformation offsets of the source object.

To scatter the source object using a distribution object:

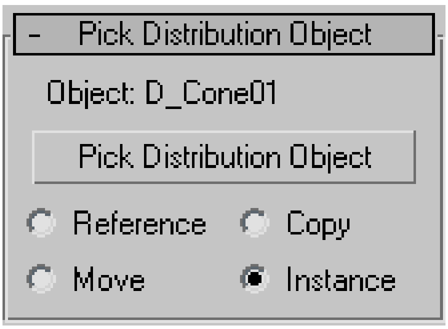
1. Make sure the source object is selected.
2. Choose the method by which you want to clone the distribution object (Reference, Copy, Move, or Instance.)
3. Click Pick Distribution Object, and then select the object you want to use as a distribution object.
4. Make sure that Use Distribution Object on the Scatter Object rollout is chosen.
5. Use the Duplicates spinner to specify the number of duplicates. (This is not necessary if you're using the All Vertices, All Edge Midpoints or All Face Centers distribution methods.)
6. Choose a distribution method in the Scatter Object rollout > Distribute Object Parameters group under Distribute Using.
7. Optionally, adjust the Transform spinners to randomly transform the duplicates.
8. If the display is too slow, or the meshes too complicated, consider choosing Proxy on the Display rollout or decreasing the percentage of displayed duplicates by reducing the Display percentage.

Most of the spinner values are animatable, so you can animate things like the number of duplicates, their transformations, and so on.

Scatter objects (the grass) with a high number of duplicates

Interface

Pick Distribution Object rollout



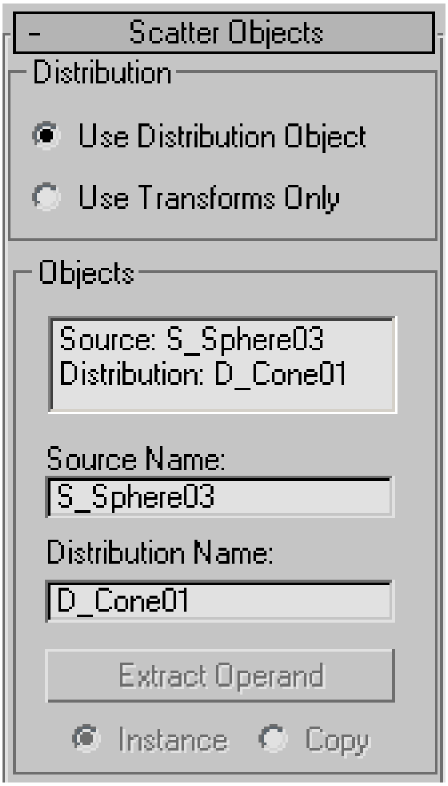
Contains the options for selecting a distribution object.

Object Displays the name of the distribution object selected with the Pick button.

Pick Distribution Object Click this button, then click an object in the scene to specify it as a distribution object.

Reference/Copy/Move/Instance Lets you specify how the distribution object is transferred to the scatter object. It can be transferred either as a reference on page 8106, a copy, an instance on page 8014, or moved, in which case the original shape is not left behind.

Scatter Objects rollout



The options on this rollout let you specify how the source object is scattered, and let you access the objects that make up the compound Scatter object.

Distribution group

These two options let you choose the basic method of scattering the source object.

Use Distribution Object Scatters the source object based on the geometry of the distribution object.

Use Transforms Only This options doesn't need a distribution object. Instead, duplicates of the source object are positioned using the offset values on the Transforms rollout. If all of the Transform offsets remain at 0, you won't see the array because the duplicates occupy the same space.

Objects group

Contains a list window showing the objects that make up the Scatter object.

List Window Click to select an object in the window so that you can access it in the Stack. For example, if your distribution object is a sphere, you can click Distribution: D_Sphere01, open the Stack list, and select Sphere to access the sphere's parameters.

Source Name Lets you rename the source object within the compound Scatter object.

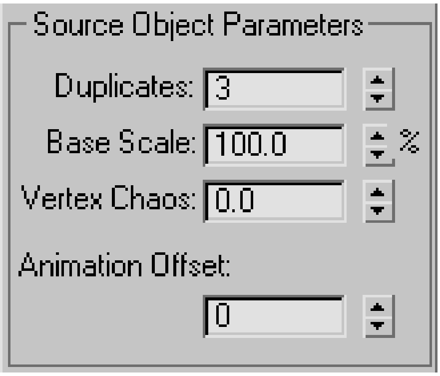
Distribution Name Lets you rename the distribution object.

Extract Operand Extract a copy or an instance of the selected operand. Choose an operand in the list window to enable this button.

NOTE This button is available only on the Modify panel. You can't extract an operand while the Create panel is active.

Instance/Copy This option lets you specify how the operand is extracted: as either an instance on page 8014 or a copy.

Source Object Parameters group



These options affect the source object locally.

Duplicates Specifies the number of scattered duplicates of the source object.

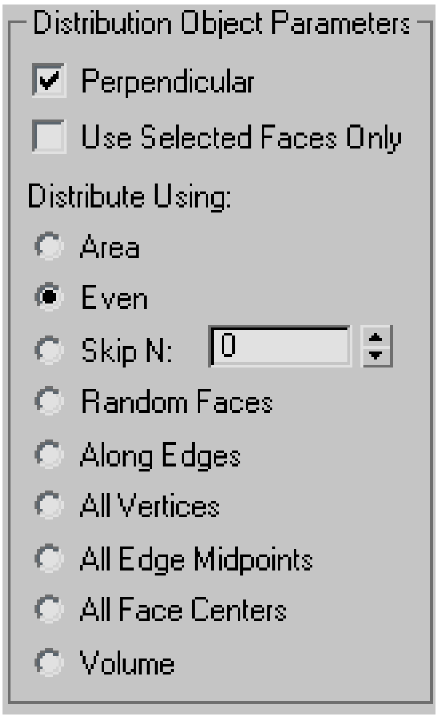
This number is set to 1 by default, but you can set it to 0 if you want to animate the number of duplicates, beginning with none. Note that the Duplicates number is ignored if you're distributing the duplicates using either Face Centers or Vertices. In these cases, one duplicate is placed at each vertex or face center, depending on your choice.

Base Scale Alters the scale of the source object, affecting each duplicate identically. This scale occurs before any other transforms.

Vertex Chaos Applies a random perturbation to the vertices of the source object.

Animation Offset Lets you specify the number of frames by which each source object duplicate's animation is offset from the previous duplicate. You can use this feature to produce wave-type animation. At the default setting of 0, all duplicates move identically.

Distribution Object Parameters group



These options affect how the duplicates of the source object are arranged, relative to the distribution object. These options have an effect only when a distribution object is used.

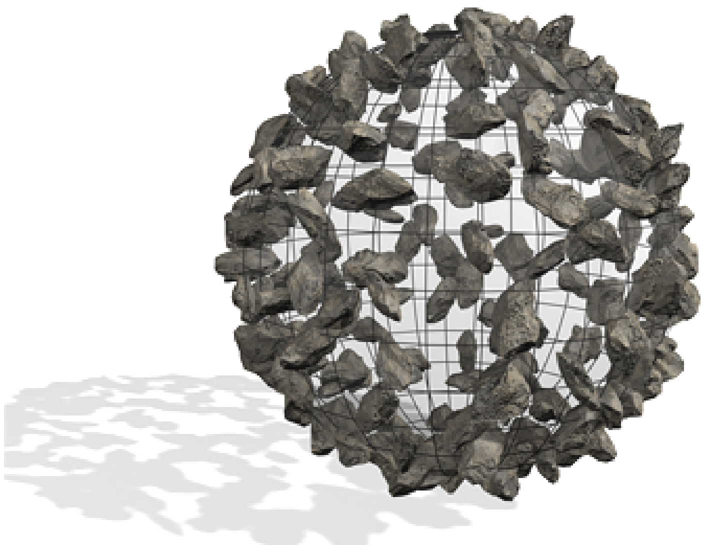
Perpendicular When on, orients each duplicate object perpendicular to its associate face, vertex, or edge in the distribution object. When off, the duplicates maintain the same orientation as the original source object.

Use Selected Faces Only When on, limits distribution to the selected faces passed up the Stack. Perhaps the easiest way to do this is to use the Instance option when picking the distribution object. You can then apply a Mesh Select modifier to the original object and select only those faces you want to use for the distribution of the duplicates.

Distribute Using

The following options let you specify how the geometry of the distribution object determines the distribution of the source object. These options are ignored if you're not using a distribution object.

Area Distributes duplicate objects evenly over the total surface area of the distribution object.



Objects distributed over a spherical surface with Area turned on

Even Divides the number of faces in the distribution object by the number of duplicates, and skips the appropriate number of faces in the distribution object when placing duplicates.

Skip N Skips N number of faces when placing duplicates. The editable field specifies how many faces to skip before placing the next duplicate. When set to 0, no faces are skipped. When set to 1, every other face is skipped, and so on.

Random Faces Applies duplicates randomly over the surface of the distribution object.

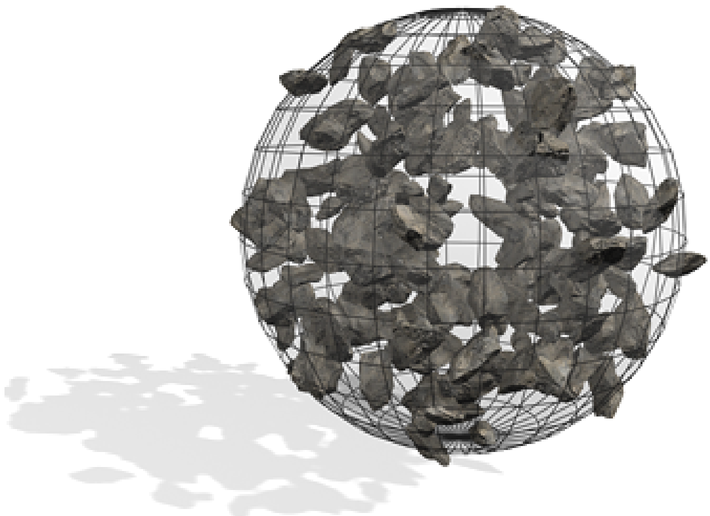
Along Edges Assigns duplicates randomly to the edges of the distribution object.

All Vertices Places a duplicate object at each vertex in the distribution object. The Duplicates value is ignored.

All Edge Midpoints Places a duplicate at the midpoint of each segment edge.

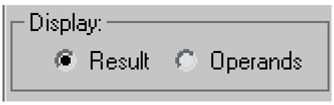
All Face Centers Places a duplicate object at the center of each triangular face on the distribution object. The Duplicates value is ignored.

Volume Scatters objects throughout the distribution object's volume. All other options restrict distribution to the surface. Consider turning on Display rollout > Hide Distribution Object with this option.



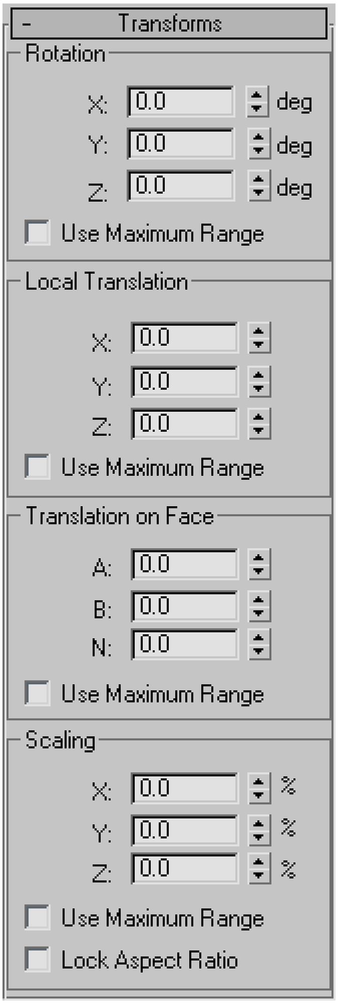
Objects fill a spherical volume with Volume turned on

Display group



Result/Operands Choose whether to display the results of the scatter operation or the operands before the scattering.

Transforms rollout



The settings in the Transforms rollout let you apply random transform offsets to each duplicate object. The values in the transform fields specify a maximum offset value that's applied randomly with a positive or negative value to each duplicate. Thus, if you set a rotation angle of 15 degrees, duplicates are rotated randomly from -15 to +15 degrees. For example, one duplicate might be rotated 8 degrees, another -13, another 5, and so on. You can use the Transform settings with or without a distribution object. When there is no distribution object, you must adjust the Transform settings in order to see the duplicates.

Rotation group

Specifies random rotation offsets.

X, Y, Z deg Enter the maximum random rotational offset you want about the local X, Y, or Z axis of each duplicate.

Use Maximum Range When on, forces all three settings to match the maximum value. The other two settings become disabled, and the setting containing the maximum value remains enabled.

Local Translation group

Specifies translation of the duplicates along their local axes.

X, Y, Z Enter the maximum random movement you want along the X, Y, or Z axis of each duplicate.

Use Maximum Range When on, forces all three settings to match the maximum value. The other two settings become disabled, and the setting containing the maximum value remains enabled.

Translation on Face group

Lets you specify the translation of duplicates along barycentric on page 7920 face coordinates of the associate face in the distribution object. These settings have no effect if you're not using a distribution object.

A, B, N The first two settings specify the barycentric coordinates on the surface of the face, while the N setting sets the offset along the normal of the face.

Use Maximum Range When on, forces all three settings to match the maximum value. The other two settings become disabled, and the setting containing the maximum value remains enabled.

Scaling group

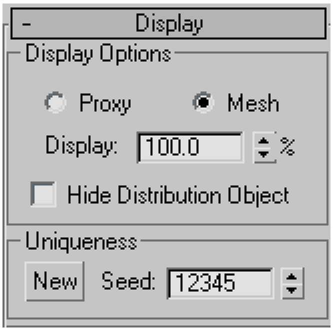
Lets you specify the scaling of duplicates along their local axes.

X, Y, Z % Specifies the percent of random scaling along the X, Y, or Z axis of each duplicate.

Use Maximum Range When on, forces all three settings to match the maximum value. The other two settings become disabled, and the one containing the maximum value remains enabled.

Lock Aspect Ratio When on, maintains the original aspect ratio of the source object. Typically, this provides uniform scaling of duplicates. When Lock Aspect Ratio is off, and any of the X, Y, and Z settings contain values greater than 0, the result is non-uniform scaling of duplicates because the values represent random scaling offsets in both positive and negative directions.

Display rollout



Provides options that affect the display of the Scatter object.

Display Options group

These options affect the display of the source and destination objects.

Proxy Displays the source duplicates as simple wedges and speeds up viewport redraws when manipulating a complex Scatter object. This has no effect on the rendered image, which always displays the mesh duplicates.

Mesh Displays the full geometry of the duplicates.

Display % Specifies the percentage of the total duplicate objects that appear in the viewports. This has no effect on the rendered scene.

Hide Distribution Object Hides the distribution object. The hidden object does not appear in the viewport or in the rendered scene.

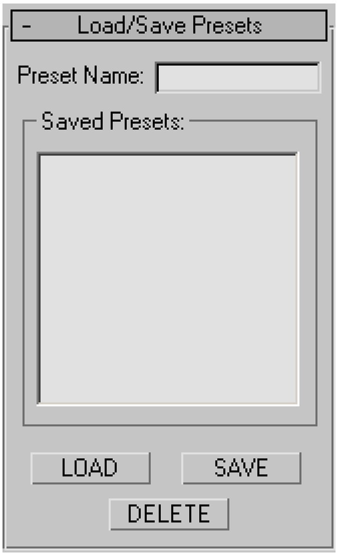
Uniqueness group

Lets you set a seed number upon which the random values are based. Thus, altering this value changes the overall effect of the scattering.

New Generates a new, random seed number.

Seed Use this spinner to set the seed number.

Load/Save Presets rollout



Lets you store preset values to use in other Scatter objects. For example, after setting all of your parameters for a specific Scatter object and saving the settings under a specific name, you can then select another Scatter object and load the preset values into the new object.

Preset Name Lets you define a name for your settings. Click the Save button to save the current settings under the preset name.

Saved Presets group

A list window containing saved preset names.

LOAD Loads the preset currently highlighted in the Saved Presets list.

SAVE Saves the current name in the Preset Name field and places it in the Saved Presets window.

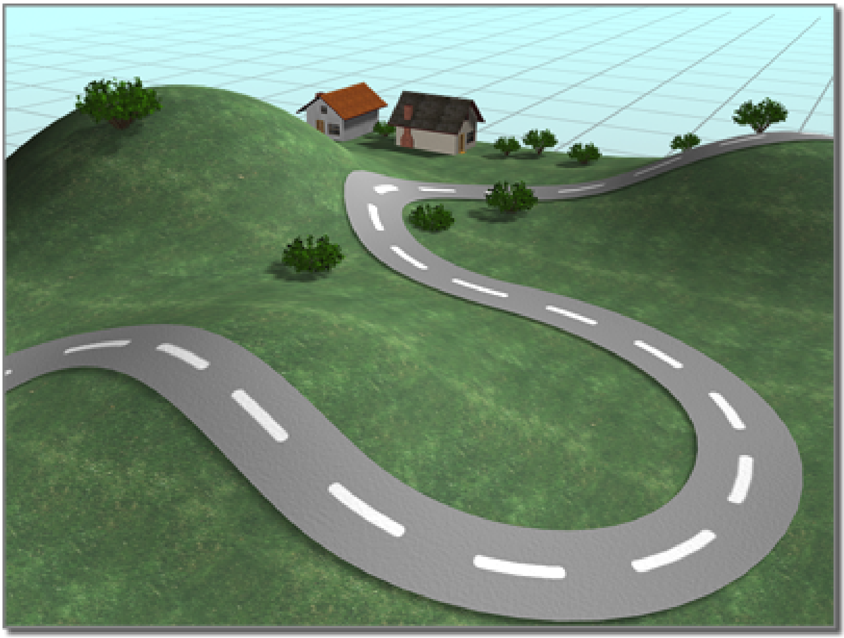
DELETE Deletes the selected items in the Save Presets window.

NOTE Animated parameter values subsequent to frame 0 are not stored.

Conform Compound Object

Select an object. > Create panel> Geometry > Compound Objects > Object Type rollout > Conform

Select an object. > Create menu > Compound > Conform



Conform fits the road to the surface of the hills.

Conform is a compound object created by projecting the vertices of one object, called the Wrapper, onto the surface of another object, called the Wrap-To. There is also a space-warp version of this function; see Conform space warp on page 2785.

Because the space-warp version is somewhat easier to use, it's a good idea to read that topic first, try the example, and then return here. This topic provides additional methods of projecting the wrapper vertices.

NOTE This tool gives you the ability to morph between any two objects, regardless of the number of vertices in each object. See Vertex Projection Direction group on page 735 for more information.

Procedures

Example: To create a Conform object:

1. Position two objects, one of which will be the Wrapper, and the other the Wrap-To. (For this example, create a box as the Wrap-To object, and then create a larger sphere that completely surrounds it. The sphere will be the Wrapper.)

2. Select the Wrapper object (the sphere), and click Create panel> Geometry > Compound Objects > Object Type rollout > Conform button.

   NOTE Both objects used in Conform must be either mesh objects or objects that can be converted to mesh objects. If the selected Wrapper object is invalid, the Conform button is unavailable.

   ---

3. Specify the method of vertex projection in the Vertex Projection Direction group. (Use Along Vertex Normals for this example.)

   NOTE If you were to choose Use Active Viewport, you would next activate whichever viewport looks in the direction that you want to project the vertices. For example, if the Wrapper hovered over a Wrap-To terrain on the home plane, you'd activate the Top viewport.

   ---

4. Choose Reference, Copy, Move, or Instance to specify the type of cloning to perform on the Wrap-To object. (Choose Instance for this example.)

5. Click Pick Wrap-To Object, and then click the object onto which to project the vertices. (You can press the H key and use the Pick Object dialog on page 228 to select the box.)

   The list windows display the two objects, and the compound object is created with the Wrapper object conforming to the Wrap-To object. (In the example, the sphere is wrapped into the shape of the box.)

6. Use the various parameters and settings to alter the vertex projection direction, or adjust the vertices that are being projected.

To project a road onto terrain:

1. Create the road and terrain objects.

TIP You can quickly make a terrain by creating a patch grid on page 2067 and applying the Noise modifier on page 1573 to it. For the road, you can use a Loft compound object on page 786 by lofting a rectangle along a curved line. Both objects must have a sufficient level of detail to conform smoothly.

2 Orient both the road and the terrain so you are looking straight down at them in the Top viewport. Position the road so it's completely above the terrain (higher on the world Z axis).

NOTE For the conform projection to work correctly, the road should not extend beyond the boundaries of the terrain when viewed in the Top viewport.

3 Select the road object.

4 Click Conform.

5 In the Pick Wrap-To Object rollout, make sure the Instance option is selected.

6 Click Pick Wrap-To Object, and click the terrain.

   An instance of the terrain object is created, with the same object color as the road.

7 Activate the Top viewport. In the Parameters rollout > Vertex Projection Direction group, choose Use Active Viewport, and click Recalculate Projection.

8 In the Update group, turn on Hide Wrap-To Object.

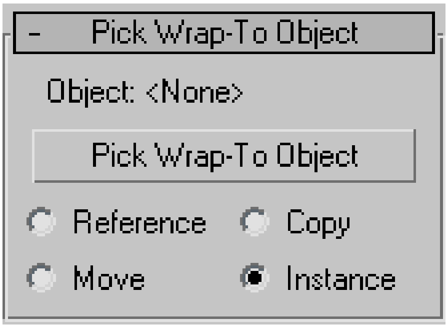
   This hides the instance of the terrain so you can clearly see the road projected onto it.

   The Parameters rollout > Wrapper Parameters group > Standoff Distance value sets the number of units by which the road sits above the terrain along the world Z axis.

9 If necessary, adjust the Standoff Distance to raise or lower the road.

Interface

Pick Wrap-To Object rollout



Object Displays the name of the selected Wrap-To object.

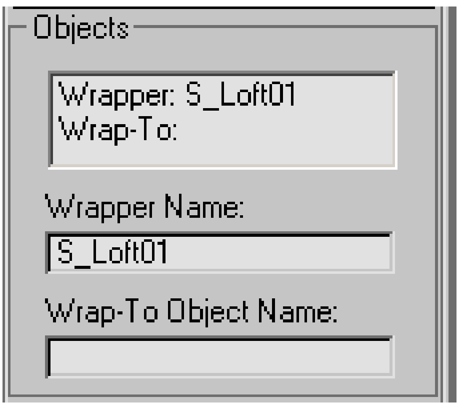
Pick Wrap-To Object Click this button, and then select the object to which you want the current object to wrap.

Reference/Copy/Move/Instance This option lets you specify how the Wrap-To object is transferred to the Conform object. It can be transferred either as a reference on page 8106, a copy, an instance on page 8014, or it can be moved, in which case the original is not left behind.

Parameters rollout

Contains all parameters for the Conform object.

Objects group



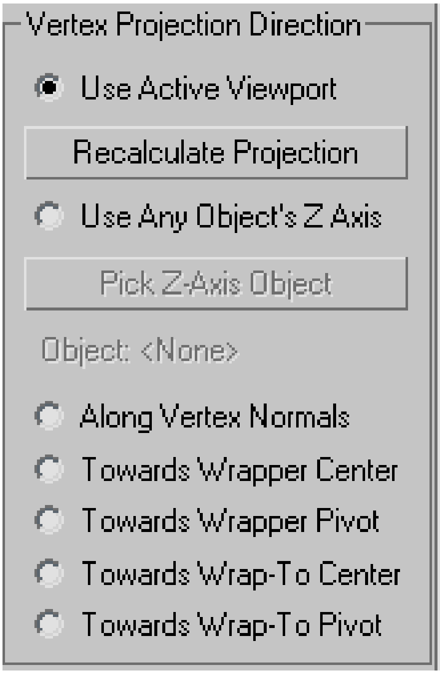
Provides a list window and two edit fields that let you navigate the compound object and rename its components.

List Window Lists the Wrapper and the Wrap-To objects. Click to select an object in the window so that you can access it in the Modifier stack.

Wrapper Name Lets you rename the wrapper object within the compound Conform object.

Wrap-To Object Name Lets you rename the Wrap-To object.

Vertex Projection Direction group



Choose one of these seven options to determine the projection of the vertices.

Use Active Viewport The vertices are projected away (inward) from the active viewport.

Recalculate Projection Recalculates the projection direction for the currently active viewport. Because the direction is initially assigned when you pick the Wrap-To object, if you want to change viewports after assignment, click this button to recalculate the direction based on the new active viewport.

Use Any Object's Z Axis Lets you use the local Z axis of any object in the scene as a direction. Once an object is assigned, you can alter the direction of vertex projection by rotating the direction object.

Pick Z-Axis Object Click this button, and then click the object you want to use to indicate the direction of the projection source.

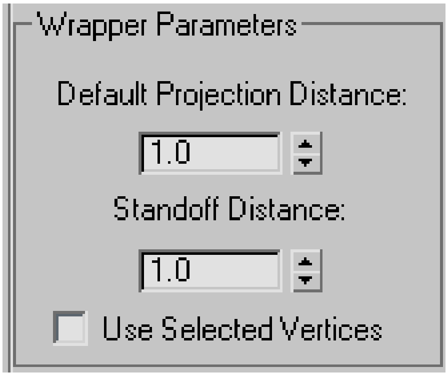
Object Displays the name of the direction object.

- **Along Vertex Normals** Projects the vertices of the Wrapper object inward along the reverse direction of its vertex normals. A vertex normal is a vector produced by averaging the normals of all faces attached to that vertex. If the Wrapper object encloses the Wrap-To object, the Wrapper takes on the form of the Wrap-To object.
- **Towards Wrapper Center** Projects the vertices toward the bounding center of the Wrapper object.
- **Towards Wrapper Pivot** Projects the vertices toward the original pivot center of the Wrapper object.
- **Towards Wrap-To Center** Projects the vertices toward the bounding center of the Wrap-To object.
- **Towards the Wrap-To Pivot** Projects the vertices toward the pivot center of the Wrap-To object.

NOTE Towards Wrapper Pivot and Towards the Wrap-To Pivot operate on the position of the original pivot point of the object before the Conform object is created. Once you create the Conform object, it's a new compound object with a single pivot point.

TIP You can animate the conforming effect by morphing between the compound object and a previously made copy of the original wrapper object. To do this, however, you must turn on Hide Wrap-To Object in the Update group so that the original object and the compound object have the same number of vertices. Using this technique, you can effectively morph between two objects with a different number of vertices.

Wrapper Parameters group



Provides controls that determine how far the vertices are projected.

Default Projection Distance The distance a vertex in the Wrapper object will move from its original location if it does not intersect the Wrap-To object.

Standoff Distance The distance maintained between the vertex of the Wrapper object and the surface of the Wrap-To object. For example, if you set Standoff Distance to 5, the vertices can be pushed no closer than 5 units from the surface of the Wrap-To object.

Use Selected Vertices When turned on, only the selected vertex sub-objects of the Wrapper object are pushed. When turned off, all vertices in the object are pushed, regardless of the Modifier stack selection. To access the Modifier stack of the Wrapper object, select the Wrapper object in the list window, open the Modifier stack, and select the base object name. At this point you can apply a Mesh Select modifier, for example, and select the vertices you want to affect.

Update group

The items in this group determine when the projection for the compound object is recalculated. Because complex compound objects can slow performance, you can use these options to avoid constant calculation.

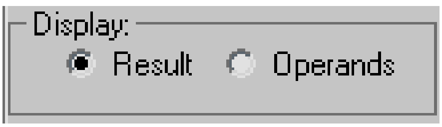
- **Always** The object is updated constantly.
- **When Rendering** The object is recalculated only when the scene is rendered.
- **Manually** Activates the Update button for manual recalculation.

Update Recalculates the projection.

Hide Wrap-To Object When on, hides the Wrap-To object.

Display group

Determines whether the shape operands are displayed.



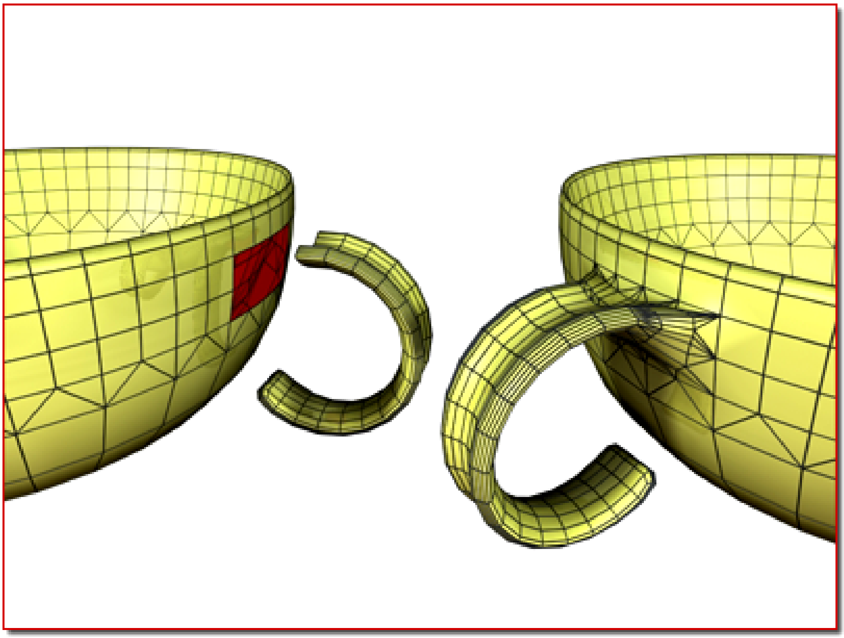
- **Result** Displays the result of the operation.
- **Operands** Displays the operands.

Connect Compound Object

Select an object. > Create panel > Geometry > Compound Objects > Object Type rollout > Connect

Select an object. > Create menu > Compound > Connect

The Connect compound object lets you connect two or more objects between "holes" in their surfaces. To do this, you delete faces in each object to create one or more holes in their surfaces, position them so that the holes face one another, and then apply Connect.



Left: Before connect

Right: After connect

NOTE Connect is not suited to NURBS objects, because they convert into many separate meshes instead of one big mesh. The workaround is simple: apply a Weld modifier to the NURBS object (thus converting it to a mesh and zipping up its seams) before using it as part of a connect.

Connect generates the best mapping coordinates it can for the bridges between the various holes in the meshes. While some ideal cases, such as a cylinder above another cylinder, can generate good UVW map interpolations, most cases cannot. You'll need to apply mapping to the bridge faces with a UVW Map modifier on page 1931.

Vertex colors, on the other hand, interpolate smoothly.

Notes:

- You can use Connect on objects that have multiple sets of holes. Connect will do its best to match up the holes between the two objects.
- The mapping coordinates assigned to the original two objects are maintained to the extent possible. You might find irregularities in the bridged area, depending on the complexity and difference between the two original sets of mapping coordinates and the types of geometry.

Procedures

To create a Connect object:

1. Create two mesh objects.
2. Delete faces on each to create holes where you want to bridge the objects.

   Position the objects so that the normals of the deleted faces of one object point toward the normals of the deleted faces of the other object (assuming that deleted faces could have normals).
3. Select one of the objects. On the Create panel > Geometry > Compound Object Type rollout, click Connect.
4. Click the Pick Operand button, and then select the other object.
5. Faces are generated connecting the holes in the two objects.
6. Adjust the connection with the various options.

Example: To connect two cylinders:

1. Create a cylinder with a radius of 15 and a height of 30. Use the default settings for the remaining parameters.
2. Create a second cylinder centered on the first with a radius of 30, a height of 30, and 13 sides. (The fewer sides are to demonstrate the mesh interpolation in the connection.)
3. Move the first, narrower cylinder straight up along Z so its bottom cap is about 15 units above the top cap of the larger cylinder.

4 Convert both cylinders to editable meshes.

5 Delete the lower cap of the upper cylinder, and the upper cap of the bottom cylinder. (Hint: Go to Editable Mesh (Polygon) mode, select each end in turn, and then press the Delete key.)

6 Exit sub-object mode, select the lower cylinder, and click Connect.

7 Click the Pick Operand button, and then click the upper cylinder.

   New faces are created that span the openings in the two cylinders.

Example continued: To try out some options and create animation:

1 Go to the Modify panel and increase the Segments spinner to **5** or more.

   As the segments increase, the connection becomes curved.

2 Set the Tension spinner to **0** to straighten the connecting surface, increase it to 1, and then return it to **0.5**.

3 Try different combinations of the Bridge and Ends options.

4 Select the upper cylinder, turn on the Auto Key button, and apply various transforms at different frames.

5 Play the animation.

Interface

Pick Operand rollout

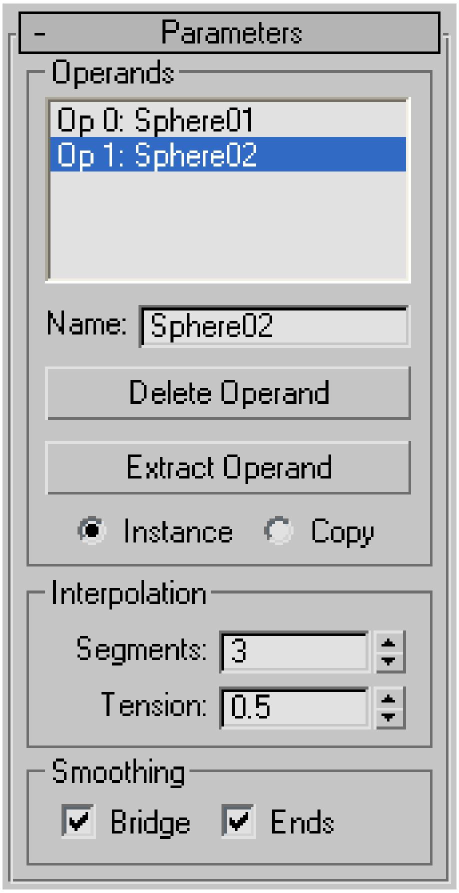
Pick Operand Click this button to connect an additional operand to the original object.

For example, you might begin with a single object with two holes, and arrange two additional objects, each with one hole, outside of those holes. Click the Pick Operand button and select one of the objects, which is connected, and then click Pick Operand again and select the other object, which is connected. Both connected objects are added to the Operands list.

Reference/Copy/Move/Instance Lets you specify how the operand is transferred to the compound object. It can be transferred either as a reference on page 8106, a copy, an instance on page 8014, or moved, in which case the original is not left behind.

NOTE Connect works only with objects that are capable of being converted into editable surfaces, such as editable meshes on page 2075.

Parameters rollout



Operands group

Operands list Displays the current operands. Select an operand to rename, delete or extract by clicking it in this list.

Name Renames a selected operand. Type in a new name, and then press Tab or Enter.

Delete Operand Deletes a selected operand from the list.

Extract Operand Extracts a copy or an instance of the selected operand. Choose an operand in the list to enable this button.

NOTE This button is available only in the Modify panel. You can't extract an operand while in the Create panel.

Instance/Copy Lets you specify how the operand is extracted: as either an instance on page 8014 or a copy.

Interpolation group

Segments Sets the number of segments in the connecting bridge.

Tension Controls the curvature in the connecting bridge. A value of 0 provides no curvature, while higher values create curves that attempt to more smoothly match the surface normals on either end of the connecting bridge. This spinner has no apparent effect when Segments is set to 0.

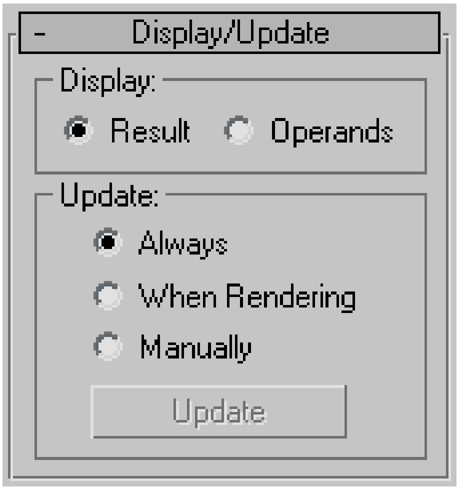
Smoothing group

Bridge Applies smoothing between the faces in the connecting bridge.

Ends Applies smoothing between the faces that border the old and new surfaces of the connecting bridge and the original objects. When turned off, 3ds Max assigns a new material ID number to the bridge. The new number is one higher than the highest ID number assigned to either of the original objects. When on, the ID number is taken from one of the original objects.

NOTE If both Bridge and Ends are on, but the original objects contain no smoothing groups, then smoothing is assigned to the bridge and to the faces bordering the bridge.

Display/Update rollout



Display group

Determines whether the shape operands are displayed.

- **Result** Displays the result of the operation.
- **Operands** Displays the operands.

Update group

These options determine when the projection for the compound object is recalculated. Because complex compound objects can slow performance, you can use these options to avoid constant calculation.

- **Always** The object is updated constantly.
- **When Rendering** The object is recalculated only when the scene is rendered.
- **Manually** Activates the Update button for manual recalculation.

Update Recalculates the projection.

BlobMesh Compound Object

Create panel > Geometry > Compound Objects > Object Type rollout > BlobMesh

Create menu > Compound > BlobMesh

The BlobMesh compound object creates a set of spheres from geometry or particles, and connects the spheres together as if they were made of a soft, liquid substance. When the spheres move within a certain distance of one another, they connect together. When they move apart, they take on a spherical form again.

In the 3D industry, the general term for spheres that operate in this way is metaballs on page 8043. The BlobMesh compound object generates metaballs based on specified objects in the scene, and the metaballs, in turn, form a mesh result called a *blobmesh*. A blobmesh is ideal for simulating thick liquids and soft substances that move and flow when animated.

When you associate an object or particle system with the BlobMesh compound object, the metaballs are placed and sized differently depending on the object used to generate them:

- For geometry and shapes, a metaball is placed at each vertex, and the size of each metaball is determined by the size of the original BlobMesh object. Soft selection can be used to vary the sizes of the metaballs.
- For particles, a metaball is placed at each particle, and the size of each metaball is determined by the size of the particle on which it's based.
- For helpers, a metaball is placed at the pivot point, and the size of the metaball is determined by the original BlobMesh object.

NOTE You can apply motion blur on page 8050 to a BlobMesh object to enhance the effects of motion in renderings. For particle systems other than Particle Flow, use Image motion blur. For Particle Flow particle systems and all other types of objects including geometry, shapes, and helpers, use Object motion blur.

Procedures

To create a blobmesh from geometry or helpers:

1. Create one or more geometry or helper objects. If the scene requires animation, animate the objects as desired.
2. Click BlobMesh, and click anywhere on the screen to create the initial metaball.
3. Go to the Modify panel.
4. In the Blob Objects group, click Add. Select the objects you wish to use to create metaballs. A metaball appears at each vertex of each selected object, or at the centers of helper objects.
5. In the Parameters rollout, set the Size parameter as necessary to cause the metaballs to connect.

To create a blobmesh with soft selection on geometry:

1. Create a geometry object, and convert it to an Editable Mesh or Editable Poly.
2. Apply a Mesh Select modifier to the object, and select some of the vertices on the object.
3. In the Soft Selection rollout, turn on Use Soft Selection. Set the Falloff value as desired.
4. Apply a Turn to Mesh or Turn to Poly modifier to the object.

   This will retain the soft selection and pass it up the stack regardless of whether you exit the sub-object mode.
5. Click Create panel > Compound Objects > BlobMesh, and click anywhere on the screen to create the initial metaball.
6. Go to the Modify panel.
7. In the Blob Objects group, click Add. Select the Editable Mesh or Editable Poly object.

   A metaball appears at each vertex of the selected object.

8 In the Parameters rollout, turn on Use Soft Selection.

   Metaballs are limited to those vertices that are affected by the soft selection.

9 Set the Size and Min. Size parameters to set the sizes of the metaballs.

To create a blobmesh with soft selection on a spline:

1 Create the spline, and convert it to an Editable Spline.

2 In the Rendering rollout, turn on both Renderable and Display Render Mesh.

3 Apply a Mesh Select modifier, and select the appropriate vertices for soft selection.

4 In the Soft Selection rollout, turn on Use Soft Selection. Set the Falloff value as desired.

5 Apply a Turn to Mesh or Turn to Poly modifier to the object.

   This will retain the soft selection and pass it up the stack regardless of whether you exit the sub-object mode.

6 Click Create panel > Compound Objects > BlobMesh, and click anywhere on the screen to create the initial metaball.

7 Go to the Modify panel.

8 In the Blob Objects group, click Add. Select the Editable Spline.

   A metaball appears at each vertex of the selected object.

9 In the Parameters rollout, turn on Use Soft Selection.

   Metaballs are limited to those vertices that are affected by the soft selection.

10 Set the Size and Min. Size parameters to set the sizes of the metaballs.

To create a blobmesh from a particle system:

When you use BlobMesh with a particle system, a metaball is created at each particle's location. The size of the metaball is determined by the size of the particle.

1 Create a particle system on page 2795, and set up its parameters to animate the particles.

2 Click Create panel > Compound Objects > BlobMesh, and click anywhere on the screen to create the initial metaball.

3 Go to the Modify panel.

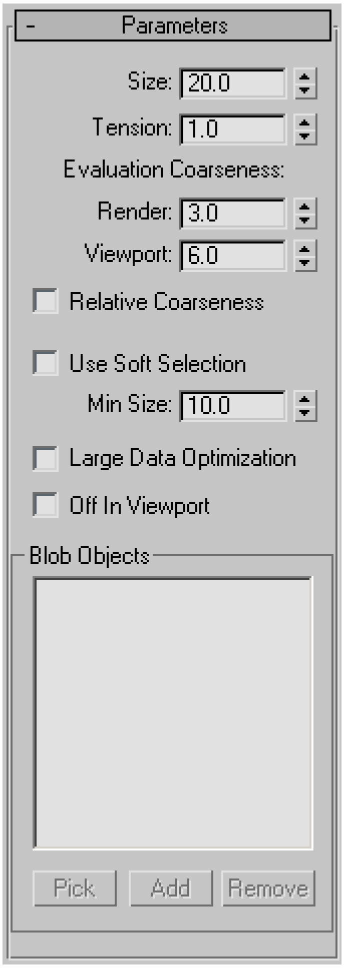
4 In the Blob Objects group, click Add. Select the particle system. A metaball appears at each particle in the system.

5 If you have added a Particle Flow system on page 2795 to the blobmesh and you want to create metaballs only for particles in specific events, click Add on the Particle Flow Parameters rollout to choose the events from a list.

TIP If you need to prevent the particles from rendering, do not hide them as this can prevent the blobmesh from generating correctly. Instead, turn off the particle system's Renderable option on the Object Properties dialog on page 305.

Interface

Parameters rollout



Size The radius of each metaball for objects other than particles. For particles, the size of each metaball is determined by the size of the particle, which is set by parameters in the particle system. Default=20.

NOTE The apparent size of the metaballs is affected by the Tension value. When Tension is set to its lowest possible value, the radius of each metaball accurately reflects the Size setting. Higher Tension values will tighten the surface, and make the metaballs smaller.

Tension Determines how relaxed or tight the surface will be. A smaller value makes a looser surface. This value can range from 0.01 to 1.0. Default=1.0.

Evaluation Coarseness Sets the coarseness, or density, of the resulting blobmesh. When Relative Coarseness (see following) is off, the Render and Viewport values set the absolute height and width of blobmesh faces, and lower values create a smoother, denser mesh. When Relative Coarseness is on, the height and width of blobmesh faces is determined by the ratio of metaball size to this value. In this case, higher values create a denser mesh. Range (both)=0.001 to 1000.0. Render default=3.0, Viewport default =6.0.

The lower end of the range for both Coarseness settings is 0.001, which allows for high-resolution metaball geometry when Relative Coarseness is off. Using such low values can also cause lengthy calculation delays; if this happens and you wish to halt calculation, press Esc.

Relative Coarseness Determines how the coarseness values will be used. If this option is turned off, the Render Coarseness and View Coarseness values are absolute, where the height and width of each face on the blobmesh is always equal to the coarseness value. This means the faces on the blobmesh will retain a fixed size even if the metaballs change size. If this option is turned on, the size of each blobmesh face is based on the ratio of the metaball size to the coarseness, which will cause the blobmesh face size to change as the metaballs become larger or smaller. Default=Off.

Large Data Optimization This option provides an alternate method for calculating and displaying the blobmesh. This method is more efficient than the default method only when a large number of metaballs are present, such as 2,000 or more. Turn on this option only when using a particle system or other object that produces a large number of metaballs. Default=Off.

Off in Viewport Turns off the display of the blobmesh in viewports. The blobmesh will still appearing in renderings. Default=Off.

Use Soft Selection If soft selection has been used on geometry you add to the blobmesh, turning on this option causes the soft selection to be used for the size and placement of metaballs. Metaballs are placed at selected vertices with the size set by the Size parameter. For vertices that lie within the falloff set on the geometry's Soft Selection rollout, smaller metaballs are placed. For vertices outside the falloff, no metaballs are placed. This option has an effect only if the Vertex sub-object level for the geometry is still enabled, and Use Soft

Selection on the geometry's Soft Selection rollout is turned on. If Use Soft Selection is turned off either for the blobmesh for the geometry, metaballs are placed at all vertices on the geometry. Default=Off.

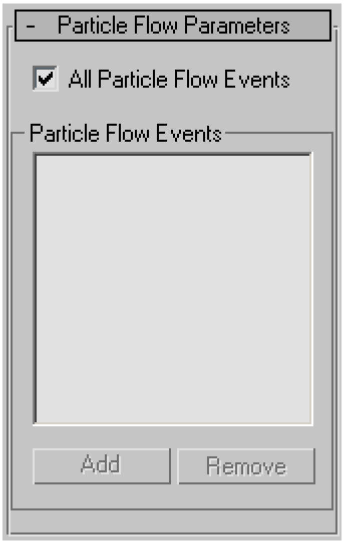
Min Size Sets the minimum size for metaballs within the falloff when Use Soft Selection is turned on. Default=10.0.

Pick Allows you to pick objects or particle systems from the screen to add to the blobmesh.

Add Displays a selection dialog where you can select objects or particle systems to add to the blobmesh.

Remove Removes objects or particles from the blobmesh.

Particle Flow Parameters rollout



Use this rollout if you have added a Particle Flow system to the blobmesh, and want particles to generate metaballs only during specific events. Before you can specify events on this rollout, you must add the Particle Flow system to the blobmesh on the Parameters rollout.

All Particle Flow Events When turned on, all Particle Flow Events will generate metaballs. When turned off, only Particle Flow Events specified in the PFlow Events list will generate metaballs.

Add Displays a list of PFlow events in the scene so you can pick events to add to the PFlow Events list.

Remove Removes the selected event from the PFlow Events list.

ShapeMerge Compound Object

Select an object. > Create panel > Geometry > Compound Objects > Object Type rollout > ShapeMerge

Select an object. > Create menu > Compound > ShapeMerge

ShapeMerge combines the lettering, a text shape, with the mesh that models the tire.

ShapeMerge creates a compound object consisting of a mesh object and one or more shapes. The shapes are either embedded in the mesh, altering the edge and face patterns, or subtracted from the mesh.

Procedures

To create a ShapeMerge object:

1 Create a mesh object and one or more shapes

2 Align the shapes in the viewport so they can be projected toward the surface of the mesh object.

3 Select the mesh object, and click the ShapeMerge button.

4 Click Pick Shape, and then select the shape.

The geometry of the surface of the mesh object is altered to embed a pattern matching that of the selected shape.

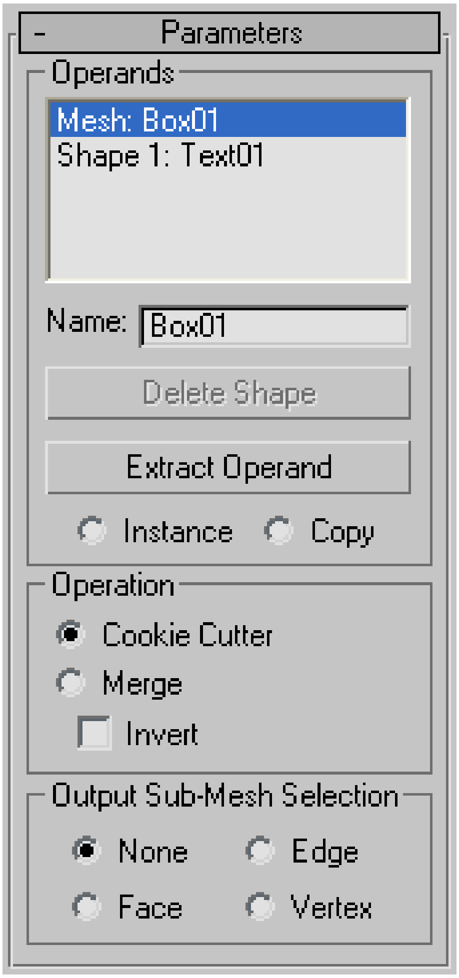
Interface

Pick Operand rollout

Pick Shape Click this button, and then click the shape you want to embed in the mesh object. The shape is projected onto the mesh object in the direction of the shape's local negative Z axis. For example, if you create a box, and then create a shape in the Top viewport, the shape is projected onto the top of the box. You can repeat this process to add shapes, and the shapes can be projected in different directions. Simply click Pick Shape again, and then pick another shape.

Reference/Copy/Move/Instance Lets you specify how the shape is transferred to the compound object. It can be transferred either as a reference on page 8106, a copy, an instance on page 8014, or moved, in which case the original shape is not left behind.

Parameters rollout



Operands group

Operands list Lists all operands in the compound object. The first operand is the mesh object, and any number of shape-based operands can follow.

Delete Shape Remove selected shapes from the compound object.

Extract Operand Extracts a copy or an instance of the selected operand. Choose an operand in the list window to enable this button.

Instance/Copy Lets you specify how the operand is extracted. It can be extracted either as an instance on page 8014 or a copy.

Operation group

These options determine how the shape is applied to the mesh.

Cookie Cutter Cuts the shape out of the mesh object's surface.

Merge Merges the shape with the surface of the mesh object.

Invert Reverses the effect of Cookie Cutter or Merge. With the Cookie Cutter option, the effect is obvious. When Invert is off, the shape is a hole in the mesh object. When Invert is on, the shape is solid and the mesh is missing. When you're using Merge, Invert reverses the sub-object mesh selection. As an example, if you merge a circle shape and apply a Face Extrude, the circular area is extruded when Invert is off, and all but the circular area is extruded when Invert is on.

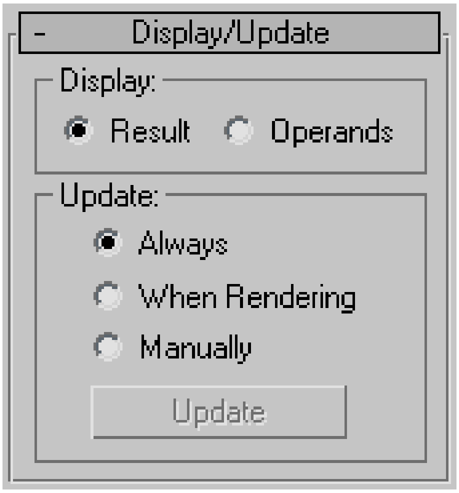
Output Sub-Mesh Selection group

Provides options that let you specify what selection level is passed up the Stack. The ShapeMerge object stores all selection levels; that is, it stores the vertices, faces, and edges of the merged shape with the object. (If you apply a Mesh Select modifier and go to the various sub-object levels, you'll see that the merged shape is selected.) Thus, if you follow the ShapeMerge with a modifier that acts on a specific level, such as Face Extrude, that modifier will work properly.

If you apply a modifier that can work on any selection level, such as Volume Select or XForm, the options will specify which selection level is passed to that modifier. Although you can use a Mesh Select modifier on page 1527 to specify a selection level, the Mesh Select modifier considers the selection only at frame 0. If you've animated the shape operand, that animation will be passed up the Stack for all frames only by using the Output Sub-Mesh Selection options.

- **None** Outputs the full object.
- **Face** Outputs the faces within the merged shape.
- **Edge** Outputs the edge of the merged shape.
- **Vertex** Outputs the vertices defined by the spline of the shape.

Display/Update rollout



Display group

Determines whether the shape operands are displayed.

- **Result** Displays the result of the operation.
- **Operands** Displays the operands.

Update group

These options specify when the display is updated. Typically, you use them when you've animated the merged shape operands and the viewport display is slow.

- **Always** Updates the display at all times.
- **When Rendering** Updates the display only when the scene is rendered.
- **Manually** Updates the display only when you click the Update button.

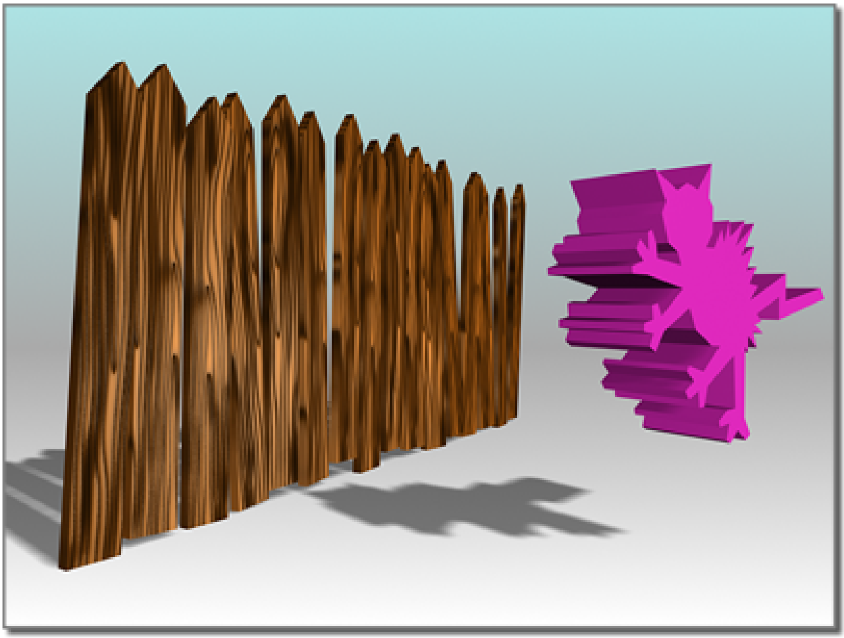
Update Updates the display when any option except Always is chosen.

Boolean Compound Object

Select an object. > Create panel > Geometry > Compound Objects > Object Type rollout > Boolean

Select an object. > Create menu > Compound > Boolean

A Boolean object combines two other objects by performing a Boolean operation on them.



Operand A (left); Operand B (right)

These are the Boolean operations for geometry:

Union The Boolean object contains the volume of both original objects. The intersecting or overlapping portion of the geometry is removed.

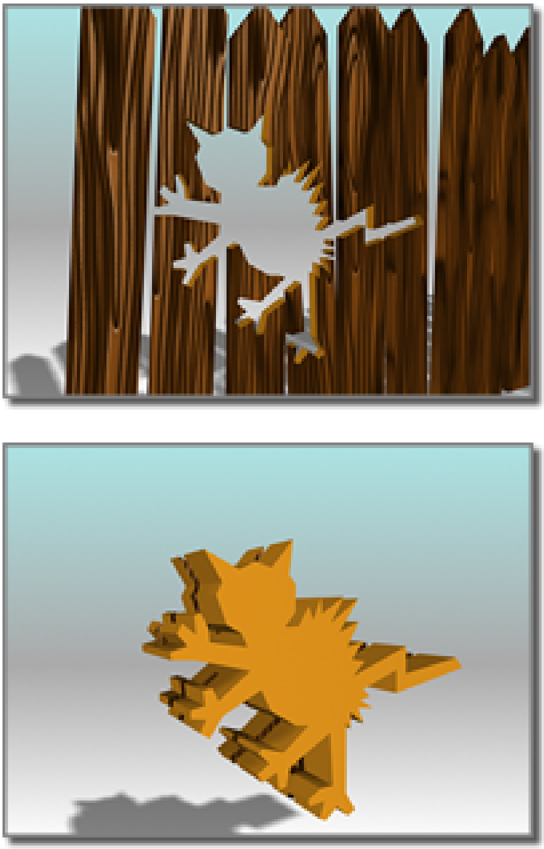
Intersection The Boolean object contains only the volume that was common to both original objects (in other words, where they overlapped).

Subtraction (or difference) The Boolean object contains the volume of one original object with the intersection volume subtracted from it.

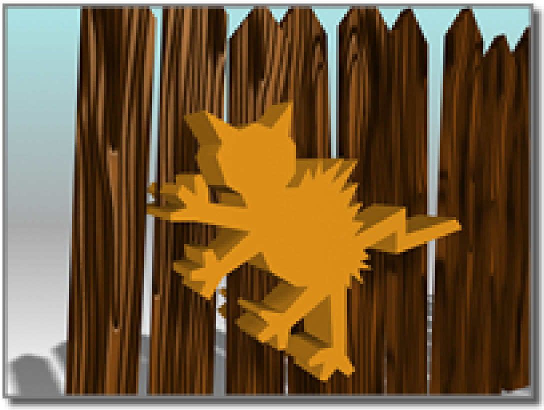
The two original objects are designated as operand A and B.

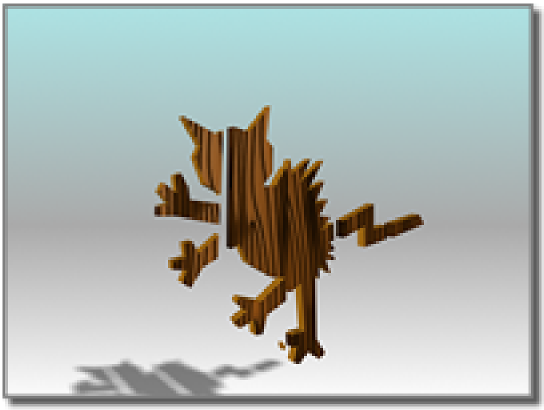
Beginning with version 2.5 of 3ds Max, a new algorithm computes the Boolean operation. This algorithm produces more predictable results and less complex geometry than earlier 3D Studio Booleans. If you open a file that contains a Boolean from an earlier version of 3ds Max, the Modify panel displays the interface for the earlier Boolean operation.

You can layer Booleans in the stack display, so that a single object can incorporate many Booleans. By navigating through the stack display, it's possible to revisit the components of each Boolean and make changes to them.



Subtraction: A-B (above); B-A (below)





Union (above); Intersection (below)

Booleans with Objects That Have Materials Assigned to Them

Most primitives use several material IDs on page 8038 on their surfaces. For example, a box uses material IDs 1–6 on its sides. If you assign a Multi/Sub-Object material on page 5720 with six sub-materials, 3ds Max assigns one to each side. If you assign a multi/sub-object material with two sub-materials, 3ds Max assigns the first material to sides 1, 3, and 5, and the second goes to sides 2, 4, and 6.

When you create a Boolean from objects that have materials assigned to them, 3ds Max combines the materials in the following way:

- If operand A doesn't have a material, it inherits operand B's material.

- If operand B doesn't have a material, it inherits operand A's material.
- If both operands have materials, the new material is a multi/sub-object material that combines the materials from both operands.

For more information, see Material Attach Options Dialog on page 772.

Solutions When Working with Booleans

The Boolean algorithm caused unpredictable behavior in earlier releases. The solutions are discussed here.

Surface Topology

Boolean requires that operands' surface topology be intact: This means no missing or overlapping faces and no unwelded vertices. The surface should be one continuous closed surface.

The Boolean corrects operands that fail to meet this requirement. However, the automatic correction may not be exactly what you want, so in some cases it might be safer to correct the surfaces manually.

To check for holes in the geometry, use the STL-Check modifier on page 1749 or the Measure utility on page 2680.

To fill holes, use the Cap Holes modifier on page 1228.

Face Normals

Booleans require that the face normals of the surface be consistent. Flipped normals can produce unexpected results. Surfaces where some faces are facing one way and adjacent faces are flipped are also problematic, and are commonly found in geometry imported from CAD programs. The Boolean fixes these faces as best it can. Again, it might make more sense to correct these manually.

Use shaded viewports to look for normal problems, watching for objects that appear inside-out or look otherwise incorrect. You can also turn on Show in the Editable Mesh (Face) on page 2099 > Surface Properties rollout > Normals group. Fix normals here, or with a Normal modifier on page 1581.

Overlapping Elements

Because Boolean operations depend on a clear understanding of what is inside and what is outside a mesh, meshes that overlap themselves can produce invalid results. For instance, if you use the Collapse utility on page 2016 with two overlapping objects without turning on the Boolean feature, the resulting object will not make a good Boolean operand. This is also a problem for the Teapot primitive on page 437 (with all parts turned on), which overlaps itself.

If you need to use such an object as a Boolean operand, you might reconstruct it as a single non-overlapping mesh by separating the components and combining them with Boolean.

Working with Inverted Meshes

Boolean doesn't always produce the ideal result on "inverted meshes" (meshes that have been turned inside-out by having their normals flipped). The problem is that the area inside the flipped mesh is correctly seen as "outside," but the area outside it may also be seen as “outside.” To remedy this, instead of inverting the mesh, make a very large box or other primitive centered on (but not touching) the mesh and subtract the mesh from it using Boolean. Then convert it to an editable mesh, and delete the box faces. This produces a correctly inverted mesh that works correctly with Boolean.

Alignment

If two Boolean operands are perfectly aligned without actually intersecting, the Boolean operation might produce the wrong result. Although this is rare, if it does occur, you can eliminate it by making the operands overlap slightly.

Relative Complexity Between Operands

Boolean works best when the two operands are of similar complexity. If you wish to subtract text (a complex object made of many faces and vertices) from a box without any segments, the result is many long, skinny faces that are prone to rendering errors. Increasing the number of box segments produces better results. Try to maintain a similar complexity between operands.

Coplanar Faces/Colinear Edges

Previously, Boolean required that objects overlap. If two objects did not overlap but merely touched an edge to an edge, or a face to a face, the Boolean would fail.

Boolean allows for non-overlapping objects. Coincident faces/edges and vertices are no longer a problem. You can use objects completely encased within another object, where no edges intersect, to create Booleans.

See also Collapse Utility on page 2016 to create Booleans with multiple objects.

See also:

- Fixing Boolean Problems on page 7863

Procedures

To create a Boolean object:

1. Select an object. This object becomes operand A.
2. Click Boolean. The name of operand A appears in the Operands list on the Parameters rollout.
3. On the Pick Boolean rollout, choose the copy method for operand B: Reference, Move, Copy, or Instance. (These methods are described in the Pick Boolean rollout section, later in this topic.)
4. On the Parameters rollout, choose the Boolean operation to perform: Union, Intersection, Subtraction (A-B), or Subtraction (B-A). You can also choose one of the Cut operations, described later in the Operation group section.
5. On the Pick Boolean rollout, click Pick Operand B.
6. Click in a viewport to select operand B. 3ds Max performs the Boolean operation.

   The operand objects remain as sub-objects of the Boolean object. By modifying the creation parameters of the Boolean's operand sub-objects, you can later change operand geometry in order to change or animate the Boolean result.

Example: To create and modify a single object that contains multiple Booleans:

Suppose you want to create a box with two holes in it. One hole is to be cut by a sphere, and the second by a cylinder. If you want to make changes to the sphere or the cylinder later, you can do so by following these steps:

1. Create a Boolean following the steps in the previous sections. The original object (the box) is converted to a Boolean, and is designated operand A. The second object (the sphere) is converted to operand B.
2. Deselect the Boolean object. Build the cylinder if it does not already exist.
3. Select the Boolean object; and under Compound Objects, click Boolean again.

4 Click Pick Operand B and click the cylinder in the viewport. It is converted to operand B.

5 On the Modify panel, choose Operand B from the Parameters rollout > Operands list. If you want to see operand B, choose Display/Update rollout > Display group > Operands or Result + Hidden Ops.

   If you want to animate the Cylinder or the Cylinder's parameters you can now access them in the modifier stack display.

6 If you want to modify the sphere's parameters, choose the box in the Operands list.

7 Now there are two entries labeled Boolean in the stack display. Choose the lower entry. The Sphere is displayed in the Operands list.

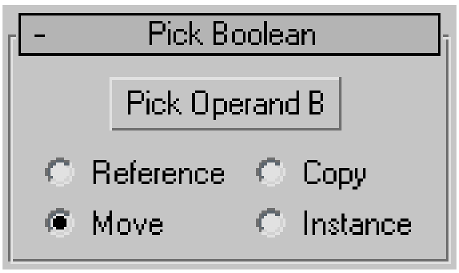
8 Choose the Sphere from the Operands list. The sphere's parameters are available by clicking the sphere's name in the modifier stack display.

9 Use this technique to change parameters or animate any of the operands within the multiple Boolean.

You can also navigate multiple Booleans through Track View. Clicking the operand in Track View gives you direct access to its entry in the modifier stack display. In complex objects with many Booleans, this is an easier method than the one outlined above.

Interface

Pick Boolean rollout



When you select operand B, you designate it as a Reference, Move (the object itself), Copy, or Instance, according to your choice in the Pick Boolean rollout for Boolean objects. Base your selection on how you want to use the scene geometry after you create the Boolean.

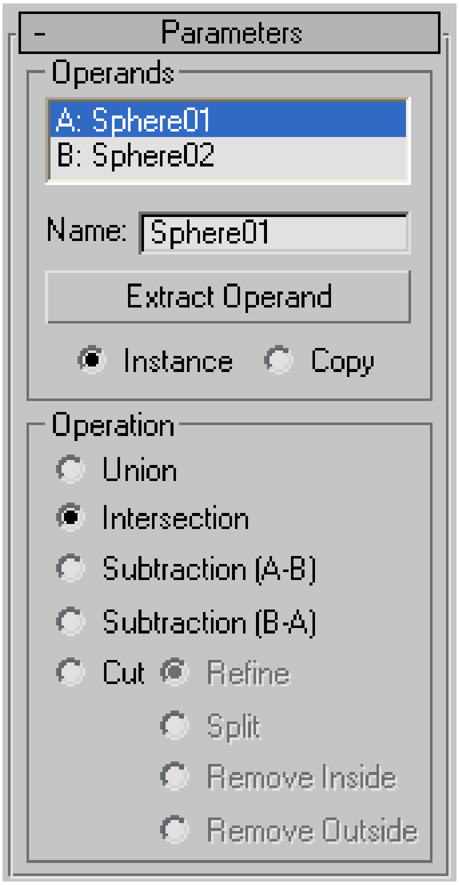
Because you usually create Boolean objects from overlapping objects, if the B object isn't removed (if you don't use the default Move option), it often obstructs your view of the completed Boolean. You can move the Boolean or the B object to better see the result.

Pick Operand B Use this button to select the second object to use to complete the Boolean operation.

Reference/Copy/Move/Instance Lets you specify how operand B is transferred to the Boolean object. It can be transferred either as a reference on page 8106, a copy, an instance on page 8014, or moved.

- Use Reference to synchronize modifier-induced changes to the original object with operand B, but not vice-versa.
- Use Copy when you want to reuse the operand B geometry for other purposes in the scene.
- Use Instance to synchronize animation of the Boolean object with animated changes to the original B object, and vice-versa.
- Use Move (the default) if you've created the operand B geometry only to create a Boolean, and have no other use for it.
  Object B geometry becomes part of the Boolean object regardless of which copy method you use.

Parameters rollout



Operands group

Operands list field Displays the current operands.

Name Edit this field to change the name of the operands. Choose an operand in the Operands list and it will also appear in the Name box.

Extract Operand Extracts a copy or an instance of the selected operand. Choose one of the operands in the list window to enable this button.

NOTE This button is available only in the Modify panel. You can't extract an operand while the Create panel is active.

Instance/Copy Lets you specify how the operand is extracted: as either an instance on page 8014 or a copy.

Operation group

Union The Boolean object contains the volume of both original objects. The intersecting or overlapping portion of the geometry is removed.

Intersection The Boolean object contains only the volume that was common to both original objects (in other words, where they overlapped).

Subtraction (A-B) Subtracts the intersection volume of operand B from operand A. The Boolean object contains the volume of operand A with the intersection volume subtracted from it.

Subtraction (B-A) Subtracts the intersection volume of operand A from operand B. The Boolean object contains the volume of operand B with the intersection volume subtracted from it.

Cut Cuts operand A with operand B, but doesn't add anything to the mesh from operand B. This works like the Slice modifier on page 1729, but instead of using a planar gizmo, Cut uses the shape of operand B as the cutting plane. Cut treats the geometry of the Boolean object as volumes rather than closed solids. Cut does not add geometry from operand B to operand A. Operand B intersections define cut areas for altering geometry in operand A.

There are four types of Cut:

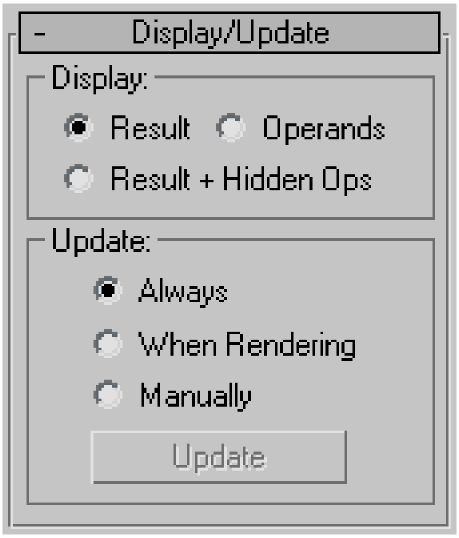
- **Refine** Adds new vertices and edges to operand A where operand B intersects the faces of operand A. 3ds Max refines the resulting geometry of operand A with additional faces inside the intersected area of operand B. Faces cut by the intersection are subdivided into new faces. You might use this option to refine a box with text so that you can assign a separate material ID to the object.

- **Split** Works like Refine but also adds a second or double set of vertices and edges along the boundary where operand B cuts operand A. Split produces two elements belonging to the same mesh. Use Split to break an object into two parts along the bounds of another object.

- **Remove Inside** Deletes all operand A faces inside operand B. This option modifies and deletes faces of operand A inside the area intersected by operand B. It works like the subtraction options, except that 3ds Max adds no faces from operand B. Use Remove Inside to delete specific areas from your geometry.

- **Remove Outside** Deletes all operand A faces outside operand B. This option modifies and deletes faces of operand A outside the area intersected by operand B. It works like the Intersection option, except that 3ds Max adds no faces from operand B. Use Remove to delete specific areas from your geometry.

Display/Update rollout



Display group

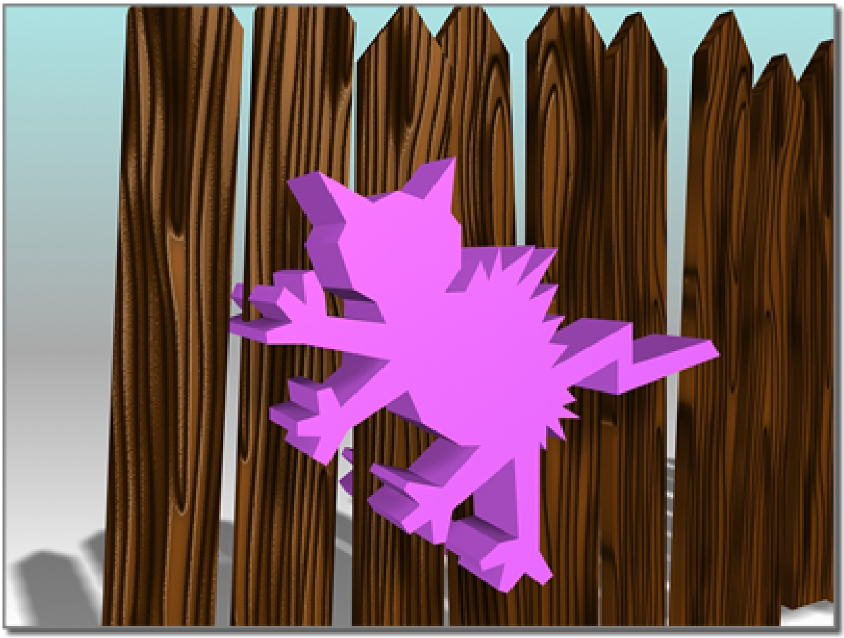
Visualizing the result of a Boolean can be tricky, especially if you want to modify or animate it. The Display options on the Boolean Parameters rollout help you visualize how the Boolean is constructed.

The display controls have no effect until you've created the Boolean.

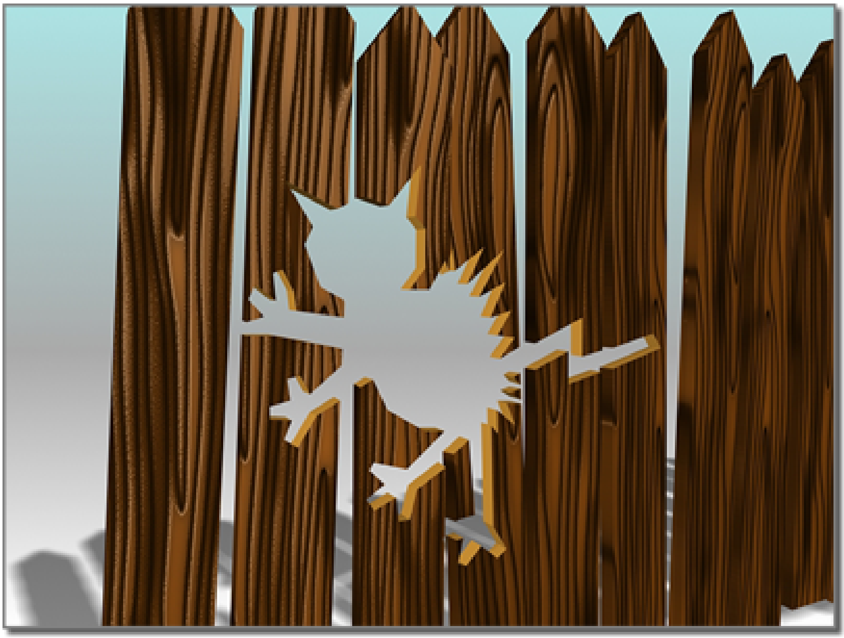
- **Result** Displays the result of the Boolean operation; that is, the Boolean object itself.
- **Operands** Displays the operands instead of the Boolean result.

TIP When operands are difficult to see in a viewport, you can use the Operand list to select one or the other. Click the name of the A or B operand to select it.

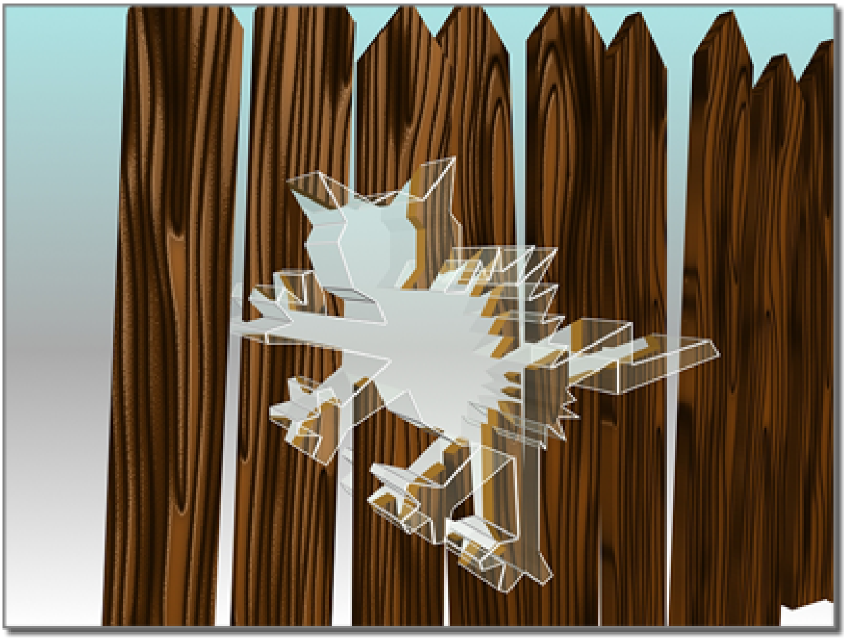
- **Results + Hidden Ops** Displays the "hidden" operands as wireframe. Operand geometry remains part of the compound Boolean object, although it isn't visible or renderable. The operand geometry is displayed as wireframes in all viewports.



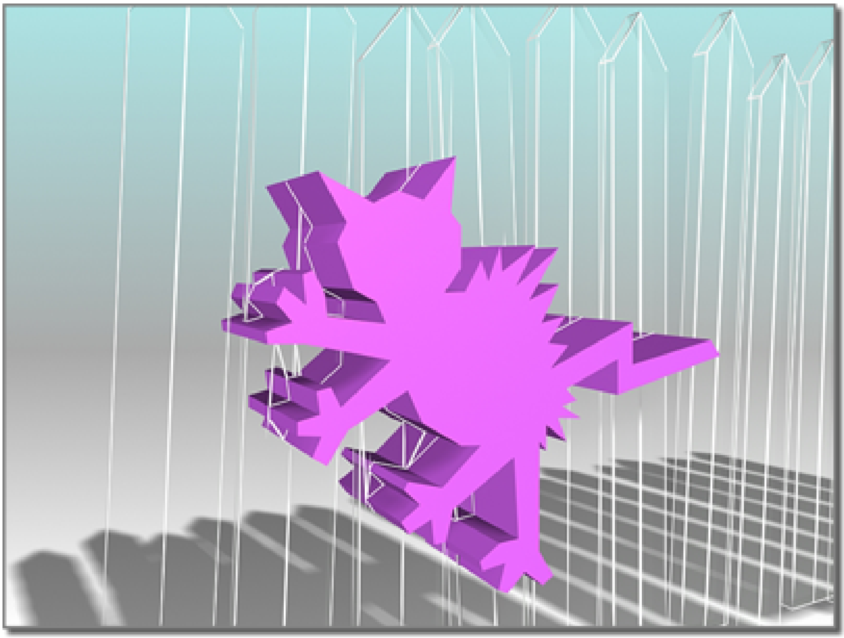
Displaying the operands



Displaying the result (A-B)



Displaying the hidden operand after A-B



Displaying the hidden operand after B-A

Update group

By default, Booleans are updated whenever you change the operands. A scene that contains one or more complicated, animated Booleans can impede performance. The update options provide alternate methods to improve performance.

- **Always** Updates Booleans immediately when you change an operand, including the original object of an instanced or referenced B operand. This is the default behavior.
- **When Rendering** Updates Booleans only when you render the scene or click Update. With this option, viewports don't always show current geometry, but you can force an update when necessary.
- **Manually** Updates Booleans only when you click Update. With this option, the viewports and the render output don’t always show current geometry, but you can force an update when necessary.

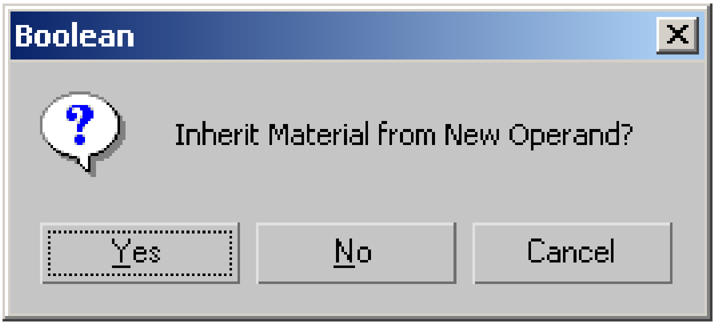
Update Updates the Boolean. The Update button is not available when Always is selected.

Material Attach Options Dialog

Use objects with different materials assigned to them. > Create panel > Geometry > Compound Objects > Object Type rollout > Boolean > Pick Boolean rollout > Pick Operand B button > Select object in the viewport that is operand B.

When you use Boolean operations with objects that have been assigned different materials, 3ds Max displays the Material Attach Options dialog. This dialog offers five methods for handling the materials and the material IDs on page 8038 in the resultant Boolean object.

NOTE If operand A has no material, and operand B has a material assigned, the Boolean dialog lets you choose to inherit the material from operand B.



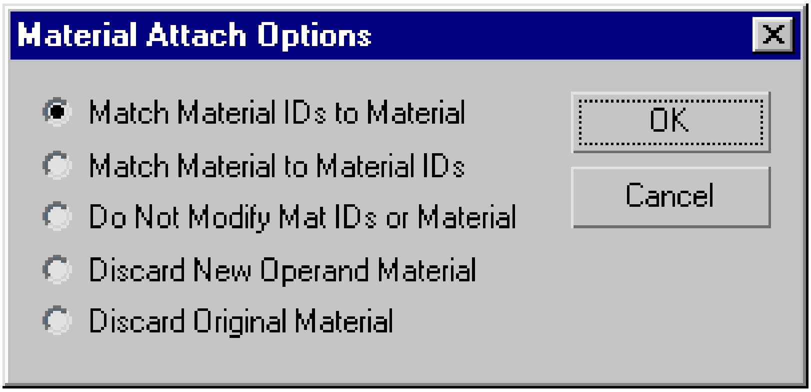
If operand A has a material assigned and operand B has no material assigned, the Boolean object automatically inherits materials from operand A.

Procedures

To create a Boolean from objects that match material IDs to material:

1. Create a Boolean on page ? using at least one object that has a multi/sub-object material on page 5720 assigned to it.
2. On the Pick Boolean rollout, click Pick Operand B.
3. Click in a viewport and select the B operand. 3ds Max displays the Match Attach Options dialog.
4. Choose Match Material IDs to Material to complete the Boolean operation.

Interface



Match Material IDs to Material 3ds Max modifies the number of material IDs in the combined object to be no greater than the number of sub-materials assigned to the operands. For example, if you combine two boxes that have standard materials and each box is assigned six material IDs (the default), the resulting combined object has two operands with one material ID each, rather than the 12 that would result from using the Match Material to Material ID option. After you complete the operation, 3ds Max creates a new multi/sub-object material with two slots. 3ds Max assigns the sub-materials to the operands as they appeared before the operation. The number of resulting material IDs matches the number of materials between the original objects. You might use this option to reduce the number of material IDs.

Match Material to Material IDs Maintains the original material ID assignment in the operands by adjusting the number of sub-materials in the resultant multi/sub-object material. For example, if you combine two boxes, both assigned single materials, but with their default assignment of six material IDs, the result would be a multi/sub-object material with 12 slots (six containing instances of one box's material, and six containing instances of the other box's material). Use this option when it's important to maintain the original material ID assignments in your geometry. Also use this option when material IDs have been assigned, but materials have not been assigned.

NOTE To make the instanced sub-materials unique, select them in Track View, and click the Make Unique button on the Track View toolbar. You can also make them unique one at a time with the Make Unique button on page 5346 in the Material Editor.

Do Not Modify Mat IDs or Material If the number of material IDs in an object is greater than the number of sub-materials in its multi/sub-object material, then the resultant face-material assignment might be different after the Boolean operation.

Discard New Operand Material Discards the material assignment of operand B. 3ds Max assigns operand A's material to the Boolean object.

Discard Original Material Discards the material assignment of operand A. 3ds Max assigns operand B's material to the Boolean object.

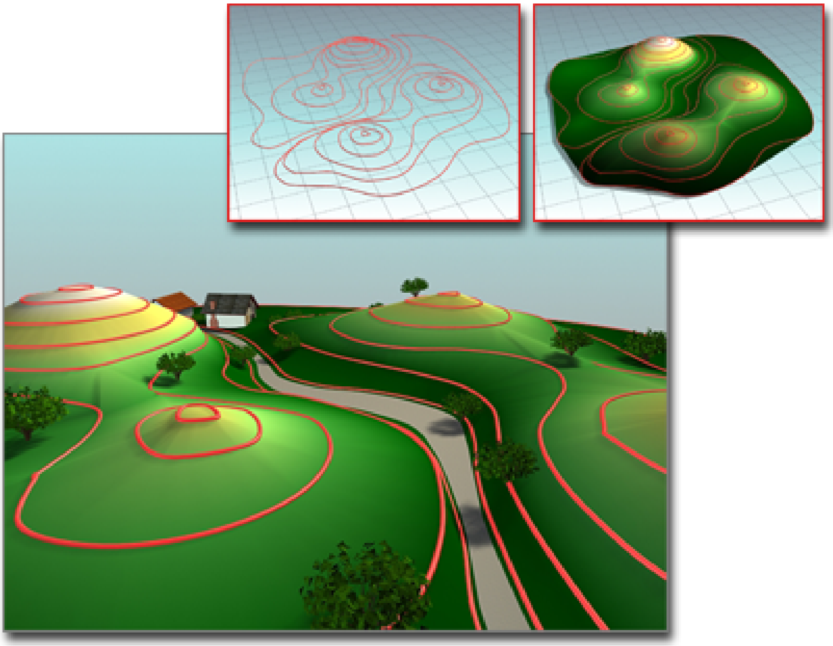
NOTE A UVW Map modifier on page 1931 must be used with compound objects to apply mapping coordinates.

Terrain Compound Object

Select spline contours. > Create panel > Geometry > Compound Objects > Object Type rollout > Terrain

Select spline contours. > Create menu > Compound > Terrain

The Terrain button lets you produce terrain objects. 3ds Max generates these objects from contour line data. You select editable splines representing elevation contours and create a mesh surface over the contours. You can also create a "terraced" representation of the terrain object so that each level of contour data is a step, resembling traditional study models of land forms.



Using contours to build a terrain

Upper left: The contours

Upper right: The terrain object

Lower left: Terrain object used as the basis of a landscape

If you import an AutoCAD drawing file to use as contour data, 3ds Max names each object based on the AutoCAD object's layer, color, or object type. A number is appended to each name. For example, an AutoCAD object on the layer *BASE* becomes an object named *BASE.01*. See Importing DWG Files on page 7182 for more information.

After you import or create the contour data, select the objects, and click the Terrain button, 3ds Max creates a new triangulated mesh object based on the contour data. The name of the first selected spline becomes the name of the terrain object. Other splines in the selection are treated according to the previously set Reference, Move, Copy, or Instance selection in the Pick Operand rollout, described below.

Keep in mind that the Terrain object can use any spline objects as operands, whether they are horizontal splines or not. Though the most common scenario is when sets of elevational contours are used to create terrain forms, it is possible to append or refine Terrain objects by using non-horizontal splines.

NOTE To ensure that 3ds Max imports polylines as splines, when you import an AutoCAD drawing file, turn off Import AutoCAD DWG File dialog > Geometry Options group > Cap Closed Entities.

Following are examples of uses of the Terrain feature:

- Visualizing the effects of grading plans in 3D.
- Maximizing views or sunlight by studying topographical undulation of land forms.
- Analyzing elevation changes by using color on the data.
- Adding buildings, landscaping, and roads to a terrain model to create virtual cities or communities.
- Viewing corridors and completing ridge analyses from particular locations on a site by adding cameras to the scene.

Procedures

To analyze elevation changes:

1 Import or create contour data.

2 Select the contour data, and click the Terrain button.

3 On the Color By Elevation rollout, enter elevation zone values between the maximum and minimum elevations in the Base Elev box. Click Add Zone after entering the value.

   3ds Max displays the zones in the list under the Create Defaults button.

4 Click the Base Color swatch to change the color of each elevation zone. For example, you could use a deep blue for low elevations, a light blue for intermediate elevations, and perhaps greens for higher elevations.

5 Click Solid To Top of Zone to see the elevation changes in a striped effect.

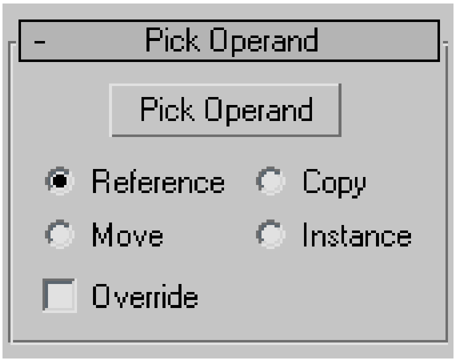
6 Click Blend To Color Above to see the elevation changes blended.

Interface

Name and Color rollout

Displays the name of the terrain object. 3ds Max uses the name of one of the selected objects to name the terrain object.

Pick Operand rollout

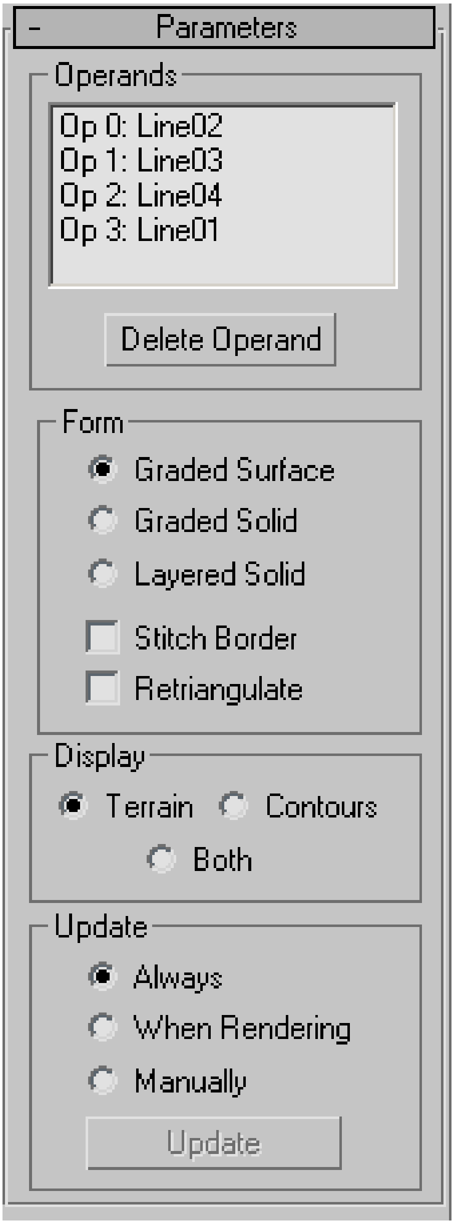


Pick Operand Adds splines to the terrain object. You might do this if you didn't select all the objects before generating the terrain object, or if some objects in the imported data weren't included in the terrain object. You can also use this option to add existing splines in the current scene to the terrain object.

Reference/Copy/Move/Instance When you click Pick Operand, the copy method you designate determines how the operands are used. When Move is the method, the original contour data is moved from the scene and into the operands of the new terrain object. Copy, Reference, and Instance retain the original contour data in the scene and create copies, references or instances of the contour data as operands in the terrain object. This is similar to the copy method for Boolean on page 757.

Override Allows you to select closed curves that override any other operand data within their interior. Within the area an Override operand encloses (as seen in plan), other curves and points of the mesh are disregarded and the elevation of the Override operand supersedes them. An Override operand is indicated in the operands list by a # after its name. Override is only effective on closed curves. If multiple override operands overlap, later overrides (higher operand numbers) take preference.

Parameters rollout



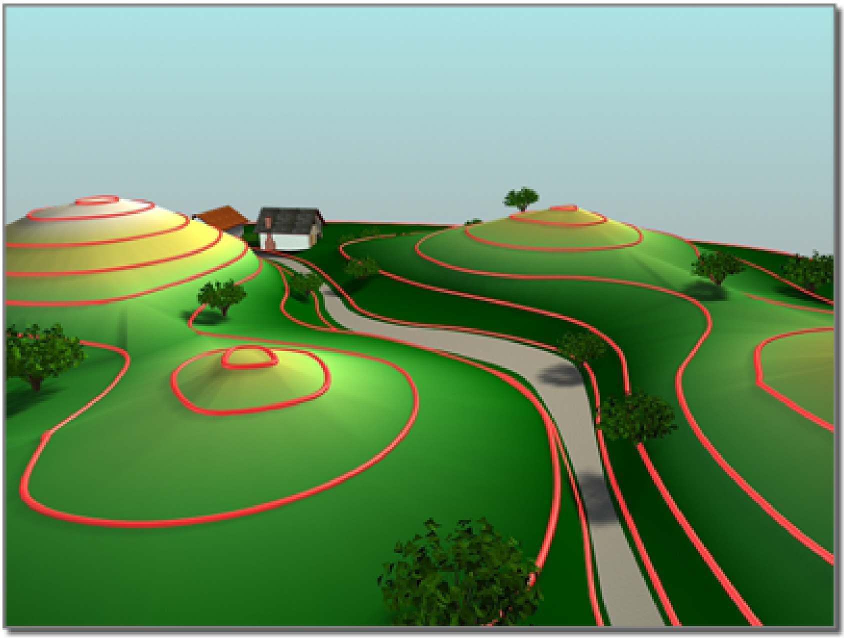
Operands group

Operand list Displays the current operands. Each operand is listed as "Op" followed by a number and the name of the object that is being used as the operand. The operand name comprises layer, color, or object type name plus a numeric suffix.

Delete Operand Deletes a selected operand from the Operands list.

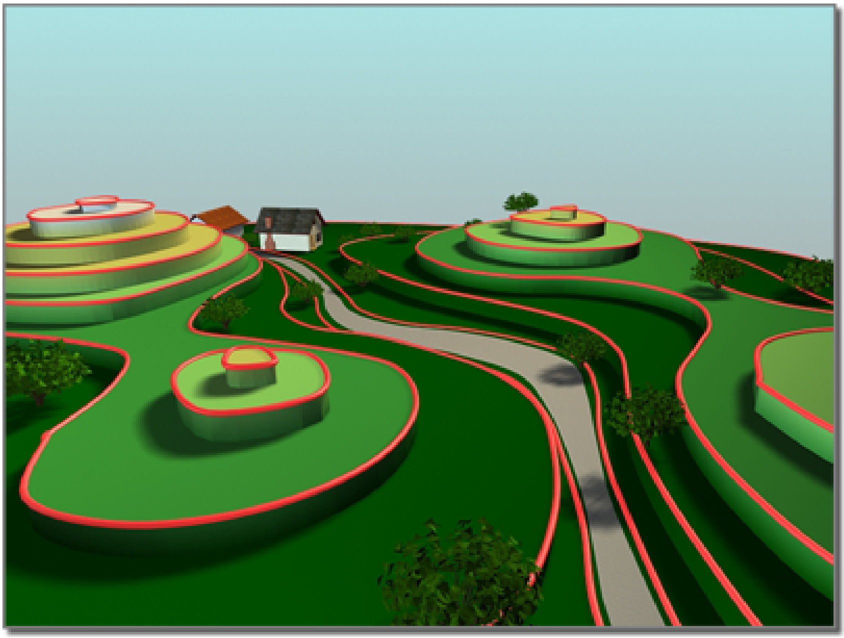
Form group

- **Graded Surface** Creates a graded surface of the mesh over the contours.



Terrain created as a graded surface

- **Graded Solid** Creates a graded surface with skirts around the sides and a bottom surface. This represents a solid that is visible from every direction.
- **Layered Solid** Creates a "wedding cake" or laminated solid similar to cardboard architectural models.



Terrain created as a "layered solid" surface, with levels

Stitch Border When on, suppresses the creation of new triangles around the edges of terrain objects when edge conditions are defined by splines that are not closed. Most terrain forms display more reasonably when this is turned off.

Retriangulate The basic Terrain algorithm tends to flatten or notch contours when they turn sharply upon themselves. A typical situation in which this may happen is when a narrow creek bed is described with contours; the resulting form may look more like a series of cascades at each elevational contour, rather than a smoothly descending ravine. When Retriangulate is checked, a somewhat slower algorithm is used that follows contour lines more closely. This may be particularly evident in the Layered Solid display mode. For additional precision, try using Retriangulate in conjunction with horizontal interpolation.

Display group

- **Terrain** Displays only the triangulated mesh over the contour line data.

- **Contours** Displays only the contour line data of the terrain object.
- **Both** Displays both the triangulated mesh and the contour line data of the terrain object. You can select the terrain object by clicking its surface, but not by clicking a contour line. When Both is selected, contour lines may not be apparent in Wireframe display modes or when Edged Faces are displayed.

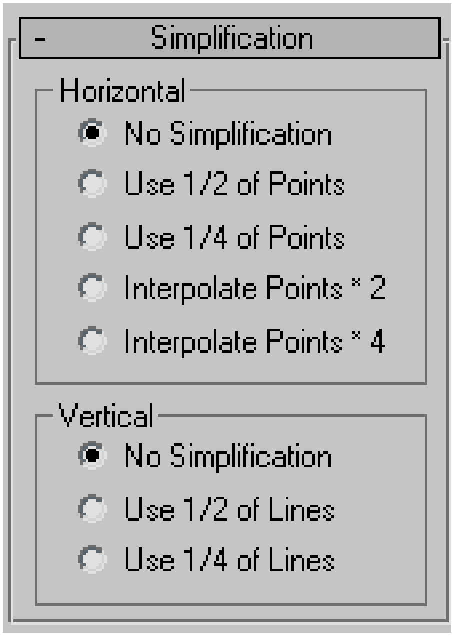
Update group

The items in this group box determine when 3ds Max recalculates the projection for the terrain object. Because complex terrain objects can slow performance, you can use these options to avoid constant calculation.

- **Always** Updates the terrain object immediately when you change an operand, including the original object of an instanced or referenced operand.
- **When Rendering** Updates the terrain object when you render the scene or when you click Update. With this option, viewports won't show current geometry unless you click Update.
- **Manually** Updates the terrain object when you click Update.

Update Updates the terrain object. This button is not enabled only when Always is the active option.

Simplification rollout



Horizontal group

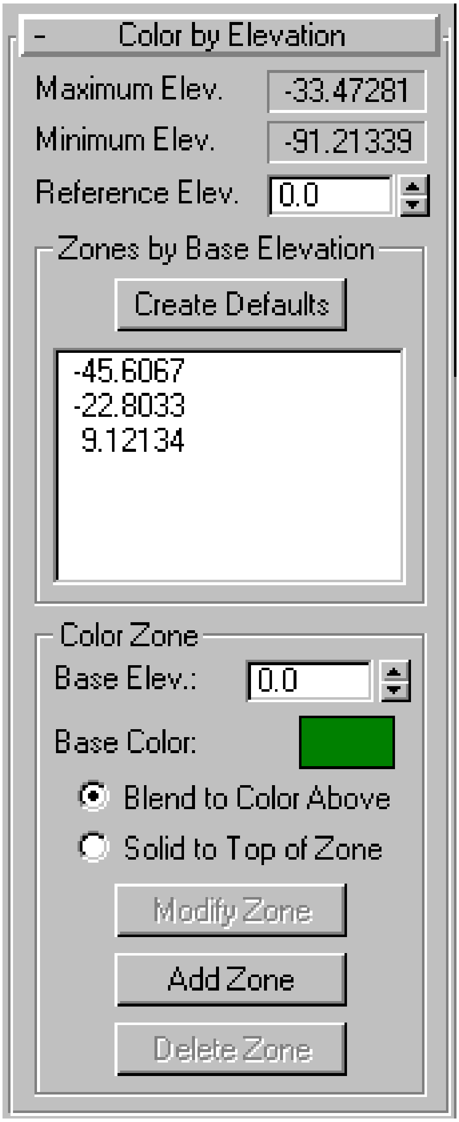
- **No Simplification** Uses all the operands' vertices to create a complex mesh. This results in greater detail and a larger file size than the two fractional options.
- **Use 1/2 of Points** Uses half the set of vertices in the operands to create a less complex mesh. This results in less detail and a smaller file size than using No Simplification.
- **Use 1/4 of Points** Uses a quarter of the of vertices in the operands to create a less complex mesh. This results in the least detail and smallest file size of these options.
- **Interpolate Points * 2** Doubles the set of vertices in the operands to create a more refined but more complex mesh. This is most effective in terrain forms that use constructive curves such as circles and ellipses. This results in more detail and a larger file size than using No Simplification.
- **Interpolate Points * 4** Quadruples the set of vertices in the operands to create a more refined but more complex mesh. This is most effective in

terrain forms that use constructive curves such as circles and ellipses. This results in more detail and a larger file size than using No Simplification.

Vertical group

- **No Simplification** Uses all the spline operandsvertices of the terrain object to create a complex mesh. This results in greater detail and a larger file size than the other two options.
- **Use 1/2 of Lines** Uses half the set of spline operands of the terrain object to create a less complex mesh. This results in less detail and a smaller file size than using No Simplification.
- **Use 1/4 of Lines** Uses a quarter of the of spline operands of the terrain object to create a less complex mesh. This results in the least detail and smallest file size of the three options.

Color by Elevation rollout



Maximum Elev. Displays the maximum elevation in the Z axis of the terrain object. 3ds Max derives this data from the contour data.

Minimum Elev. Displays the minimum elevation in the Z axis of the terrain object. 3ds Max derives this data from the contour data.

Reference Elev. This is the reference elevation, or datum, that 3ds Max uses as a guide for assigning colors to zones of elevation. After entering a reference elevation, click the Create Defaults button. 3ds Max treats elevations above the reference elevation as solid land and those below the reference elevation as water.

If you enter a value no greater than the minimum elevation in the object, 3ds Max divides the range between the reference and minimum elevations into five color zones: dark green, light green, yellow, purple, and light gray.

If you enter a value between the minimum and maximum elevations, 3ds Max creates six color zones. Two zones (dark blue and light blue) are used for elevations below the reference elevation. These are considered to be under water. One zone (dark yellow) is used for a narrow range around the reference elevation. Three zones (dark green, light green, light yellow) are used for elevations above the reference elevation.

If you enter a value at or above the maximum elevation, 3ds Max divides the range between the minimum and reference elevations into three zones (dark blue, medium blue, light blue).

Zones by Base Elevation group

Create Defaults Creates elevation zones. 3ds Max lists the elevation at the bottom of each zone, referenced to the datum (the reference elevation). 3ds Max applies the color of the zone at the base elevation. Whether the colors blend between zones depends on your choice of the Blend to Color Above or Solid to Top of Zone option.

Color Zone group

The items in this group box assign colors to elevation zones. For example, you might want to change levels of blue to indicate the depth for water. Your changes in the Color Zone area don't affect the terrain object until you click the Modify Zone or Add Zone button.

Base Elev This is the base elevation of a zone to which you assign color. After entering a value, click Add Zone to display the elevation in the list under Create Defaults.

Base Color Click the color swatch to change the color of the zone.

- **Blend to Color Above** Blends the color of the current zone to the color of the zone above it.

- **Solid to Top of Zone** Makes a solid color at the top of the zone without blending to the color of the zone above it.

Modify Zone Modifies selected options of a zone.

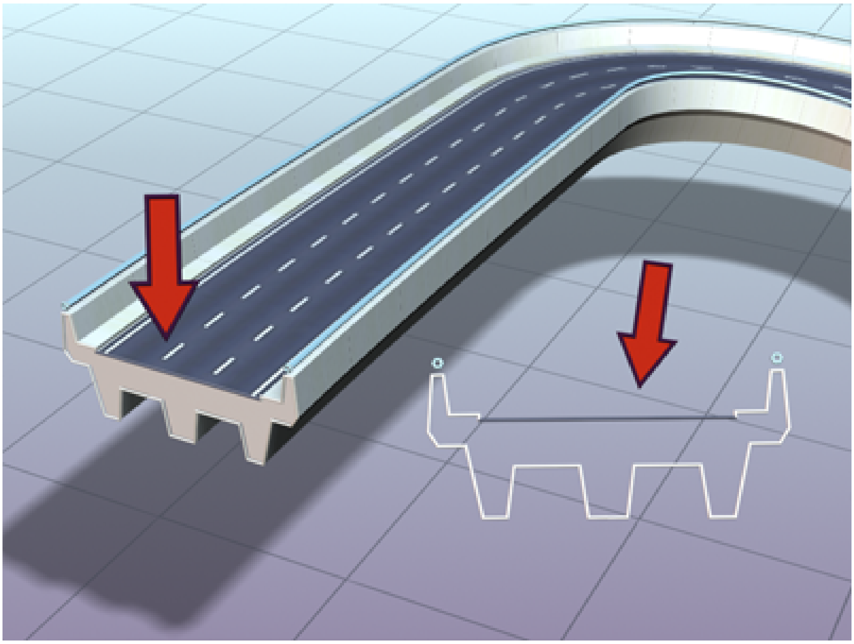
Add Zone Adds values and selected options for a new zone.

Delete Zone Deletes a selected zone.

Loft Compound Object

Select a path or shape. > Create panel > Geometry > Compound Objects > Object Type rollout > Loft

Select a path or shape. > Create menu > Compound Objects > Loft



Roadway created as a lofted shape

Loft objects are two-dimensional shapes extruded along a third axis. You create loft objects from two or more existing spline objects. One of these splines serves the path. The remaining splines serve as cross-sections, or shapes, of the loft object. As you arrange shapes along the path, 3ds Max generates a surface between the shapes.

You create shape objects to serve as a path for any number of cross-section shapes. The path becomes the framework that holds the cross sections forming your object. If you designate only one shape on the path, 3ds Max assumes an identical shape is located at each end of the path. The surface is then generated between the shapes.

3ds Max places few restrictions on how you create a loft object. You can create curved, three-dimensional paths and even three-dimensional cross sections.

When using Get Shape, as you move the cursor over an invalid shape, the reason the shape is invalid is displayed in the prompt line.

Unlike other compound objects, which are created from the selected object as soon as you click the compound-object button, a Loft object is not created until you click Get Shape or Get Path, and then select a shape or path.

Loft is enabled when the scene has one or more shapes. To create a loft object, first create one or more shapes and then click Loft. Click either Get Shape or Get Path and select a shape in the viewports.

Once you create a loft object, you can add and replace cross-section shapes or replace the path. You can also change or animate the parameters of the path and shapes.

You can't animate the path location of a shape.

You can convert loft objects to NURBS surfaces on page 2304.

Procedures

To create a loft object:

Creating loft objects is detailed and offers many choices, but the basic process is quite simple.

1. Create a shape to be the loft path.
2. Create one or more shapes to be loft cross sections.
3. Do one of the following:
   - Select the path shape and use Get Shape to add the cross sections to the loft.
   - Select a shape and use Get Path to assign a path to the loft. Use Get Shape to add additional shapes.

You can use the loft display settings to view the skin generated by your loft in both wireframe and shaded views.

To create a loft with Get Path:

1 Select a shape as the first cross-section shape.

2 Click Create panel > Geometry > Compound Objects > Loft.

3 On the Creation Method rollout, click Get Path.

4 Choose Move, Copy, or Instance.

5 Click a shape for the path.

The cursor changes to the Get Path cursor as you move it over valid path shapes. If the cursor does not change over a shape, that shape is not a valid path shape and cannot be selected. The first vertex of the selected path is placed at the first shape's pivot and the path tangent is aligned with the shape's local Z axis.

To create a loft with Get Shape:

1 Select a valid path shape as the path.

2 If the selected shape is not a valid path, the Get Shape button is unavailable.

3 Click Create panel > Geometry > Compound Objects > Loft.

4 On the Creation Method rollout, click Get Shape.

5 Choose Move, Copy, or Instance.

6 Click a shape.

The cursor changes to the Get Shape cursor as you move it over potential shapes. The selected shape is placed at the first vertex of the path.

TIP You can flip the shape along the path by holding down Ctrl when using Get Shape. For example, if you select the lowercase letter "b" with a Ctrl+click, the loft will look like the letter "d".

Interface

You use the following rollouts for setting loft object parameters:

Creation Method Rollout on page 789

Surface Parameters Rollout on page 790

Path Parameters Rollout on page 794

Skin Parameters Rollout on page 797

Once you've created a loft object, you can also use the Modify panel's Deformations rollout to add complexity. See Deformations on page 814 for further information.

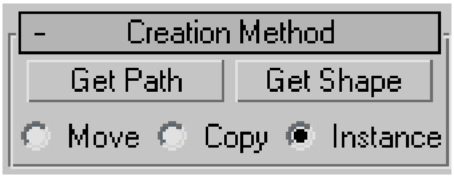
Creation Method Rollout

Select a path or shape. > Create panel > Geometry > Compound Objects > Object Type rollout > Loft > Creation Method rollout

Select a path or shape. > Create menu > Compounds > Loft > Creation Method rollout

You can choose between a shape or a path for creating the loft object using the Creation Method rollout, as well as the type of action for the loft object.

Interface



On the Creation Method rollout, you determine whether to use a shape or path for creating the loft object, and the type of action you want for the resulting loft object.

Get Path Assigns a path to the selected shape or changes the current assigned path.

Get Shape Assigns a shape to the selected path or changes the current assigned shape.

TIP Hold down Ctrl while getting the shape to flip the direction of the shape's Z axis.

Move/Copy/Instance Lets you specify how the path or shape is transferred to the loft object. It can be moved, in which case no copy is left behind, or transferred as a copy or an instance on page 8014.

TIP Use the Instance option if you expect to edit or modify the path after the loft is created.

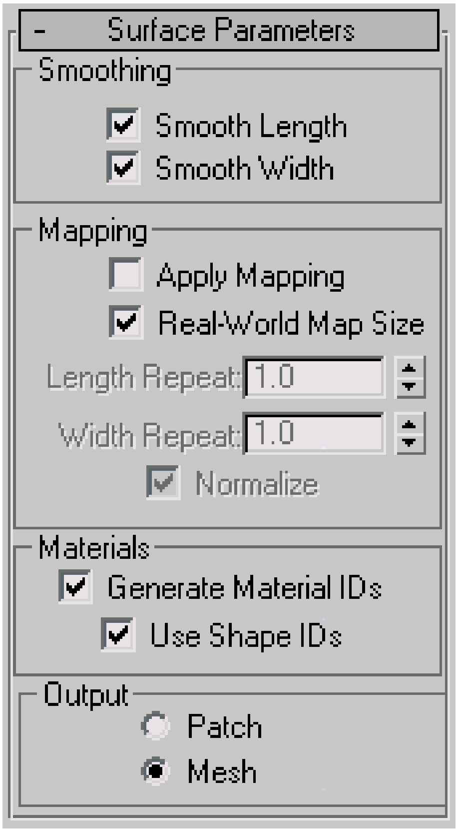
Surface Parameters Rollout

Select a path or shape. > Create panel > Geometry > Compound Objects > Object Type rollout > Loft > Surface Parameters rollout

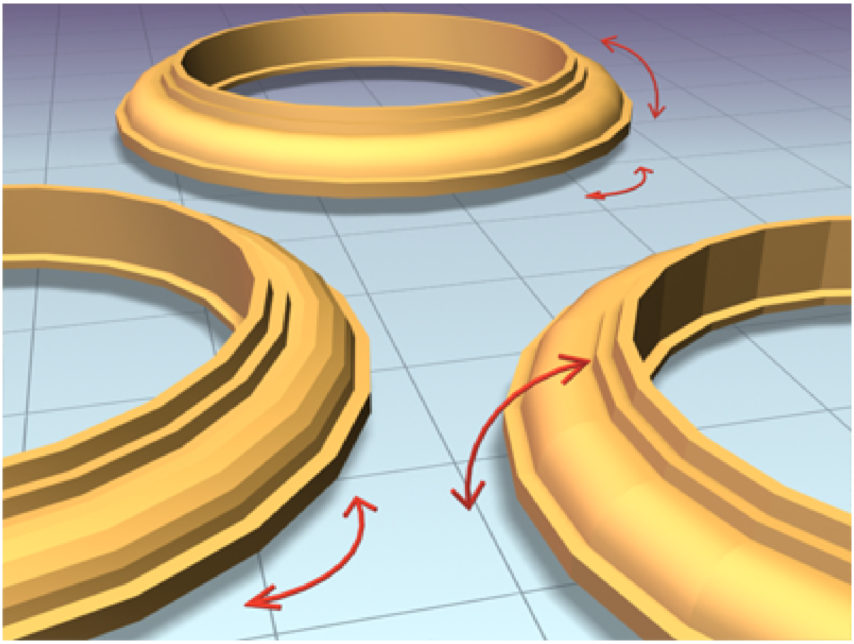
Select a path or shape. > Create menu > Compounds > Loft > Surface Parameters rollout

On the Surface Parameters rollout, you control smoothing of the surface of the loft as well as designate if texture mapping is applied along the loft object.

Interface



Smoothing group



Left: Smoothing the length

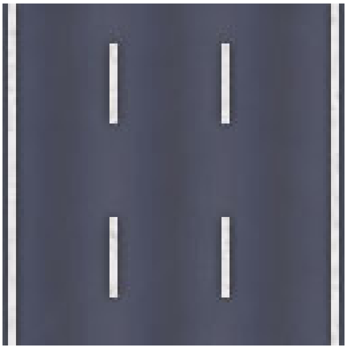
Right: Smoothing the width

Rear: Smoothing both length and width

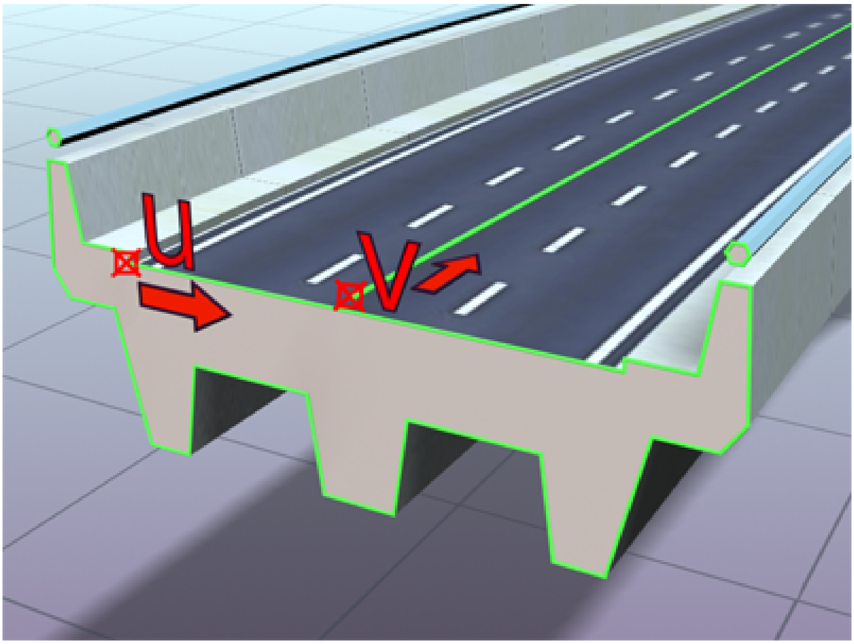
Smooth Length Provides a smooth surface along the length of the path. This type of smoothing is useful when your path curves or when shapes on the path change size. Default=on.

Smooth Width Provides a smooth surface around the perimeter of the cross-section shapes. This type of smoothing is useful when your shapes change the number of vertices or change form. Default=on.

Mapping group



Bitmap used to create the lines on the road



Mapped roadway showing U and V dimensions for the loft

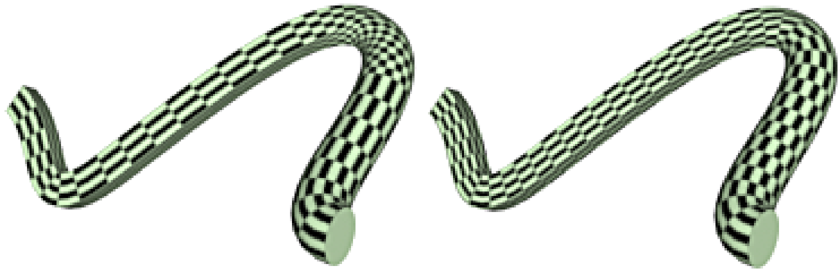
Apply Mapping Turns lofted mapping coordinates on and off. Apply Mapping must be on in order to access the remaining items.

Real-World Map Size Controls the scaling method used for texture mapped materials that are applied to the object. The scaling values are controlled by the Use Real-World Scale settings found in the applied material's Coordinates rollout on page 5782. Default=off.

Length Repeat Sets the number of times a map repeats along the length of the path. The bottom of the map is placed at the first vertex of the path.

Width Repeat Sets the number of times a map repeats around the perimeter of cross-section shapes. The left edge of a map is aligned with the first vertex of each shape.

Normalize Determines how path vertex spacing affects a map along both the path length and shape width. When on, vertices are ignored. Map coordinates and Repeat values are applied evenly along the length of the path and around the shapes. When off, major path divisions and shape vertex spacing affects map coordinate spacing. Map coordinates and Repeat values are applied proportionally according to the path division spacing or shape vertex spacing.



Before and after applying Normalize to loft

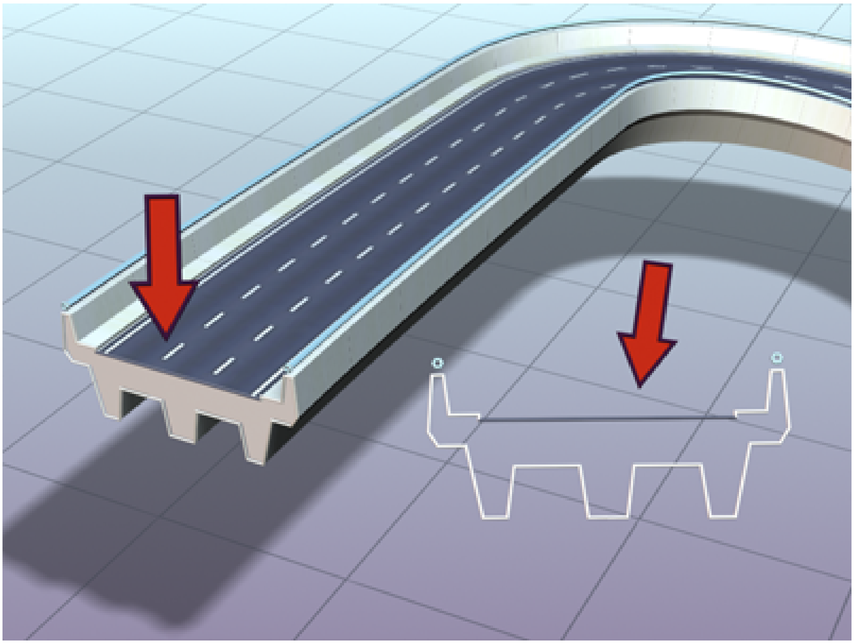
Materials group

Generate Material IDs Creates Material IDs during the loft process.

Use Shape IDs Offers the choice of using the spline material IDs to define the material IDs.

NOTE Prior to version 3 of 3ds Max, splines could not hold material IDs.

NOTE Shape IDs are inherited from shape cross sections, not from the path spline.



Shape material IDs used to give the roadway two materials: concrete for supports and railings, asphalt with white lines for the traffic lanes

Output Group

Patch The lofting process produces a patch object.

Mesh The lofting process produces a mesh object. This is the default, and was the only output type available with Loft in versions prior to version 3 of 3ds Max.

You can also create NURBS objects from lofting by choosing Convert To: NURBS from the modifier stack right-click menu on page 7648.

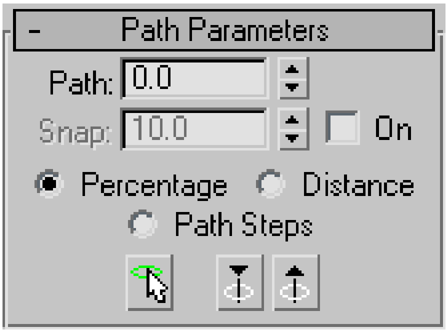
Path Parameters Rollout

Select a path or shape. > Create panel > Geometry > Compound Objects > Object Type rollout > Loft > Path Parameters rollout

Select a path or shape. > Create menu > Compounds > Loft > Path Parameters rollout

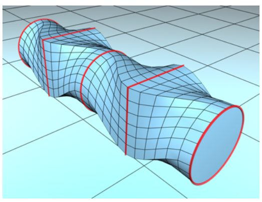
The Path Parameters rollout lets you control the position of shapes at various intervals along the path of the loft object.

Interface



On the Path Parameters rollout, you control the position of multiple shapes at different intervals along the path of the loft object.

Path Lets you set a path level by entering a value or dragging the spinner. If Snap is on, the value will jump to the previous snap increment. The Path value depends on the selected measuring method. Changing the measuring method causes the Path value to change.



Inserting different shapes at different positions on the path

Snap Lets you set a consistent distance between shapes along the path. The Snap value depends on the selected measuring method. Changing the measuring method also changes the Snap value to keep snap spacing constant.

On When On is turned on, Snap is active. Default=off.

Percentage Expresses the path level as a percentage of the total path length.

Distance Expresses the path level as an absolute distance from the first vertex of the path.

Path Steps Places shapes on path steps and vertices, rather than as a percentage or a distance along the path.

When Path Steps is on, the following take place:

- The Path spinner specifies the step along the path. The first step, at 0, is the first vertex.
- The total number of steps, including vertices, appears in parentheses beside the Path spinner.
- The current path level is indicated by the standard yellow X when it's a step, and by a small boxed X when it's a vertex.
- Get Shape places a selected shape on the specified step or a vertex of the path.
- Adaptive Path Steps on the Skin Parameters rollout is unavailable. (If it were available, the path steps and shapes would change positions along the path, depending on the result of the adaptive algorithm.)

Please note the following when using the Path Steps option:

- When you switch to Path Steps with a loft object that already contains one or more shapes, an alert message tells you that this action may relocate shapes. This is because there are a limited number of path steps, and only one shape can be on a single step or vertex. The Percentage and Distance options, on the other hand, provide an almost unlimited number of levels on which to place shapes. Thus, if you change from Percentage or Distance to Path Steps, the shapes must be moved to existing steps. If there are more shapes than can be moved to nearby steps, you could end up with more than one shape on a step. Switching from Path Steps to either Percentage or Distance, however, can always be done without loss of data.
- If you alter the Path Steps spinner while in Path Steps mode, the location of your shapes might change. An alert message warns you of this.
- If you animate the topology of the path while in Path Steps mode (such as animating the number of sides of an NGon), your shapes might jump around trying to find a legitimate position, and you could end up with more than one shape on the same path level.

Pick Shape Sets the current level at any shape on the path. When you pick a shape on the path, Snap is turned off and Path is set to the level of the picked shape, where a yellow X appears. Pick Shape is available only from the Modify panel.

Previous Shape Jumps the path level from its current location to the previous shape along the path. A yellow X appears at the current level. Clicking this button turns Snap off.

Next Shape Jumps the path level from its current location to the next shape along the path. A yellow X appears at the current level. Clicking this button turns Snap off.

Skin Parameters Rollout

Select a path or shape. > Create panel > Geometry > Compound Objects > Object Type rollout > Loft > Skin Parameters rollout

Select a path or shape. > Create menu > Compounds > Loft > Skin Parameters rollout

On the Skin Parameters rollout, you adjust the complexity of the mesh of the loft object. You can also optimize the mesh by controlling the face count.

Procedures

Example: To use a constant cross-section:

1. Enlarge the Front viewport to full screen, and then draw a Rectangle object on page 624 with Ctrl held down to create a square about 20 x 20 units.
2. Create another rectangle beside it about 200 x 100 units.
3. Apply a Skew modifier to the large rectangle, but don't alter the Skew parameters.
4. Create a loft object in which the larger rectangle is the path and the square is the shape.
5. On the Modify panel, open the Skin Parameters rollout, and make sure Skin is on in the Display group.

   You can now see the wireframe structure of the lofted rectangle, with cross-sectional sides parallel to its corners.

   Make sure the color assigned the loft object is easily visible. Change it if necessary.

6 Turn off Constant Cross-Section, and observe the corners.

When Constant Cross-Section is off, the corners become pinched.

7 Turn on Constant Cross-Section to restore the corners.

Acute angles can cause problems when the cross sections formed by the path steps intersect at the corners. You can mitigate this by avoiding acute angles or by reducing the path steps.

8 Press H on the keyboard to open the Select From Scene dialog on page 228, and select *Rectangle02* (the second larger rectangle).

9 On the Modify panel, change the Skew Axis to **Y**, and then set the Amount spinner to **95**.

10 Use Zoom Region to zoom in on the upper-right corner of the rectangle so you can see the mesh in detail.

At a skew of less than 100, the acute angle still works because the path cross-sections haven't intersected.

11 Set the Skew Amount to **300**, and examine the same corner.

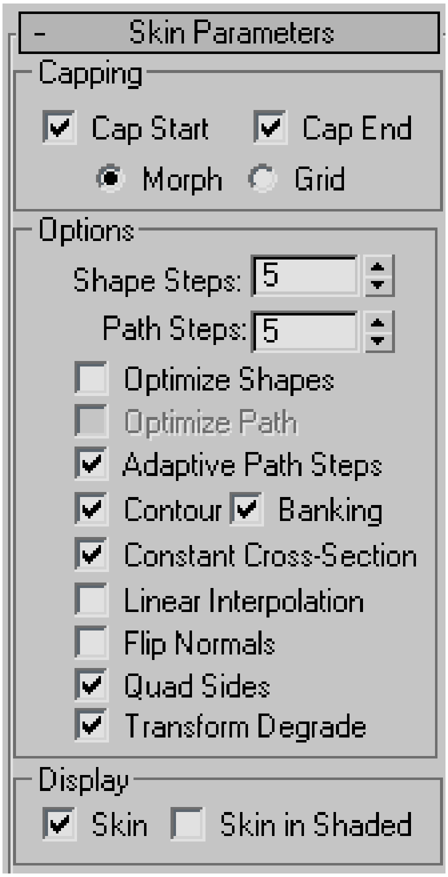
At this angle, the path cross sections intersect, causing problems in the mesh.

12 Select the loft object, and set Path Steps to **1**.

The cross sections no longer intersect, and the corner is clean.

When creating straight-edge molding for architectural modeling, you can avoid mangled corners simply by reducing the path steps to 0.

Interface



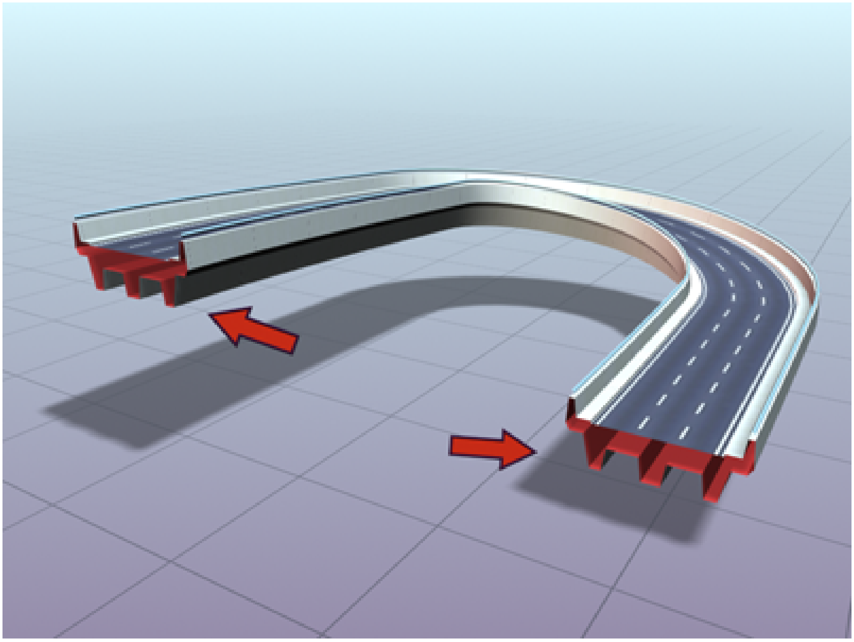
Capping group

Cap Start When on, the end of a loft at the first vertex of the path is covered, or capped. When off, the end is open, or uncapped. Default=on.

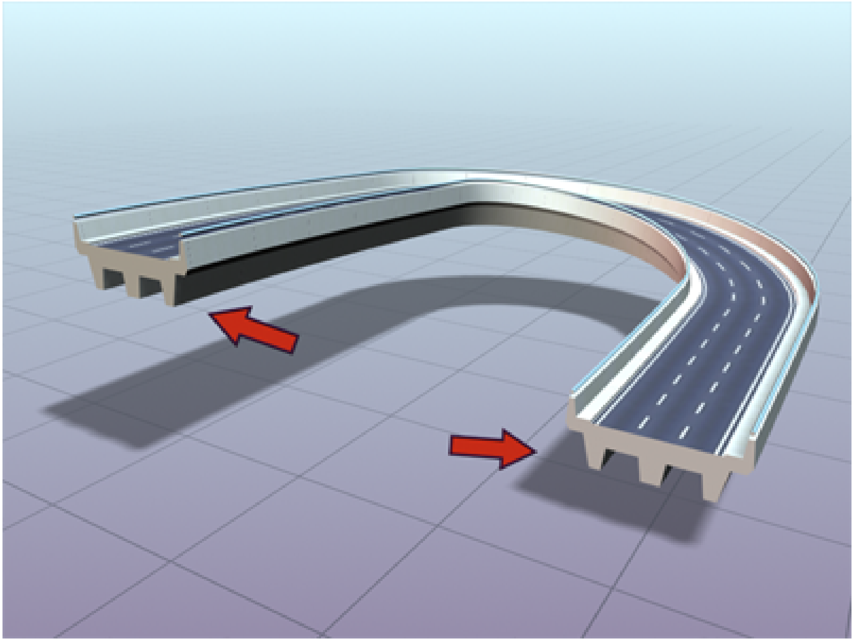
Cap End When on, the end of a loft at the last vertex of the path is covered, or capped. When off, the end is open, or uncapped. Default=on.

Morph Arranges cap faces in a predictable, repeatable pattern necessary for creating morph targets. Morph capping can generate long, thin faces that do not render or deform as well as those created with grid capping.

Grid Arranges cap faces in a rectangular grid trimmed at the shape boundaries. This method produces a surface of evenly sized faces that can be deformed easily by other modifiers.



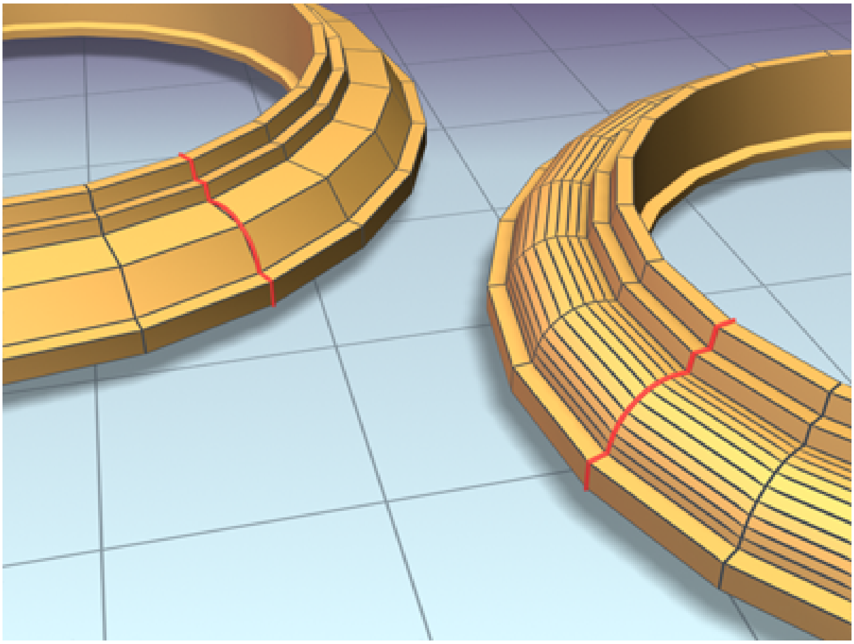
Roadway lofted with capping turned off



Roadway lofted with capping turned on

Options group

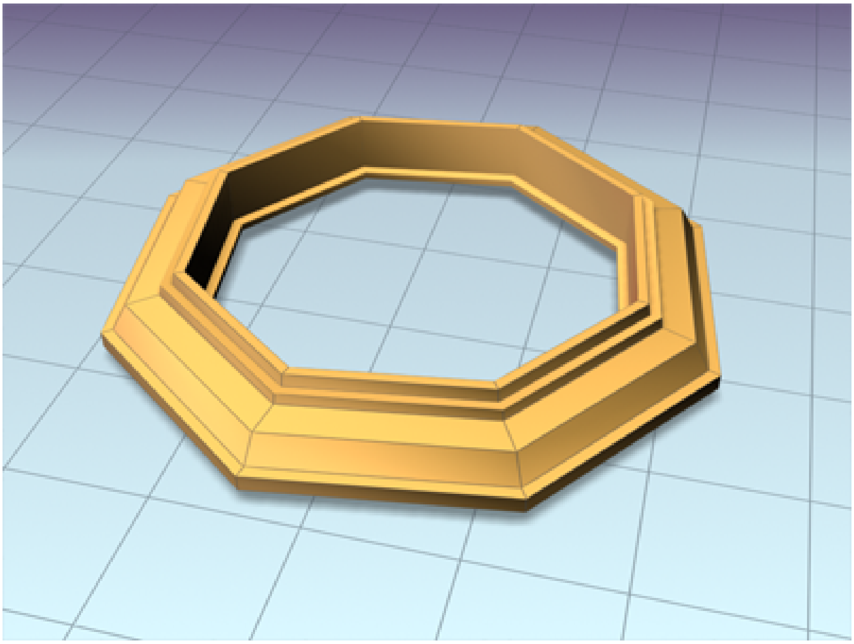
Shape Steps Sets the number of steps between each vertex of the cross-section shapes. This value affects the number of sides around the perimeter of the loft.



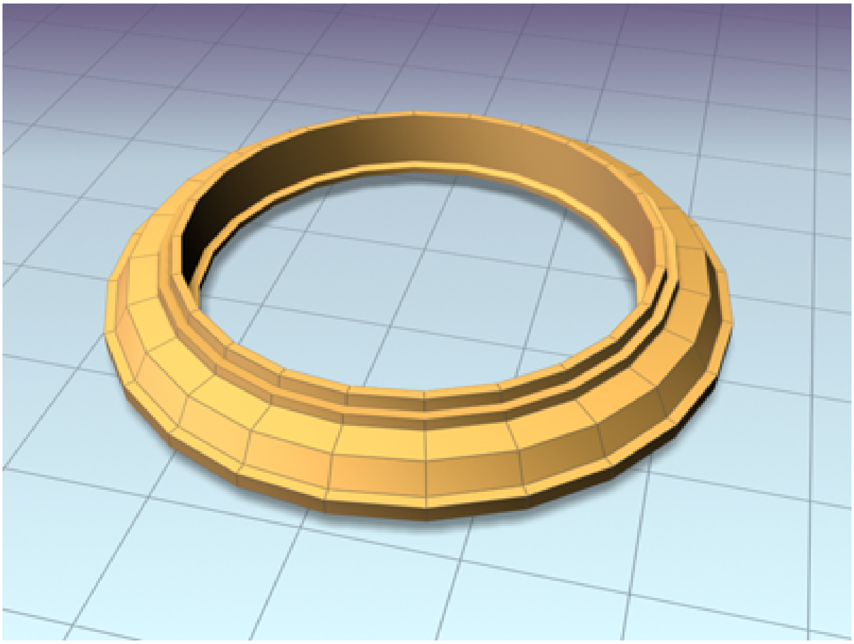
Left: Shape Steps=0.

Right: Shape Steps=4.

Path Steps Sets the number of steps between each main division of the path. This value affects the number of segments along the length of the loft.

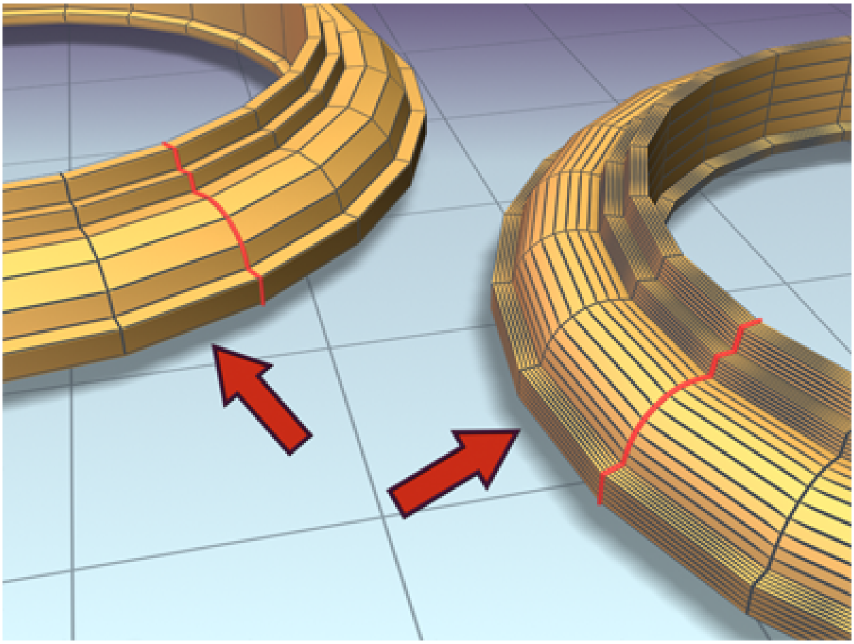


Frame lofted with Path Steps=1



Frame lofted with Path Steps=5

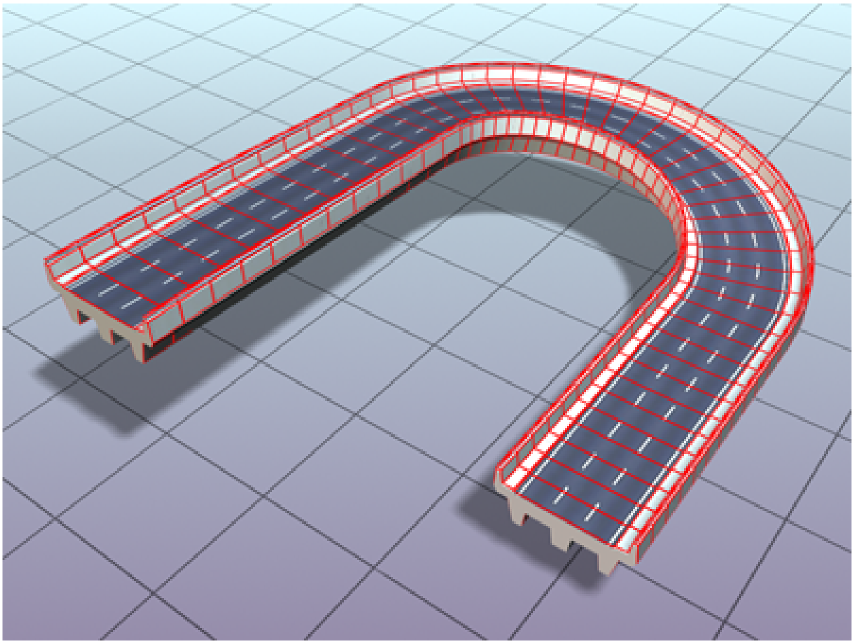
Optimize Shapes When on, the Shape Steps setting is ignored for straight segments of cross-section shapes. If multiple shapes are on the path, only straight segments that have a match on all shapes are optimized. Default=off.



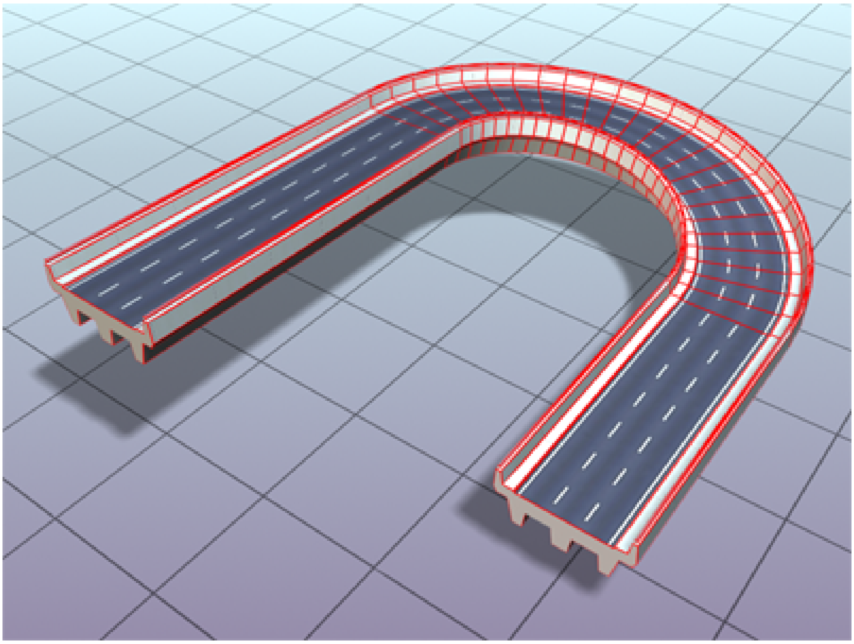
Left: Optimize Shapes turned on

Right: Optimize Shapes turned off

Optimize Path When on, the Path Steps setting is ignored for straight segments of the path. Curved sections respect the Path steps setting. Available only with Path Steps mode. Default=off.



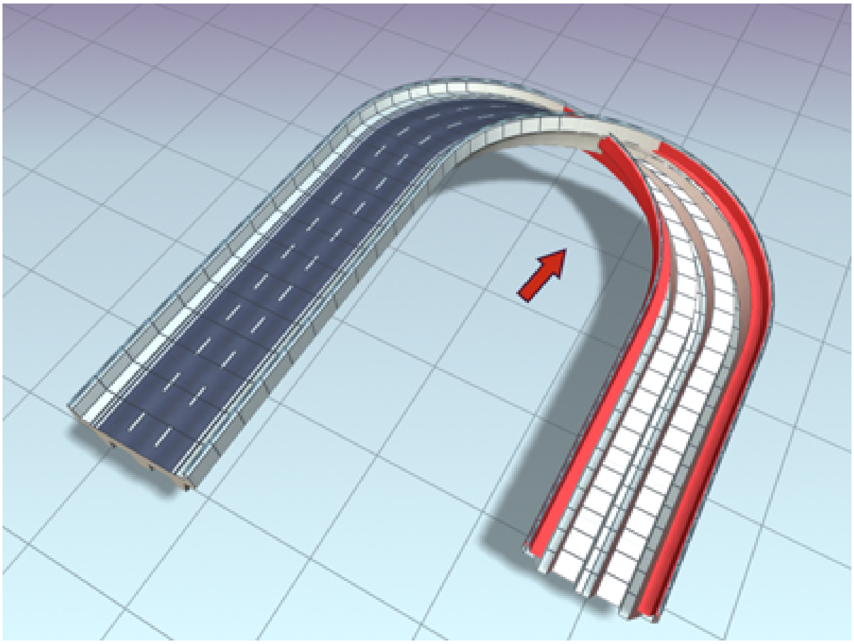
When Optimize Path is off, the lofted roadway uses more steps.



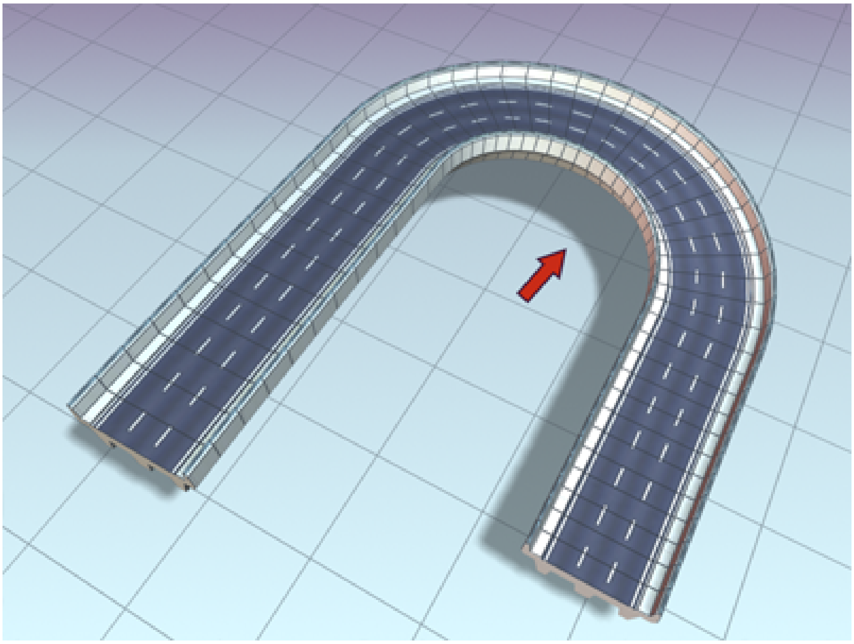
When Optimize Path is on, straight sections of the lofted roadway don't require additional steps.

Adaptive Path Steps When on, analyzes the loft and adapts the number of path divisions to generate the best skin. Main divisions along the path occur at path vertices, shape locations, and deformation curve vertices. When off, main divisions along the path occur only at path vertices. Default=on.

Contour When on, each shape follows the curvature of the path. The positive Z axis of each shape is aligned with the tangent to the path at the shape's level. When off, shapes remain parallel and have the same orientation as a shape placed at level 0. Default=on.

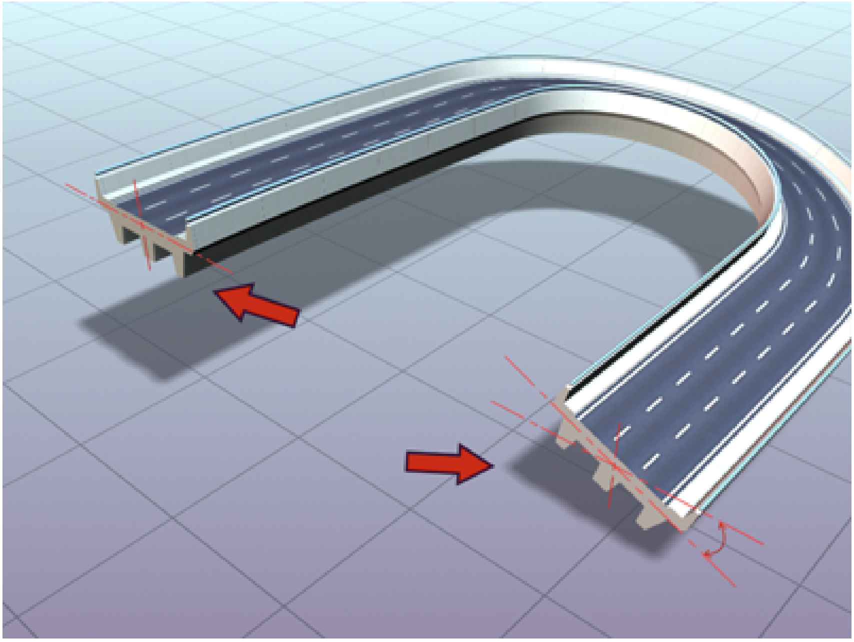


Lofting the roadway with Contour off causes it to twist.



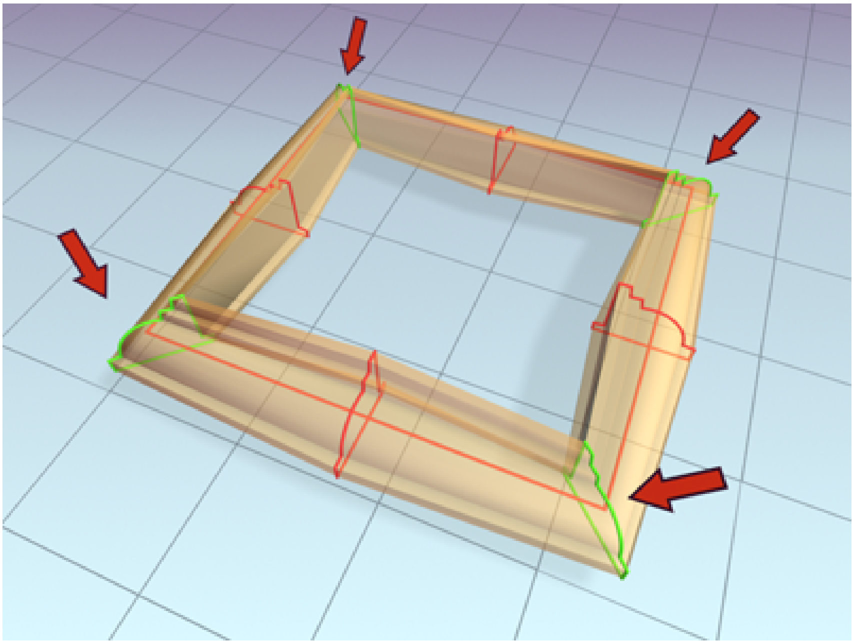
Roadway lofted with Contour turned on

Banking When on, shapes rotate about the path whenever the path bends and changes height in the path's local Z axis. The bank amount is controlled by 3ds Max. Banking is ignored if the path is 2D. When off, shapes do not rotate about their Z axis as they traverse a 3D path. Default=on.

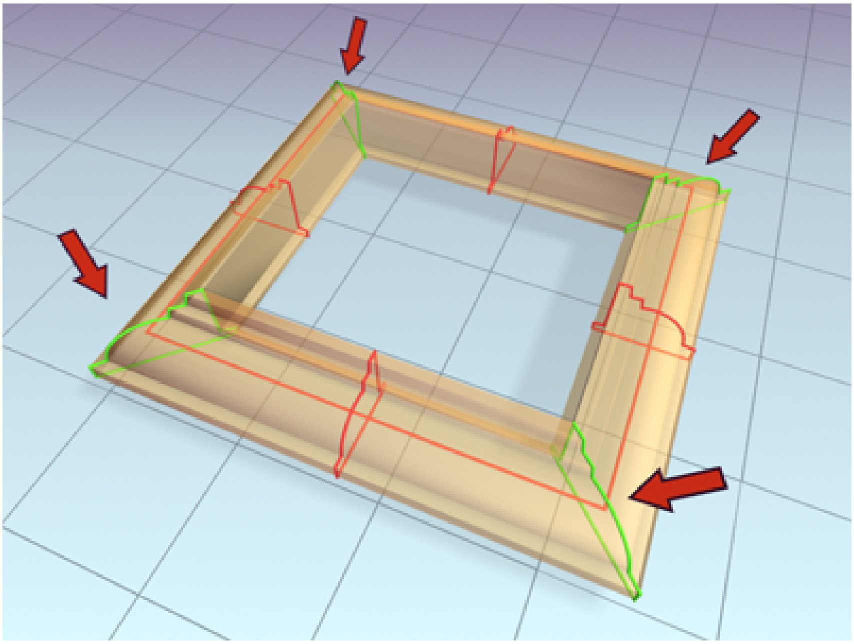


Roadway lofted with Banking turned on

Constant Cross Section When on, the cross sections are scaled at angles in the path to maintain uniform path width. When off, the cross sections maintain their original local dimensions, causing pinching at path angles.

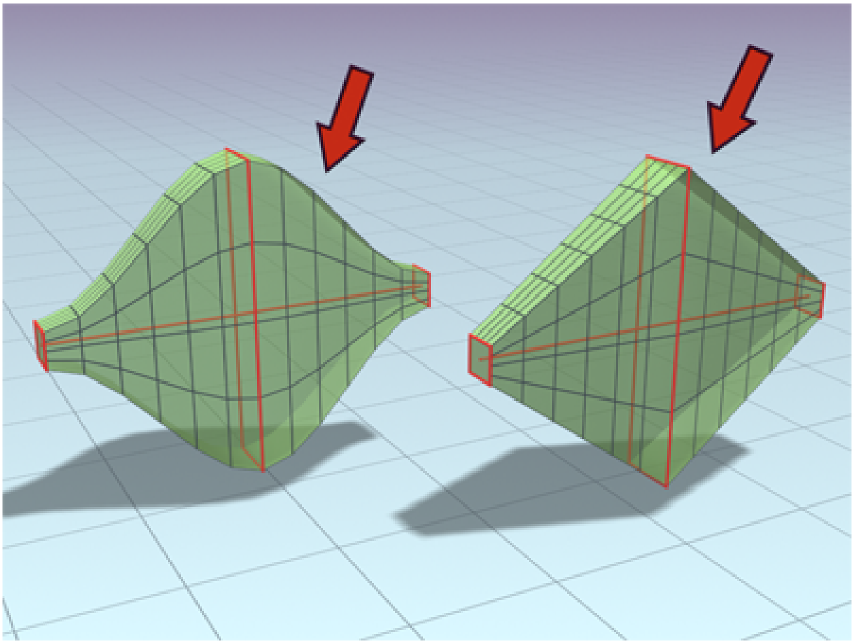


Frame lofted with Constant Cross Section turned off



Frame lofted with Constant Cross Section turned on

Linear Interpolation When on, generates a loft skin with straight edges between each shape. When off, generates a loft skin with smooth curves between each shape. Default=off.



Left: Object lofted with Linear Interpolation turned off

Right: Object lofted with Linear Interpolation turned on

Flip Normals When on, reverses the normals 180 degrees. Use this option to correct objects that are inside-out. Default=off.

Quad sides When on, and when two sections of a loft object have the same number of sides, the faces that stitch the sections together are displayed as quads. Sides between sections with different numbers of sides are not affected, and are still connected with triangles. Default=off.

Transform Degrade Causes the loft skin to disappear during sub-object shape/path transformations. For example, moving a vertex on the path causes the loft to disappear. When off, you can see the skin during these Sub-Object transformations. Default=off.

Display group

Skin When on, displays a loft's skin in all views using any shading level and ignores the Skin In Shaded setting. When off, displays only the loft sub-objects. Default=on.

Skin in Shaded When on, displays a loft's skin in shaded views regardless of the Skin setting. When off, skin display is controlled by the Skin setting. Default=on.

The loft object now retains the Skin and Skin In Shaded settings from one loft object to the next one created.

Deformations

Select a Loft object. > Modify panel > Deformations rollout

Deformation controls let you scale, twist, teeter, bevel or fit shapes along the path. The interface for all deformations is a graph. Lines with control points on the graph represent the deformations along the path. Control points on the graphs can be moved or animated for modeling purposes or for various special effects.

Manually creating and placing shapes along the path to produce these models would be a difficult task. Lofts solve this problem through the use of deformation curves. The deformation curves define changes in scale, twisting, teetering, and beveling along the path.

You gain access to loft deformation curves through the Modify panel's Deformations rollout. Deformations are not available in the Create panel. You must open the Modify panel after you've lofted to access the Deformations rollout, which offers the following features:

- Each deformation button displays its own deformation dialog.
- You can display any or all of the deformation dialogs simultaneously.
- The button to the right of each deformation button is a toggle to enable or disable the deformation's effect.

See also:

- Deformation Dialog on page 826

Procedures

To apply deformations to a loft:

1 Select a loft object.

2 Go to the Modify panel and choose Loft from the modifier stack display if it's not already displayed.

3 Expand the Deformations rollout.

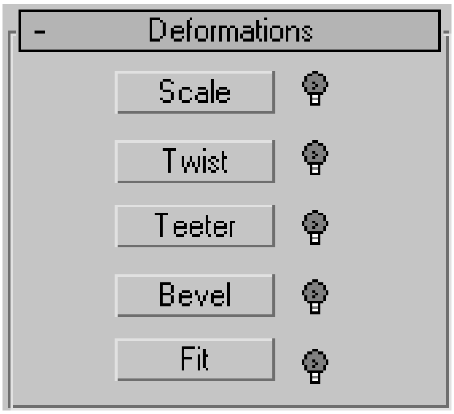
4 Click the deformation that you want to use.

   The window for the selected deformation appears.

To toggle the deformation effect:

- Click Enable/Disable to the right of the deformation buttons.

Interface



Deform Scale on page 815

Deform Twist on page 817

Deform Teeter on page 819

Deform Bevel on page 822

Deform Fit on page 824

Deform Scale

Select a Loft object. > Modify panel > Deformations rollout > Scale

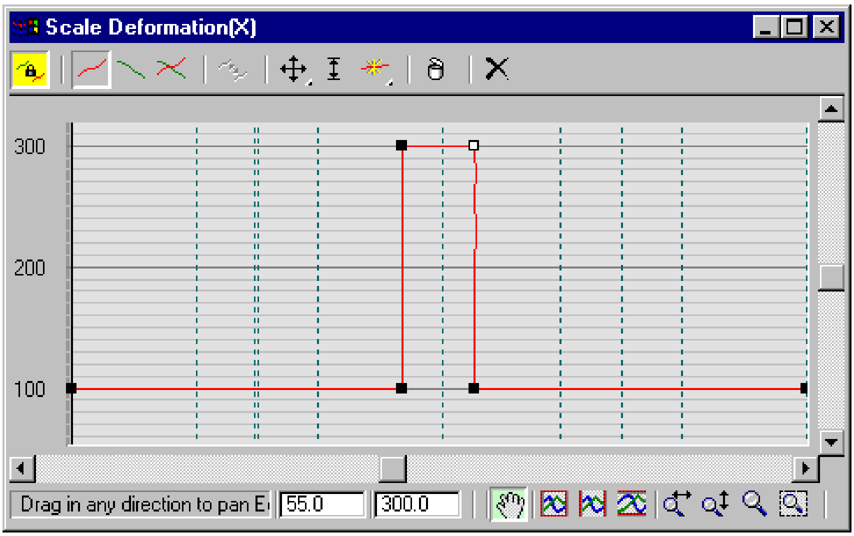
You can loft objects such as columns and bugles from a single shape that changes only its scale as it travels along a path. Use Scale deformation when you want to make these types of objects.

TIP By animating scale, a loft object can appear to travel along a path. Using this technique, you can create animations in which letters or lines write themselves onto the screen.

These are the properties of Scale deformation curves:

- The two curves are red for X-axis scaling and green for Y-axis scaling.
- Default curve values are at 100%.
- Values greater than 100% make the shape larger.
- Values between 100% and 0% make the shape smaller.
- Negative values scale and mirror the shape.

See Deformation Dialog on page 826 for specific information on the dialog controls.



Scale deformation curve dialog

Procedures

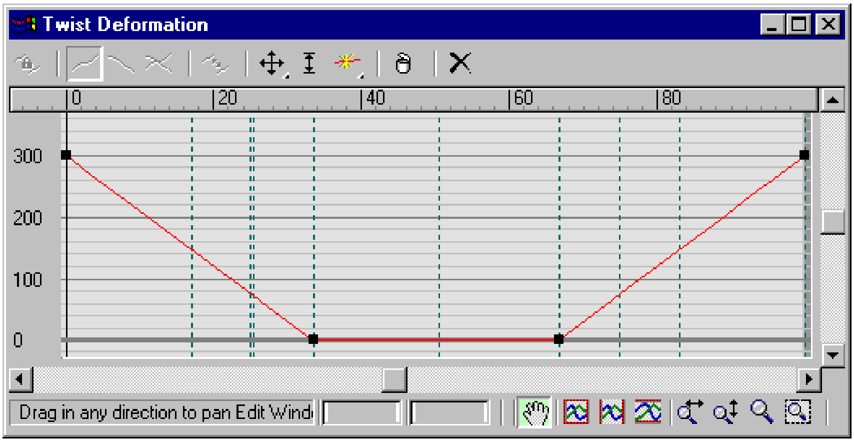
To use Scale deformation:

1. Select a loft object.
2. Click Loft in the modifier stack display.
3. Click Scale on the Deformations rollout.
4. Edit the deformation curves for the X axis and Y axis.

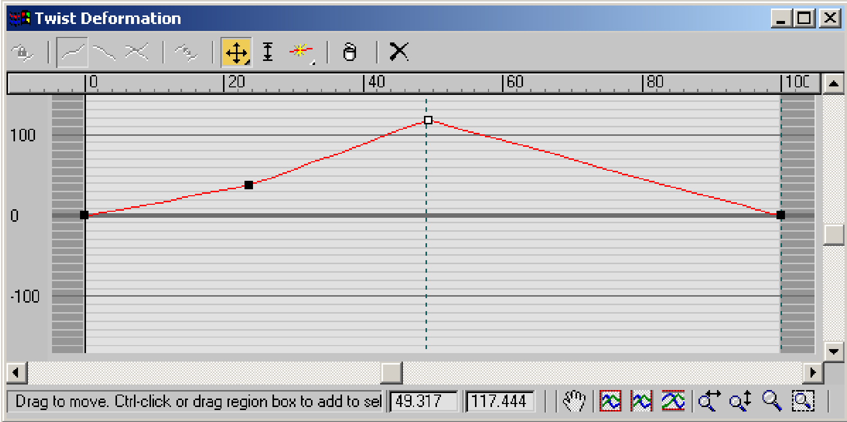
Deform Twist

Select a Loft object. > Modify panel > Deformations rollout > Twist

Twist deformation lets you create objects that spiral or twist along their length. Twist specifies the amount of rotation about the path.



Twist deformation curve dialog



Using twist to deform the lofted roadway

These are the properties of Twist deformation curves:

- A single red curve determines shape rotation about the path.
- The default curve value is 0 degrees of rotation.
- Positive values produce counterclockwise rotation, when viewed from the start of the path.
- Negative values produce clockwise rotation.
- Both twist deformation and banking produce rotation about the path. Twist rotation is added to a shape after the banking angle is applied. You can use Twist deformation to exaggerate or reduce the amount of banking.

See Deformation Dialog on page 826 for specific information on the dialog controls.

Procedures

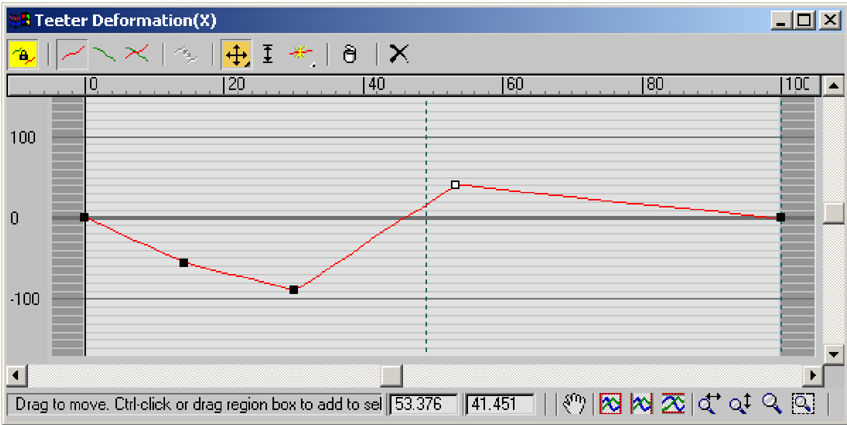
To use Twist deformation:

1. Select a loft object.
2. Click Loft in the modifier stack display.
3. Click Twist on the Deformations rollout.
4. Edit the single deformation curve to specify rotation about the path.

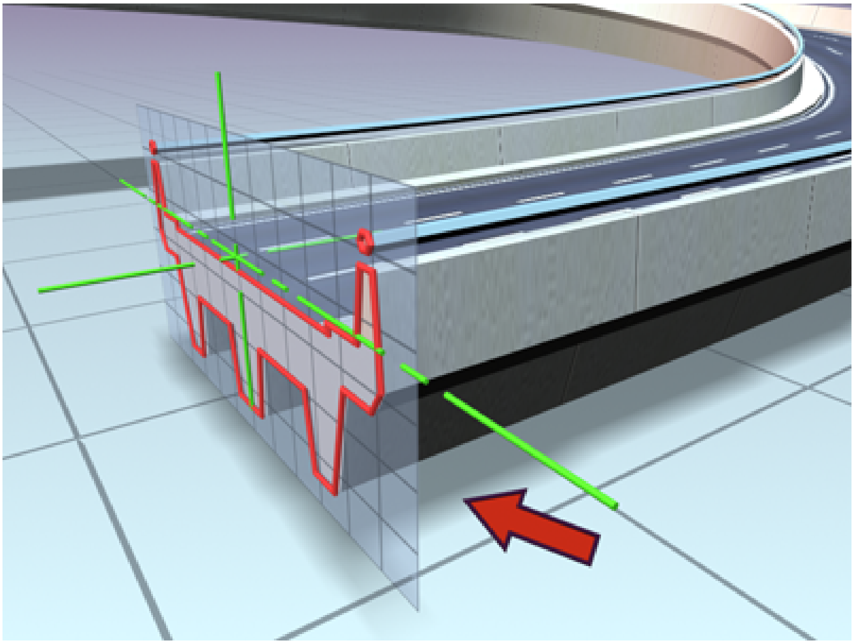
Deform Teeter

Select a Loft object. > Modify panel > Deformations rollout > Teeter

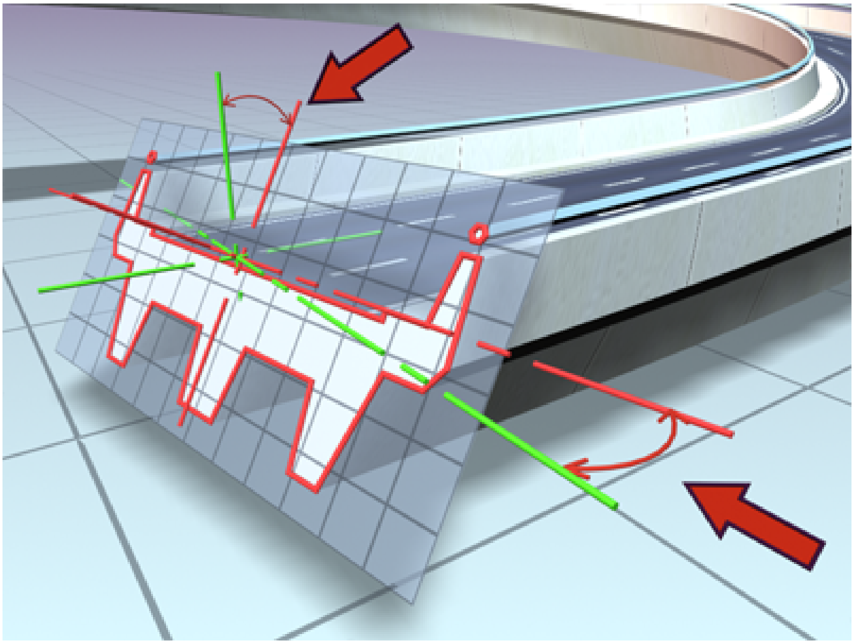
Teeter deformation rotates shapes about their local X axis and Y axis. Teetering is what 3ds Max does automatically when you select Contour on the Skin Parameters rollout. Use Teeter deformation when you want to manually control contour effects.



Teeter deformation curve dialog



Roadway lofted with no teeter



Roadway lofted with teeter turned on. Teeter affects the X and Y axis orientation of the shape in relation to the path.

These are the properties of Teeter deformation curves:

- The two curves are red for X-axis rotation and green for Y-axis rotation.
- Default curve values are at 0 degrees rotation.
- Positive values rotate the shape counterclockwise about the shape's positive axis.
- Negative values rotate the shape clockwise about the shape's positive axis.

See Deformation Dialog on page 826 for specific information on the dialog controls.

Procedures

To use Teeter deformation:

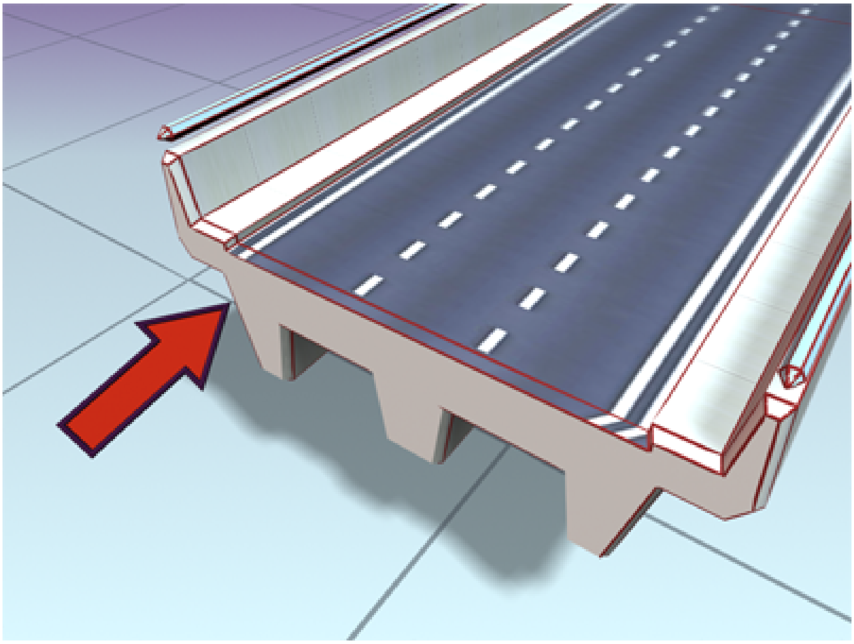
1. Select a loft object.
2. Click Loft in the modifier stack display.

3 Click Teeter on the Deformations rollout.

4 Edit the deformation curves for X axis and Y axis rotation.

Deform Bevel

Select a Loft object. > Modify panel > Deformations rollout > Bevel



Roadway with beveled edges

Nearly every object that you encounter in the real world is beveled. Because it is difficult and expensive to manufacture a perfectly sharp edge, most objects are created with chamfered, filleted, or eased edges. Use Bevel deformation to simulate these effects.

NOTE Bevel is not available when loft output is set to Patch.

These are the properties of Bevel deformation curves:

- The single red curve is for bevel amount.
- Bevel values are specified in current units.

- The default curve value is 0 units.
- Positive values reduce the shape, bringing it closer to the path.
- Negative values add to the shape, moving it away from the path.

When shapes are nested, the bevel direction is reversed for interior shapes.

See Deformation Dialog on page 826 for specific information on the dialog controls.

Normal and Adaptive Beveling

The Bevel Deformation dialog provides three types of beveling: Normal, Adaptive Linear, and Adaptive Cubic. These are available from a flyout at the right end of the dialog toolbar.

With normal beveling, the beveled shape remains parallel to the original, regardless of the crotch angle of the shape. Steep crotch angles combined with excessive bevel amounts result in overshooting at the crotch.

Adaptive beveling alters the length of the bevel shape based on the crotch angle. Adaptive Linear alters the length-to-angle in a linear fashion. Adaptive Cubic alters it more on steep angles than on shallow angles, producing a subtly different effect. Both forms of adaptive beveling result in nonparallel beveled edges, and both are less likely to produce invalid bevels due to overshoots at the crotch.

To see the differences in the three types of beveling, loft a star shape along a straight path and apply a bevel. When you switch among the three types of beveling, you'll see the difference in the bevel outline. Alter one radius of the star to examine the beveling with shallow and with sharp crotch angles.

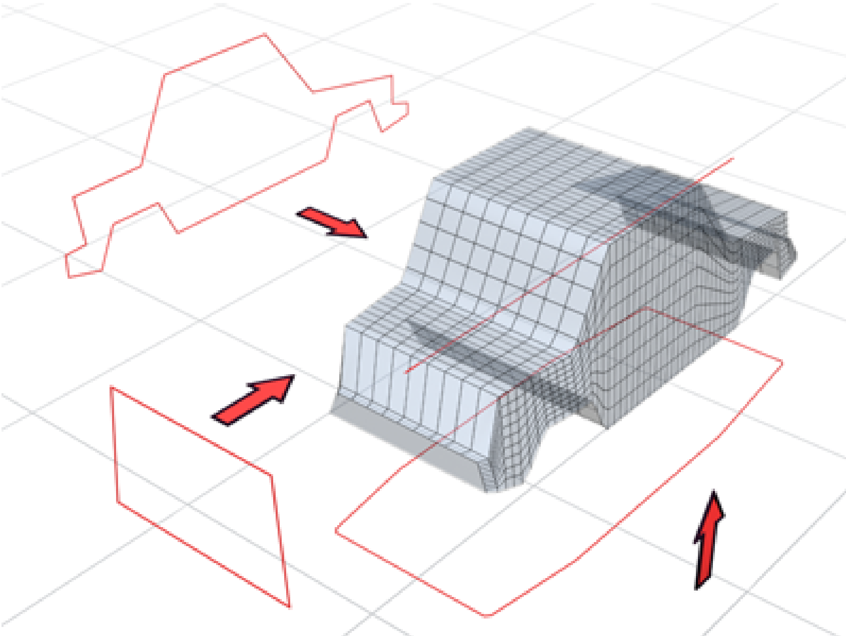
Procedures

To use Bevel deformation:

1. Select a loft object.
2. Click Loft in the modifier stack display.
3. Click Bevel on the Deformations rollout.
4. Adjust the deformation curve.

Deform Fit

Select a Loft object. > Modify panel > Deformations rollout > Fit



Fit curves define a lofted shape.

Fit deformation lets you use two Fit curves to define the top and side profiles of your object. Use Fit deformation when you want to generate loft objects by drawing their profiles.

Fit shapes are really scale boundaries. As your cross-section shape travels along the path, its X axis is scaled to fit the boundaries of the X-axis fit shape and its Y axis is scaled to fit the boundaries of the Y-axis fit shape.

NOTE Fit is not available when loft output is set to Patch.

Procedures

To use Fit deformation:

1 Select a loft object.

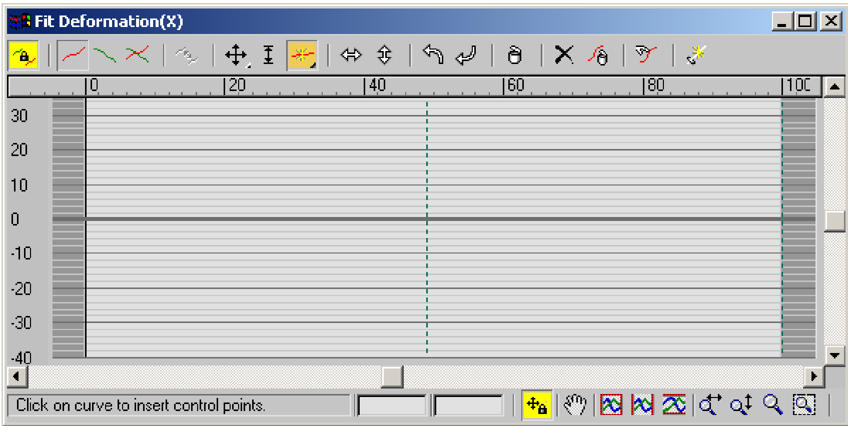
2 Click Loft in the modifier stack display.

3 Click Fit on the Deformations rollout.

4 Select shapes in the viewport to use as fit curves.

Interface

Fit Deformation dialog



The Fit Deformation dialog contains different buttons than the other deformations. For descriptions of the first eight buttons on the toolbar, see Deformation Dialog on page 826. The following descriptions apply to the tools specific to Fit deformation, and are listed from left to right in the order they appear on the toolbar.

Fit Deformation toolbar

Mirror Horizontally Mirrors the shape across the horizontal axis.

Mirror Vertically Mirrors the shape across the vertical axis.

Rotate 90 CCW Rotates the shape 90 degrees counterclockwise.

Rotate 90 CW Rotates the shape 90 degrees clockwise.

Delete Control Point Deletes the selected control point.

Reset Curve Replaces the displayed Fit curve with a rectangle 100 units wide and centered on the path. If Make Symmetrical is on, both Fit curves are reset even though only one might be displayed.

Delete Curve Deletes the displayed Fit curve. If Make Symmetrical is on, both Fit curves are deleted even though only one might be displayed.

Get Shape Lets you select the shape to use for Fit deformation. Click Get Shape, and then click the shape to use in a viewport.

Generate Path Replaces the original path with a new straight-line path.

Deformation Dialog

Select a Loft object. > Modify panel > Deformations rollout > Scale, Twist, Teeter, Bevel, or Fit

The Deformation dialogs for Scale, Twist, Teeter, Bevel, and Fit use the same basic layout. The buttons in the window's toolbar and prompt area perform the following functions:

- Change deformation curve display.
- Edit control points (these can be animated).
- Navigate the Deformation dialog.

Editing Deformation Curves

A deformation curve starts as a straight line using a constant value. To produce more elaborate curves, you insert control points and change their properties.

Use the buttons in the center of the Deformation dialog toolbar to insert and change deformation curve control points (see Interface, later in this topic).

Control Point Types

Control points on a deformation curve can produce curves or sharp corners, depending on the control point type. To change a control point type, right-click the control point and choose one of these from the shortcut menu:

- **Corner** Non-adjustable linear control point producing a sharp corner.
- **Bezier Corner** Adjustable Bezier control point with discontinuous tangent handles set to produce a sharp corner. This type produces a curve that

looks like the corner type but has control handles like the Bezier Smooth type.

- **Bezier Smooth** Adjustable Bezier control point with locked continuous tangent handles set to produce a smooth curve.

Selecting Control Points

Use the Move Control Point and Scale Control Point buttons with standard selection techniques to select control points.

Procedures

To drag Bezier tangent handles:

1. Select one or more Bezier Smooth or Bezier Corner control points to display their tangent handles.
2. Click one of the Move Control Point buttons.
3. Drag any tangent handle.
   - Only the tangent handle you drag is affected. Tangent handles on other selected control points do not change.
   - If the tangent handle you drag is part of a Bezier Smooth control point, both handles move to maintain the Bezier Smooth type.
   - If the tangent handle you drag is part of a Bezier Corner control point, only that handle moves.

To move a control point using the Position and Amount fields:

1. Select a single control point.
2. Do one of the following:
   - Move the control point horizontally by entering a value in the Position field.
   - Move the control point vertically by entering a value in the Amount field.

To change the control point type:

You can change control point types at any time by right-clicking a selection of one or more control points.

1 Select one or more control points.

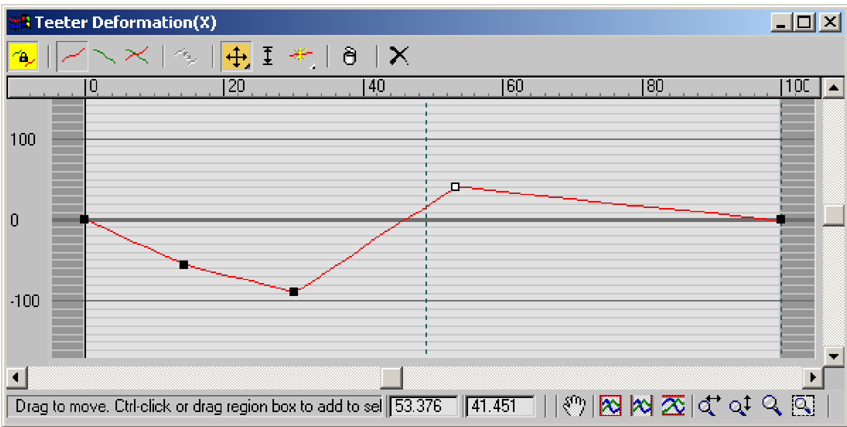
2 Right-click any selected control point.

3 Choose a control point type from the shortcut menu.

The following conditions apply to changing control point types:

- The first and last control points must use the Corner or Bezier Corner type.
- Converting a Bezier Smooth point to a Bezier Corner point unlocks the tangent handles but does not change their position. The curve appears smooth until you drag one of the tangent handles.
- Converting a Bezier Corner point or inserted Bezier point to Bezier Smooth locks the tangent handles and changes their position and magnitude. The handles are rotated to the average between their two angles. The handle magnitudes are averaged and set equal.

Interface



Toolbar

Buttons for working with a second curve are disabled for the Twist and Bevel deformations, which use only one curve. The disabled buttons are Make

Symmetrical, Display X Axis, Display Y Axis, Display XY Axes, and Swap Deform Curves.

Make Symmetrical You can apply the same deformation to both axes of a shape using Make Symmetrical, which is both an action button and a curve editing mode. Turning on Make Symmetrical has the following effect:

- When a single curve is displayed, it copies the displayed deformation curve to the curve for the hidden axis.
- When both axes are displayed, the Apply Symmetry dialog is also displayed. Click the button for the curve you want to apply to both axes.
- Changes you make to the selected curve are duplicated on the other curve.

When Make Symmetrical is not active, curve editing is applied only to the selected curve.

Display X Axis/Y Axis/XY Axes You can display one or both deformation curves using the curve display buttons near the upper-left corner of the Deformation dialog.

Turn on the following buttons to display deformation curves:

- **Display X Axis** Displays only the X axis deformation curve in red.
- **Display Y Axis** Displays only the Y axis deformation curve in green.
- **Display XY Axes** Displays X axis and Y axis deformation curves together, each using its own color.

Swap Deform Curves Copies curves between the X axis and Y axis. This button has no effect when Make Symmetrical is on.

Click Swap Deform Curves to copy the X axis curve to the Y axis, and the Y axis curve to the X axis. It doesn't matter which curve is currently displayed or selected.

Move Control Points This flyout contains three buttons for moving control points and Bezier handles:

- **Move Control Point** Changes the amount of deformation (vertical movement) and the location of the deformation (horizontal movement).
- **Move Vertical** Changes the amount of deformation without changing the location.
- **Move Horizontal** Changes the location of the deformation without changing the amount.

If one control point is selected, you can move it by entering values in the control point Position and Amount fields at the bottom of the Deformation dialog.

You cannot move end points horizontally. Intermediate control points are constrained horizontally to stay between the points on either side. The amount of horizontal constraint is determined by the control point type.

- You can move corner control points very close together, until one is directly above the other.
- You can move Bezier control points no closer than the length of their tangent handles.

Moving Bezier Tangent Handles You can use the Move Control Point buttons to drag a tangent handle's angle and magnitude on Bezier Smooth and Bezier Corner vertices.

Dragging a tangent handle has the following constraints:

- You cannot move tangent angles beyond vertical. This prevents deformation curves from doubling back on themselves.
- You cannot move tangent magnitudes beyond the preceding or next control point on the path.

Pressing Shift while moving a Bezier Smooth tangent handle converts the control point to a Bezier Corner type.

Scale Control Point Scales the value of one or more selected control points with respect to 0. Use this function when you want to change only the deformation amounts of selected control points while maintaining their relative ratio of values.

- Drag downward to reduce values.
- Drag upward to increase values.

Insert Control Point This flyout contains buttons for inserting two control point types.

Insert Corner Point Click anywhere on a deformation curve to insert a corner control point at that location.

Insert Bezier Point Click anywhere on a deformation curve to insert a modified Bezier control point at that location. The tangent handles of the Bezier control point are set to maintain the shape of the curve before the point was inserted.

If you are not sure which type of control point you need, or if you change your mind, you can convert the point to another type by right-clicking the point and selecting the type from the shortcut menu.

Both Insert Control Point buttons put you in insertion mode. Right-click or choose another button to exit the mode.

Delete Control Point Deletes selected control points. You can also delete selected points by pressing the DELETE key.

Reset Curve Deletes all but the end control points and sets the curves back to their default values.

Bevel Type This flyout, available only in the Bevel Deformation dialog, lets you choose Normal, Adaptive Linear or Adaptive Cubic as the bevel type. For more information, see Deform Bevel on page 822.

Deformation grid

The area in the Deformation dialog that displays the deformation curves is called the deformation grid. This grid charts the value of the deformation along the length of the path.

These are the main grid components:

Active area The light-colored area of the grid defines the first and last vertex boundaries of the path. The ends of the deformation curve lie on each boundary and cannot be moved off the boundary.

Horizontal lines Mark deformation values on the vertical scale. The following table lists each deformation curve type and the meaning of the deformation values.

Deformation Type	Deformation Value
Scale	Percentage
Twist	Rotation Angle
Teeter	Rotation Angle
Bevel	Current Units

The thick horizontal line at 0 represents the deformation value at the loft path.

Vertical lines Mark levels of the path. The levels displayed vary with the Adaptive Path Steps setting on the Skin Parameters rollout on page 797.

If Adaptive Path Steps is on, levels are displayed at all path vertices and shape locations.

If Adaptive Path Steps is off, levels are displayed only at path vertices.

Path ruler Measures the length of the path. The values on the ruler measure percentage along the path. You can drag the path ruler vertically in the Deformation dialog.

Deformation curves You can see one or two curves in the Deformation dialog, based on the deformation type and the curve display setting. The curves are color-coded by axis.

A red curve displays deformation along the shape's local X axis. A green curve displays deformation along the shape's local Y axis.

Control Point fields At the bottom of the Deformation dialog are two edit fields. When a single control point is selected these fields display the path location and deformation amount of the control point.

Control Point Position The left field displays the location of the control point on the loft path as a percentage of the total path length.

Control Point Amount The right field displays the deformation value of the control point.

Deformation Dialog status bar

The Deformation dialogs have their own view navigation buttons in the lower-right corner. These give you controls for zooming and panning the view of the deformation grid as you edit the curve values. The status bar also displays information about the current tool and the selected control point.

Numeric fields These two fields are accessible only if a single control point is selected. The first gives the point's horizontal position, and the second gives its vertical position, or value. You can edit these fields with the keyboard.

Lock Aspect This button is present only in the Fit Deformation dialog. When active, it restricts zooming to vertical and horizontal at the same time.

Zoom Extents Changes the view magnification so the entire deformation curve is visible.

Zoom Horizontal Extents Changes the view magnification along the path length so the entire path area is visible in the dialog.

Zoom Vertical Extents Changes the view magnification along the deformation values so the entire deformation curve is displayed in the dialog.

Zoom Horizontally Changes magnification along the path length.

- Drag to the right to increase magnification.
- Drag to the left to decrease magnification.

Zoom Vertically Changes magnification along the deformation value.

- Drag upward to increase magnification.
- Drag downward to decrease magnification.

Zoom Changes magnification along both the path length and the deformation value, preserving the curve aspect ratio.

- Drag upward to increase magnification.
- Drag downward to decrease magnification.

Zoom Region Drag a region on the deformation grid. The region is then magnified to fill the deformation dialog.

Pan Drag in the view to move in any direction.

Scroll bars Drag the horizontal and vertical scroll bars to pan the view in a single direction.

Path Commands

Select a Loft object. > Modify panel > Modifier stack display > Sub-object level > Path > Path Commands

The Path Commands rollout appears only when you are modifying an existing loft object and have selected Path from the Sub-Object list. The Put command allows you to make a copy or instance of the loft path.

Interface

Put group

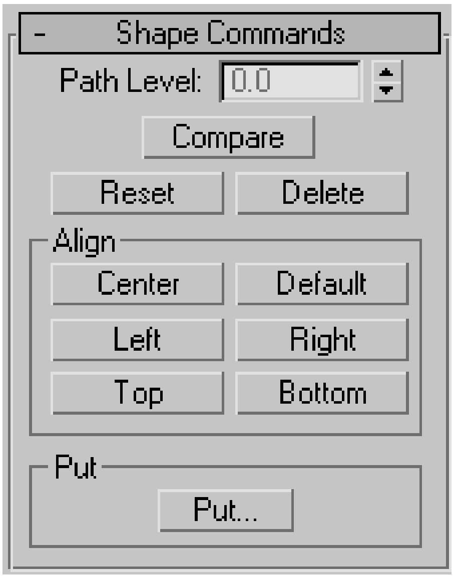
Put Places the path into the scene as a separate object (as a Copy or Instance).

Shape Commands

Select a Loft object. > Modify panel > Modifier stack display > Sub-object level > Shape > Shape Commands rollout

These controls let you align and compare shapes along the loft path.

Interface



Path Level Adjusts the shape's position on the path.

Compare Displays the Compare dialog on page 835 in which you can compare any number of cross-section shapes.

Reset Undoes rotation and scale of the shape performed with the Select and Rotate or Select and Scale.

Delete Deletes the shape from the loft object.

Align group

The six buttons in this group let you align the selected shape in relation to the path. Looking down at a shape from the viewport in which it's created, the orientation is left to right along the X axis, and top to bottom along the Y axis.

You can use a combination of these buttons for placements such as corner alignment. The operations are additive. In other words, you can use both Bottom and Left to place the shape in the lower-left quadrant.

Center Centers the shape on the path, based on the bounding box of the shape.

Default Returns the shape to its position when first placed on the loft path. When you use Get Shape, the shape is placed so that the path goes through its pivot point. This is not always the same as the center of the shape. Therefore, clicking Center is different than clicking Default.

Left Aligns the left edge of the shape to the path.

Right Aligns the right edge of the shape to the path.

Top Aligns the top edge of the shape to the path.

Bottom Aligns the bottom edge of the shape to the path.

Put group

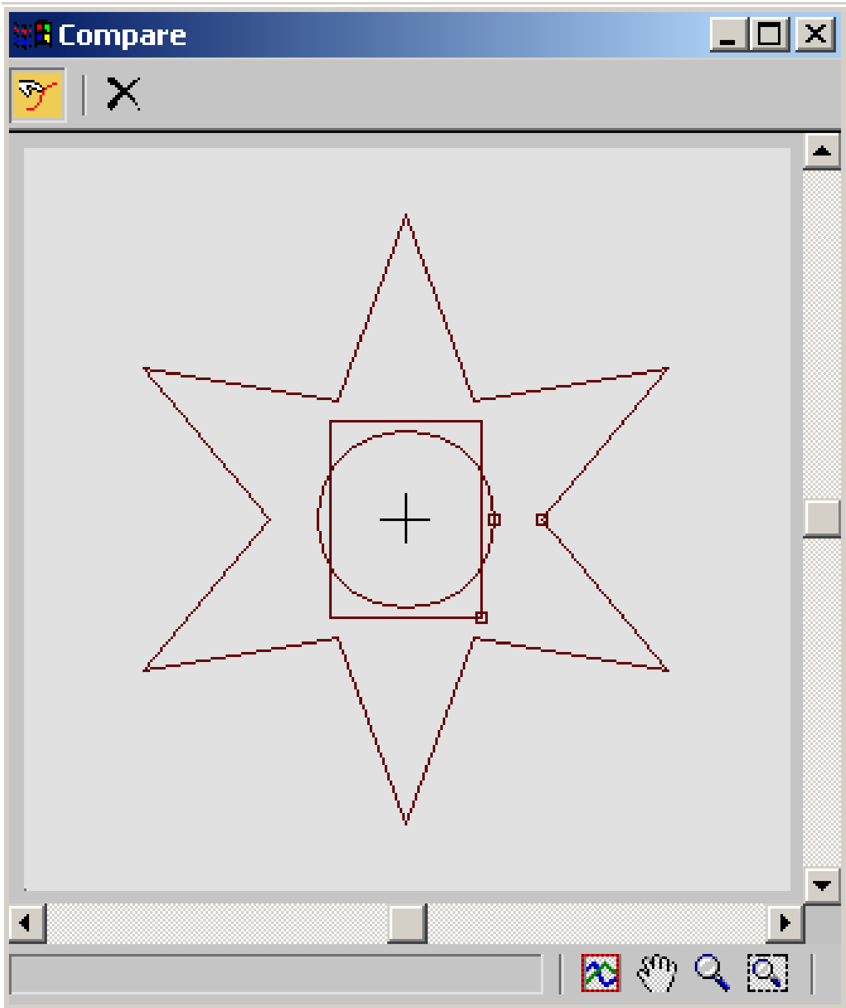
Put Puts the shape into the scene as a separate object.

Compare Dialog

Select a Loft object. > Modify panel > Modifier stack display > Sub-object level > Shape > Shape Commands rollout > Compare button

The Compare dialog lets you compare any number of cross-section shapes in a loft object for purposes of making sure their first vertices are properly aligned. If shapes' first vertices aren't aligned, unexpected lofting results can occur.

Interface



Pick Shape Lets you select shapes to display from the selected loft object. Click the Pick Shape button in the upper-left corner of the dialog. Then, in the viewport, select the shapes to display. Select a shape a second time to remove it from the display.

When you position the mouse cursor over a shape in the loft object, the cursor image changes to show whether the shape appears in the dialog window: a + sign appears if the shape isn't selected (indicating that if you select the shape, it will be added to the dialog window), and a - sign appears if the shape is already selected.

With each shape, the Compare dialog displays the first vertex as a small square. For correct lofting, the first vertices of all shapes on the path need to be in the same position.

Reset Removes all shapes from the display.

Dialog controls

You can scroll the Compare dialog with the scroll bars at the bottom and right sides. You can also use the buttons in the lower-right corner to perform View Extents, Pan, Zoom, and Zoom Region functions.

Align group

While the Compare dialog is open, you can affect the shapes' positions in the dialog window with the Shape Commands rollout > Align group buttons. Turn off Pick Shape, select a shape in the viewport, and then click the Align group buttons. See Shape Commands on page 834 for further information.

Mesher Compound Object

Create panel > Geometry > Compound Objects > Object Type rollout > Mesher

Create menu > Compound > Mesher

The Mesher compound object converts procedural objects to mesh objects on a per-frame basis so that you can apply modifiers such as Bend or UVW Map. It can be used with any type of object, but is designed primarily to work with particle systems on page 3002. Mesher is also useful for low-overhead instancing of objects with complex modifier stacks.

Procedures

To use a Mesher object:

1. Add and set up a particle system.
2. Click the Create panel> Geometry > Compound Objects > Object Type rollout > Mesher button.
3. Drag in a viewport to add the Mesher object. The size doesn't matter, but the orientation should be the same as that of the particle system.

4 Go to the Modify panel, click the Pick Object button, and then select the particle system.

   The Mesher object becomes a clone of the particle system, and shows the particles as mesh objects in the viewports no matter what the particle system's Viewport Display setting is.

5 Apply a modifier to the Mesher object, and set its parameters. For example, you might apply a Bend modifier and set its Angle parameter to 180.

6 Play the animation.

   Depending on the original particle system and its settings, as well as any modifiers applied to the Mesher object, you might be getting unexpected results. This typically occurs because the bounding box for the modifier, as applied to the particle system, is recalculated at each frame. For example, with a bent Super Spray particle system set to spread out over time, as the particles stream away and separate, the bounding box becomes longer and thicker, potentially causing unexpected results. To resolve this, you can use another object to specify a static bounding box.

7 To use another object's bounding box to limit the modified Mesher object, first add and set up the object. Its position, orientation, and size are all used in calculating the bounding box.

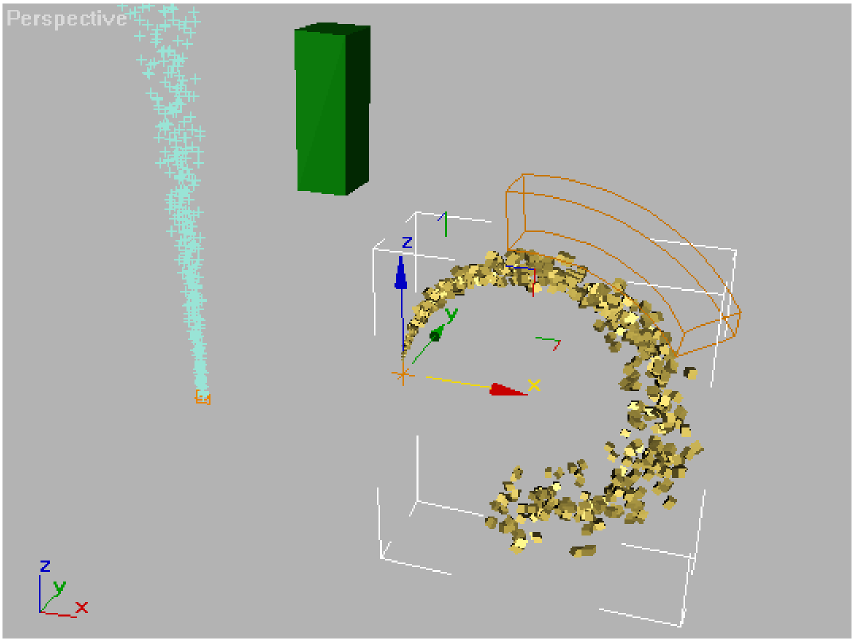
8 Select the Mesher object, and go to the Mesher stack level.

9 In the Parameters rollout, turn on Custom Bounding Box, click the Pick Bounding box button, and then select the bounding box object.

   The particle stream uses the new, static bounding box.

TIP You can use any object as a bounding box, so it is often fastest to use the particle system itself. Move to the frame where the particle system is the size you want and pick it.

In the following illustration, you can see a Super Spray particle system (left) and a Mesher object derived from the Super Spray (right). A Bend modifier is applied to the Mesher. In the center is a box object being used as a custom bounding box. The bounding box applied to the Bend modifier is visible as an orange wireframe when the modifier is highlighted in the stack.



Using a custom bounding box with a bent particle system

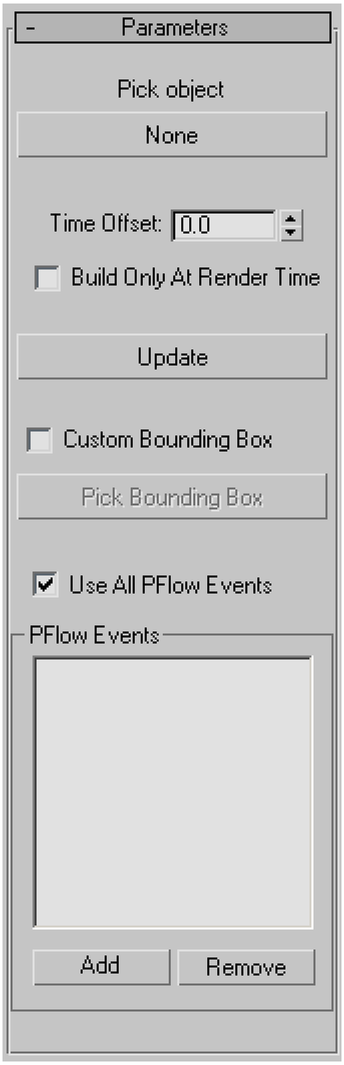
To modify the particles aspect of the Mesher, edit the original particle system.

To modify the custom bounding box, move, rotate, or scale the bounding box object, and then reapply it using the Mesher object.

At this point, both particle systems will render. The original particle system must exist in order to be able to be used by the Mesher object, so if you want only the Mesher replica to render, hide the original system before rendering.

Interface

Parameters rollout



Pick Object Click this button and then select the object to be instanced by the Mesher object. After doing so, the name of the instanced object appears on the button.

Time Offset The number of frames ahead of or behind the original particle system that the Mesher's particle system will run. Default=0.

Build Only At Render Time When on, the Mesher results do not appear in the viewports, but only when you render the scene. Default=off.

Use this option to reduce the amount of computation required for the viewport display.

Update After editing the original particle system settings or changing the Mesher Time Offset setting, click this button to see the changes in the Mesher system.

Custom Bounding Box When on, Mesher replaces the dynamic bounding box derived from the particle system and modifier with a static bounding box of the user's choice.

Pick Bounding Box To specify a custom bounding box object, click this button and then select the object.

The custom bounding box appears as an orange wireframe when the modifier is highlighted in the stack.

TIP You can use any object as a bounding box, so it is often fastest to use the particle system itself. Move to the frame where the particle system is the size you want and pick it.

(coordinate values) Displays the coordinates of the opposite corners of the custom bounding box.

Use All PFlow Events When on, and you've applied Mesher to a Particle Flow on page 2795 system, Mesher automatically creates mesh objects for every event on page 7966 in the system.

To use only certain events, turn this off and specify the events to use with the PFlow Events group controls (see following).

PFlow Events group

When the Mesher object is applied to a Particle Flow system, use these controls to create meshes for specific events in the system. Mesher does not create meshes for the remaining events.

[list box] Displays all Particle Flow events currently affected by Mesher.

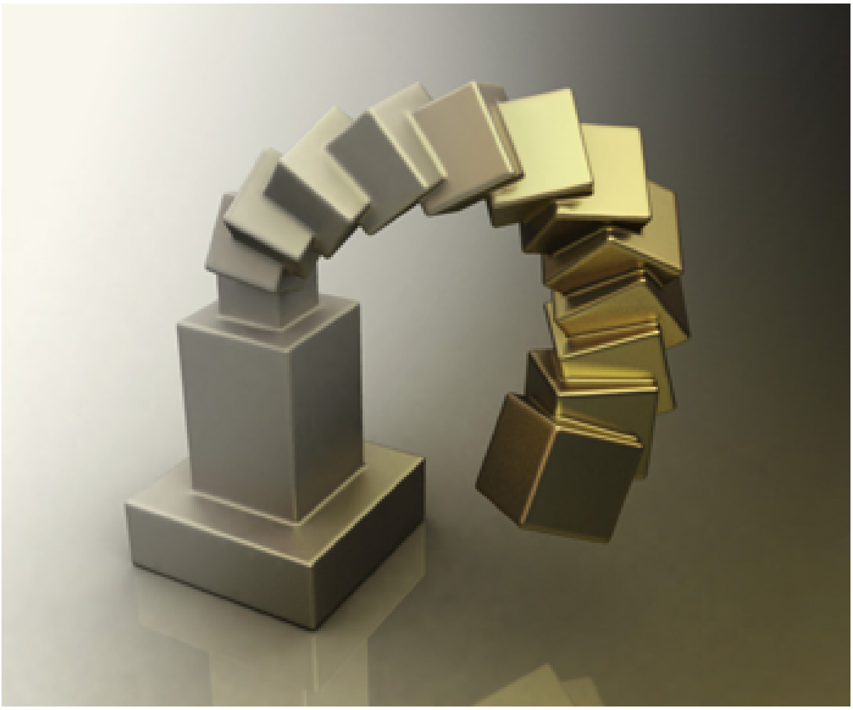
Add Lets you specify Particle Flow events to be affected by Mesher.

If the Mesher object is applied to a Particle Flow system, when you click Add, an Add PF Events dialog opens listing all events in the system. Highlight the events to add, and then click OK. The events now appear in the list.

Remove Deletes highlighted events from the list.

ProBoolean/ProCutter Compound Objects

The ProBoolean and ProCutter compound objects provide you with modeling tools for combining 2D and 3D shapes in ways that would be difficult or impossible otherwise. The ProBoolean compound object on page 844 takes a 3ds Max mesh and adds extra intelligence to it prior to performing Boolean operations. First it combines the topologies, then it determines coplanar triangles and removes incident edges. The Booleans are then performed not on triangles but N-sided polygons. Once the Boolean operations are completed, the result is retriangulated and sent back into 3ds Max with coplanar edges hidden. The result of this extra work is twofold: The reliability of the Boolean object is extremely high, and the resulting output is much cleaner in terms of having fewer small edges and triangles.



Advantages of ProBoolean over the legacy 3ds Max Boolean compound object include:

- **Better quality mesh** - fewer small edges, fewer narrow triangles.
- **Smaller mesh** - fewer vertices and faces.
- **Easier and faster to use** - unlimited objects per Boolean operation.
- **Cleaner-looking mesh** - coplanar edges remain hidden.
- **Integrated decimation and quad meshing**

In addition, ProCutter on page 862 is an excellent tool for exploding, breaking apart, assembling, sectioning, or fitting together objects such as a 3D puzzle. See the following illustration for an example of a goblet shattering.

See also:

- ProBoolean Compound Object on page 844
- ProCutter Compound Object on page 862
- Quad Meshing and Smoothing on page 869

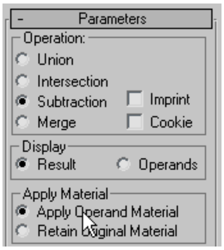
ProBoolean Compound Object

Select an object. > Create panel > Geometry > Compound Objects > Object Type rollout > ProBoolean

A Boolean object combines two or more other objects by performing a Boolean operation or operations on them. ProBoolean adds a range of functionality to the traditional 3ds Max Boolean object, such as the ability to combine multiple objects at once, each using a different Boolean operation. ProBoolean can also automatically subdivide the Boolean result into quadrilateral faces, which lends itself well to smoothing edges with MeshSmooth on page 1532 and TurboSmooth on page 1822.

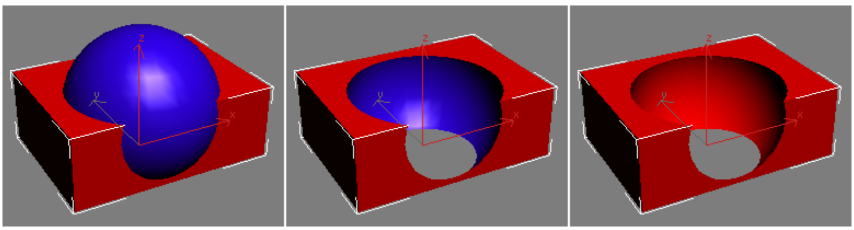
Materials, Textures, Vertex Colors

ProBoolean and ProCutter transfer texture coordinates, vertex colors, optionally materials, and maps from the operands to the final results. You can choose to apply the operand material to the resulting faces, or you can retain the original material. If one of the original operands had material maps or vertex colors, the resulting faces derived from that operand maintain those graphical attributes. However, when texture coordinates or vertex colors are present, it is impossible to remove coplanar faces, so the resulting mesh quality will be lower. We suggest that you apply textures after the ProBoolean operations.



ProBoolean provides two options for applying materials. You can see these two options in the Apply Material group on the Parameters panel (see above illustration). The default choice is Apply Operand Material. This option applies the operand material to the resulting faces. Choosing Retain Original Material causes the resulting faces to use the material of the first selected object in the Boolean operation.

You can see the difference in the following illustration. The Boolean operation starts with a red box and a blue sphere. The box is used as the base object and the sphere is the subtracted operand. Using the default Apply Operand Material option gives the result shown in the center of the illustration. Choosing Retain Original Material yields the result shown on the right side of the illustration.



Supported Boolean Operations

ProBoolean supports Union, Intersection, Subtraction, and Merge. The first three operations work similarly to their counterparts in the standard Boolean compound object. The Merge operation intersects and combines two meshes without removing any of the original polygons. This can be useful for cases in which you need to selectively remove parts of the mesh.

Also supported are two variants of the Boolean operations: Cookie Cutter and Imprint. Cookie Cutter performs the specified Boolean operation but does not add the faces from the operands into the original mesh. It can be used to cut a hole in a mesh or to get the portion of a mesh inside of another object. The Imprint option inserts (imprints) the intersection edges between the operands and the original mesh without removing or adding faces. Imprint only splits faces and adds new edges to the mesh of the base object (original selected object).

Editing the Boolean Object

When you access a ProBoolean or ProCutter object from the Modify panel, you can add operands to the existing set. You can also remove and transform (move, rotate, etc.) operands.

Polygon Reduction

ProBoolean and ProCutter have a built-in decimation function. Typically, decimation is of better quality if it is integrated with the Boolean operations. The reason for this is that the Boolean object contains meta-information about which edges are intersection edges. The decimation function takes this information into account and uses it to properly maintain intersection edges.

Text, Lofts and NURBS

When performing Boolean operations with text objects on page 638, make sure characters don't intersect each other and that each letter is closed. Also, it's easy to inadvertently create loft objects on page 786 and NURBS objects on page 2237 in such a way as to have self-intersections. With loft objects, check the ends and points where the loft curve bends.

See also:

- ProCutter Compound Object on page 862

Procedures

To create a ProBoolean compound object:

1. Set up objects for the Boolean operation. For example, to subtract spherical shapes from a box, create the box and spheres and arrange the spheres so that their volumes intersect the box where the subtractions should take place.
2. Select the base object. In the example in step 1, you would select the box.

3 On the Create panel > Geometry section, choose Compound Objects from the drop-down list, and then click ProBoolean.

4 On the Parameters rollout, choose the type of Boolean operation you want to use: Union, Intersection, Subtraction, etc. Also choose how the software will transfer the next operand you pick into the Boolean object: Reference, Copy, Move, or Instance. You can also choose to retain the original material, or keep the default Apply Material choice: Apply Operand Material.

5 Click the Start Picking button.

6 Pick one or more objects to participate in the Boolean operation.

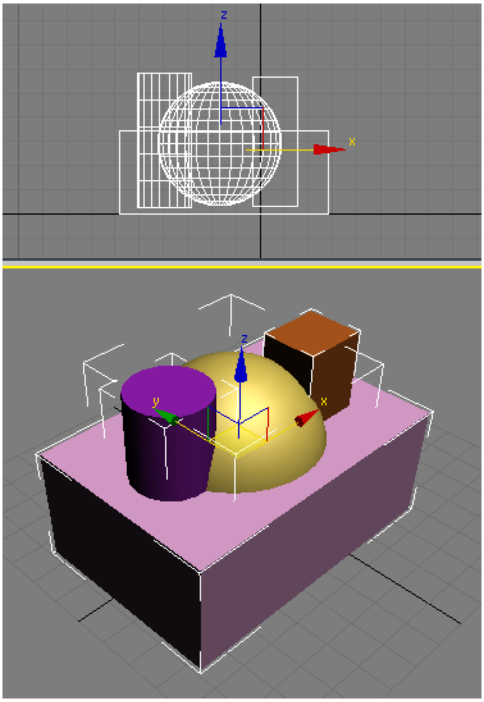
7 As you pick objects, you can also change, for each newly picked object, the Boolean operation (Merge, etc.) and options (Cookie or Imprint), as well as how the next operand is transferred to the Boolean (Reference, Copy, etc.) and the Apply Material choice. You can continue picking operands as long as the Start Picking button stays pressed in. Each of the objects you pick is added to the Boolean operation.

   When the Modify panel is active, you can add objects to a selected ProBoolean object by clicking the Start Picking button and then picking the objects to add.

Example: To change an existing Boolean with sub-object operations:

ProBoolean offers a great deal of flexibility in combining various Boolean operations simultaneously, plus the ability to change the way operands combine both as you build the Boolean object and after the fact.

1 Start by adding a box, a sphere, a smaller box, and a cylinder, as shown in the following illustration.



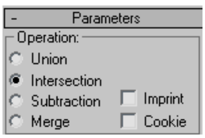
Top: Front viewport

Bottom: Perspective viewport

2 Select the box.

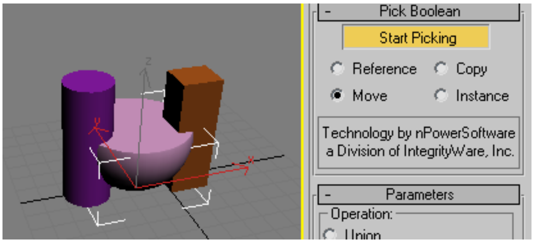
3 On the Create panel > Geometry section, choose Compound Objects from the drop-down list, and then click ProBoolean.

4 On the Parameters rollout, in the Operation group, choose Intersection.



5 On the Pick Boolean rollout, click Start Picking, and then click the sphere.

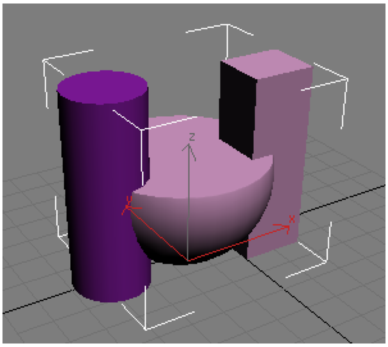
The result is the intersection of the sphere and the box; that is, a single object that represents the *common* volume both objects occupy. In this case, it's the overlap of the sphere and the box. Although neither has a material at this point, the result uses the default color originally assigned by the software, at random, to the box when it was created.



Note that Start Picking stays active (yellow). This means you can continue picking objects to incorporate into the Boolean object, optionally changing the operation as you go.

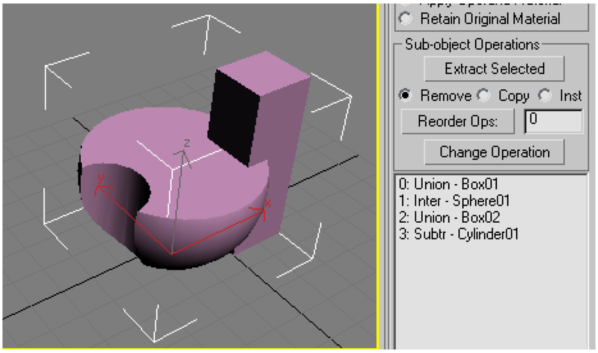
6 Set Operation to Union, and then click the small box.

The result is the union (adding) of the small box with the intersection of the sphere and larger box. Again, the original object's color is assigned to the result.



7 Set Operation to Subtraction, and then click the cylinder.

The cylinder's intersecting volume is subtracted from the previous Boolean result.



Note that the entire history of operands and operations used to build the Boolean object is listed in the hierarchy view list at the bottom of the Parameters rollout. Box01 starts the Boolean with Union, Sphere01 is then incorporated with Intersection, Box02 is incorporated with Union, and finally Cylinder01 is incorporated with Subtraction.

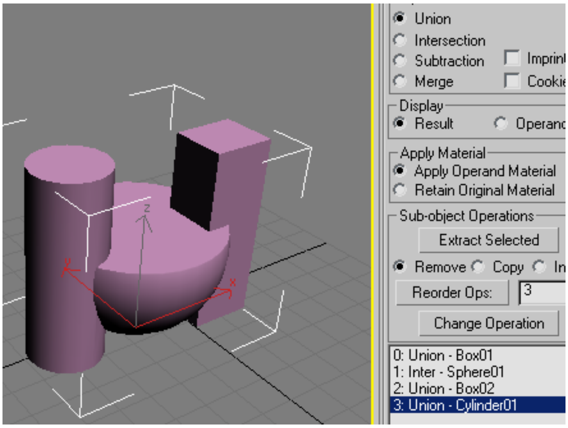
NOTE The operation for the first object in the list has no effect on the Boolean object, but if you move it to another position in the list it does. You'll see an example of this at the end of this exercise.

You can use the list and the other controls in the Sub-object Operations group to change the results.

8 In the list, highlight the Subtr - Cylinder01 entry, and then set Operation to Union.

9 In the Sub-object Operations group, click the Change Operation button.

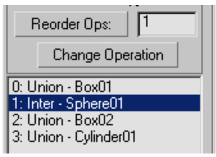
As a result of the change of operation, the cylinder appears in the Boolean object as an additive volume instead of a subtractive one. Also, its entry in the list changes to “Union - Cylinder01”, showing that the Boolean operation for the cylinder is now Union.



You can also change the order of the operations, which can affect the results.

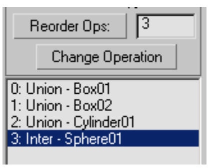
10 In the list, click the Union - Cylinder01 entry to remove the highlighting, and then highlight the entry 1: Inter - Sphere01.

Note that its position in the list, 1, appears in the editable field next to the Reorder Ops button.

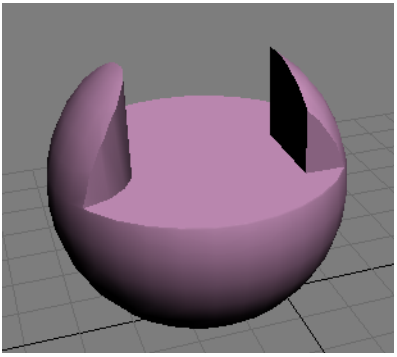


11 Change the value in the field from 1 to **3**, and then click the Reorder Ops button.

The Inter - Sphere01 item jumps to the end of the list.



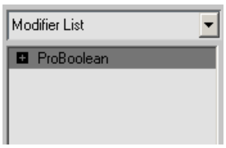
The Boolean object changes significantly. The new order in the list tells you how this shape was achieved: The two boxes and the cylinder were all combined with Union, adding their volumes together, and then the sphere was incorporated into that result with Intersection, leaving only the volume shared by all four objects.



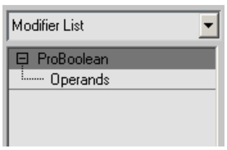
Interface

Modifier stack

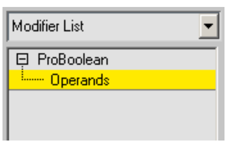
With an unmodified ProBoolean object selected, the modifier stack shows a single, expandable entry: ProBoolean.



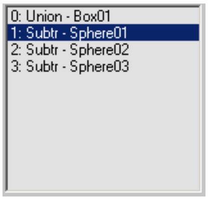
Expanding this entry (by clicking the + icon) reveals a single subsidiary branch: Operands.



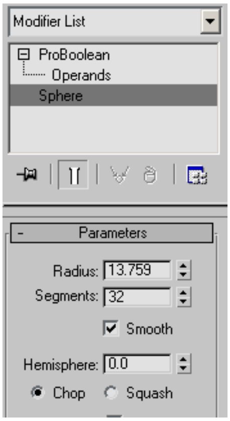
To transform operands in the ProBoolean object independent of the entire object, click the Operands branch to highlight it.



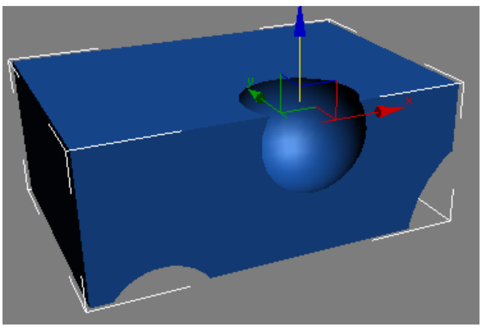
You can then select one or more operands, either by using standard selection methods in the viewport, or by highlighting their names in the hierarchy view on page 859 list at the bottom of the Parameters rollout.



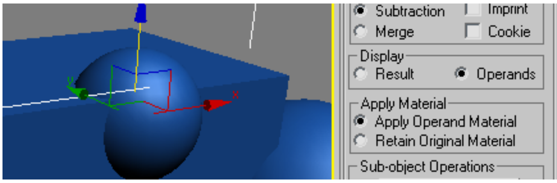
When one, and only one, operand is selected, the object type (not its name) appears as a separate stack entry below the ProBoolean entry. Clicking this entry provides direct access to the operand's parameters on the Modify panel.



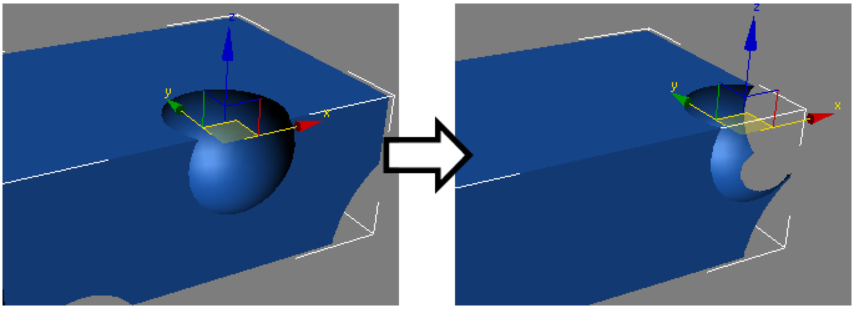
If Parameters rollout > Display is set to Result, selecting an operand displays the operand's axis tripod or transform gizmo in the viewport, although the operand itself is not visible by default.



To view the operand, set Parameters rollout > Display to Operands.

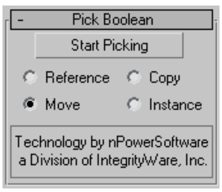


Whether or not the operands are visible, you can transform and animate them at the Operands sub-object level, as with any other object in 3ds Max.



You can also transform and animate the base object; that is, the first object in the hierarchy view list.

Pick Boolean rollout



Start Picking Click this and then click each operand to transfer to the Boolean object in turn. Before picking each operand, you can change the Reference/Copy/Move/Instance choice, the Operation options, and the Apply Material choice.

TIP When you're adding many operands to a Boolean object using the default settings, calculating the result each time you pick an object can slow down the process. To maintain optimum feedback, set Parameters rollout > Display to Operands. Then, when you're finished, set it back to Result. Alternatively, on the Advanced Options rollout set Update to Manually, and then click the Update button to view the results after performing the Boolean operations.

Choose a radio button to specify how the next operand you pick is transferred to the Boolean object:

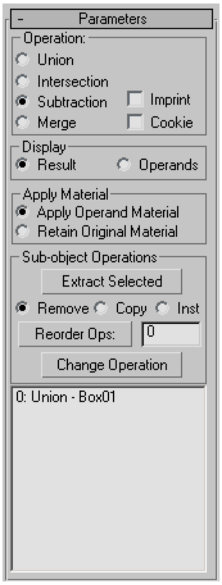
- **Reference** The Boolean operation uses a reference on page 8106 to the picked operand, so the object remains after being incorporated into the Boolean object. Future modifications to the originally picked object will also modify the Boolean operation. Use Reference to synchronize modifier-induced changes to the original operand with the new operand, but not vice-versa.

- **Copy** The Boolean operation uses a copy of the picked operand. The selected object is unaffected by the Boolean operation, but a copy of it participates in the Boolean operation.

- **Move** The picked operand becomes part of the Boolean operation and is no longer available as a separate object in the scene. This is the default choice.

- **Instance** The Boolean operation makes an instance on page 8014 of the selected object. Future modifications of the selected object will also modify the instanced object participating in the Boolean operation and vice-versa.

Parameters rollout



Operation group

These settings determine how the Boolean operands interact physically.

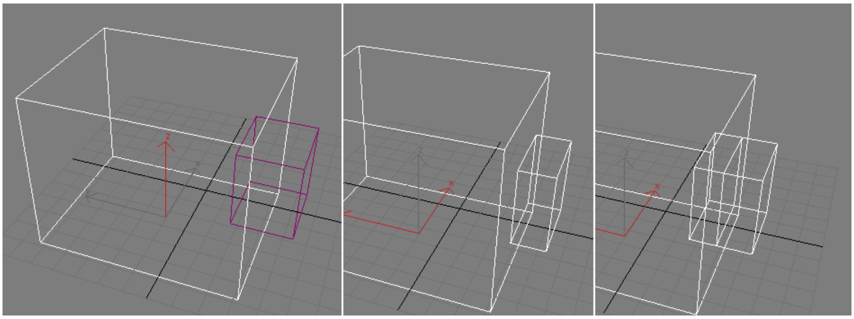
Union Combines two or more separate entities into a single Boolean object.

Intersection Creates a "new" object from the physical intersection between the original objects; the non-intersecting volumes are removed.

Subtraction Removes the volume of a selected object from the original object.

Merge Combines objects into a single object without removing any geometry. New edges are created where the objects intersect.

NOTE In the following illustration, the display property Backface Cull was turned off so that all edges are visible.

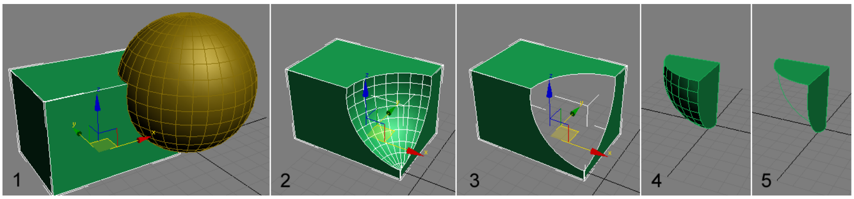


Left: Original object (box) and operand (small box)

Center: Union operation; part of the smaller box is removed.

Right: Merge operation, showing new edges at intersection

Cookie Cuts the faces of the original mesh shape, affecting only those faces. The faces of the selected operand are not added to the Boolean result.



1. Original object (box) and operand (sphere)

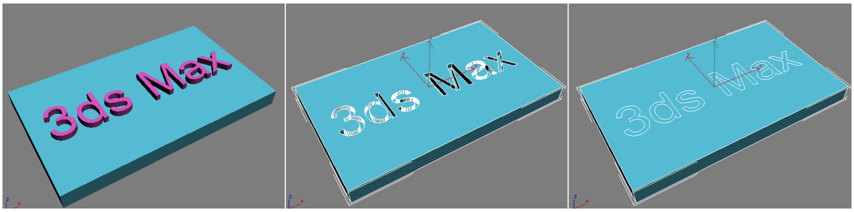
2. Standard Subtraction operation

3. Subtraction with Cookie on

4. Standard Intersection operation

5. Intersection with Cookie on

Imprint Prints the outline of the shape (or intersection edges) onto the original mesh object.



Left: Original object (box) and operand (text)

Center: Standard Subtraction operation

Right: Subtraction with Imprint

NOTE The result of the Imprint operation is always the same; the main Operation choice has no effect on it.

Display Choose one of the following display modes:

- **Result** Displays only the result of the Boolean operations, not the individual operands.
  Choosing Result also activates the ProBoolean level in the modifier stack on page 852.

- **Operands** Displays the operands that define the Boolean result. Use this mode to edit the operands and modify the result.
  Choosing Operands also activates the Operands level in the modifier stack on page 852.

  Also, when picking many operands, use this mode to avoid having to recalculate the result each time, and then set Display back to Result at the end.

Apply Material Choose one of the following material application modes:

- **Apply Operand Material** New faces created by the Boolean operation acquire the material of the operand.

- **Retain Original Material** New faces created by the Boolean operation retain the material of the original object.

Sub-object Operations group

These functions operate on operands highlighted in the hierarchy view list (see following).

NOTE For these operations, you need not be at the Operands sub-object level in the modifier stack.

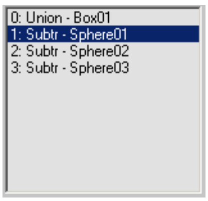
Extract Selected Based on the chosen radio button (Remove, Copy, or Inst; see following), Extract Selected applies the operation to the highlighted operand in the hierarchy view list. Three modes of extraction are available:

- **Remove** Removes the operand or operands highlighted in the hierarchy view list from the Boolean result. It essentially undoes the addition of the highlighted operand(s) to the Boolean object. Each extracted operand becomes a top-level object again.
- **Copy** Extracts a copy of the operand or operands highlighted in the hierarchy view list. The original operand remains part of the Boolean result.
- **Inst** Extracts an instance of the operand or operands highlighted in the hierarchy view list. Subsequent modifications to this extracted operand also modify the original operand, thus affecting the Boolean object.

Reorder Ops Changes the ordering of the highlighted operand in the hierarchy view list. The reordered operand is moved to the position listed in the text field next to the Reorder Ops button.

Change Operation Changes the type of operation (see Operation group on page 856) for the highlighted operand. To change the operation type, highlight the operand in the hierarchy view, then choose the operation type radio option, and then click Change Operation.

Hierachy View

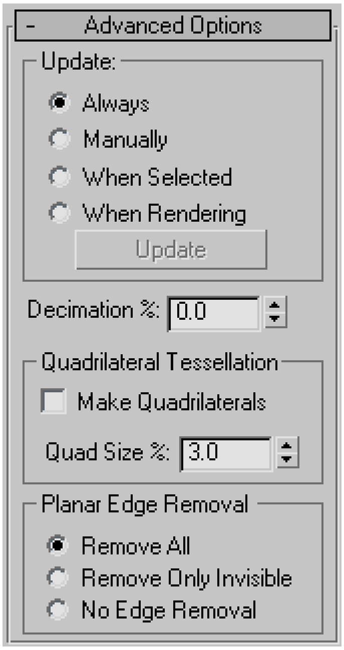


The hierarchy view, found at the bottom of the Parameters rollout, displays a list of all of the Boolean operations that define the selected mesh. Each time you perform a new Boolean operation, the software adds an entry to the list.

You can highlight operands for modification by clicking them in the hierarchy view list. To highlight multiple contiguous items in the list, click the first, and then Shift+click the last. To highlight multiple non-contiguous entries, use Ctrl+click. To remove highlighting from a list entry, Alt+click the highlighted item.

At the ProBoolean level in the modifier stack, you can perform only sub-object operations on page 858 on highlighted items. At the Operands sub-object level, you can transform highlighted operands as well as perform sub-object operations; see Modifier stack on page 852 for details.

Advanced Options rollout



Update group

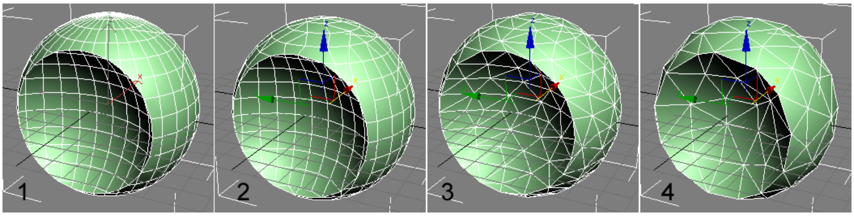
These options determine when updates are performed on the Boolean object after you make changes. Choose one of the following:

- **Always** Updates occur as soon as you make changes to the Boolean object.
- **Manually** Updates occur only when you click the Update button.
- **When Selected** Updates occur whenever the Boolean object is selected.
- **When Rendering** Updates are applied to the Boolean object only at render time, or when you click Update.

Update Applies changes to the Boolean object. Available with all options except Always.

NOTE When you first create a ProBoolean object with Manually or When Rendering active, no operands, including the base object, are visible until you update at least once. Thereafter, the base object is visible, but no subsequently picked operands are until you update again.

Decimation % The percentage of edges to remove from the polygons in the Boolean object, thus reducing the number of polygons. For example, a Decimation % setting of 20.0 removes 20 percent of the polygon edges.



1. Decimation %=0.0

2. Decimation %=30.0

3. Decimation %=60.0

4. Decimation %=80.0

Quadrilateral Tessellation group

These options enable quadrilateral tessellation of the Boolean object. This makes the object suitable for editing subdivision surfaces on page 2008 and for smoothing meshes. It also makes the object suitable for conversion to Editable Poly format.

For further discussion of this option, see the topic Quad Meshing and Smoothing on page 869.

Make Quadrilaterals When on, changes the tessellation of the Boolean object from triangles to quadrilaterals.

NOTE When Make Quadrilaterals is on, the Decimation setting has no effect.

Quad Size % Determines the size of the quadrilaterals as a percentage of the overall Boolean object length.

Planar Edge Removal group

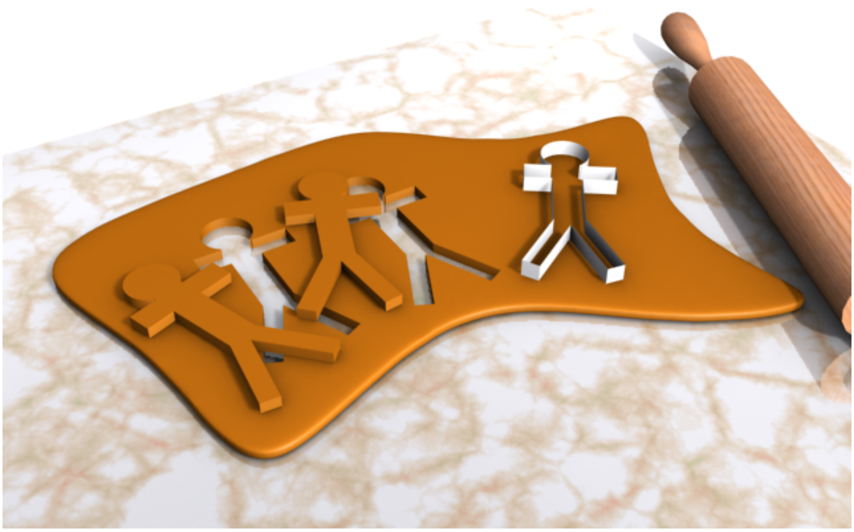
This option determines how the polygons on planar faces are handled. Choose one of the following:

- **Remove All** Removes all extra coplanar edges on a face such that the face itself will define the polygon.
- **Remove Only Invisible** Removes invisible edges on each face.
- **No Edge Removal** No edges are removed.

ProCutter Compound Object

Select an object. > Create panel > Geometry > Compound Objects > Object Type rollout > ProCutter

The ProCutter Compound object lets you perform specialized Boolean operations, primarily for the purpose of breaking apart or subdividing volumes. The results of ProCutter operations are particularly suitable for use in dynamics simulations where an object explodes or is shattered by impact with a force or another object.



Following is a list of ProCutter features:

- Break apart a stock object into elements of an editable mesh or into separate objects using cutters that are either solids or surfaces.
- Use one or more cutters on one or more stock objects at the same time.
- Perform a volume decomposition of a set of cutter objects.
- Use a single cutter many times without maintaining the history.

See also:

- ProBoolean Compound Object on page 844

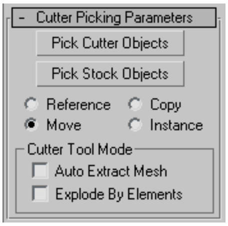
Procedures

To use ProCutter:

1. Select an object to use as a cutter.
2. Activate the ProCutter compound object.
3. On the Cutter Picking Parameters rollout, click Pick Cutter Objects, and then select additional cutters.
4. On the Cutter Picking Parameters rollout, click Pick Stock Objects, and then select objects to be cut by the cutter objects.
5. In the Cutter Parameters rollout > Cutter Options group, choose the parts of the originals you wish to keep: Stock Outside Cutters, Stock Inside Cutters, Cutters Outside Stock.
6. To get separate objects to manipulate or animate, collapse the result to an Editable Mesh on page 2075 object and use the Explode tool set to 180.0. Alternatively, use Auto Extract Mesh and Explode By Elements, described below.

Interface

Cutter Picking Parameters rollout



Pick Cutter Objects When on, objects you select are designated as cutters, used to subdivide stock objects.

Pick Stock Objects When on, objects you select are designated as stock objects; that is, objects that are subdivided by cutters.

Choose a radio button to specify how the next object you pick is transferred to the ProCutter object:

- **Reference** The Boolean operation uses a reference on page 8106 to the picked operand, so the object remains after being incorporated into the Boolean object. Future modifications to the originally picked object will also modify the Boolean operation. Use Reference to synchronize modifier-induced changes to the original operand with the new operand, but not vice-versa.
- **Copy** The Boolean operation uses a copy of the picked operand. The selected object is unaffected by the Boolean operation, but a copy of it participates in the Boolean operation.
- **Move** The picked operand becomes part of the Boolean operation and is no longer available as a separate object in the scene. This is the default choice.
- **Instance** The Boolean operation makes an instance on page 8014 of the selected object. Future modifications of the selected object will also modify the instanced object participating in the Boolean operation and vice-versa.

Cutter Tool Mode group

These options let you use the cutter as a sculpting tool, cutting the same object repeatedly in different places. You can also get separate objects without having to go through Editable Mesh conversion.

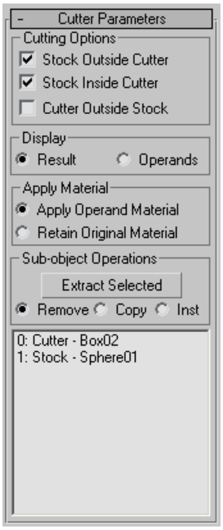
Auto Extract Mode Automatically extracts the result when you select a stock object. It does not maintain the stock as a sub-object, but edits it and replaces the object with the result of the cut. This lets you quickly cut, move the cutter, and cut again.

Explode By Elements When Auto Extract is on, detaches each element into a separate object automatically. Has no effect when Auto Extract is off.

This convenient option makes it unnecessary to convert the ProCutter object to Editable Mesh format and then use Explode, as mentioned in this procedure on page ?. This is useful when cutting up an object. For example, you could use it to slice a loaf of bread. You use the cutter to cut a piece off, move the cutter, and then cut again.

Parameters rollout

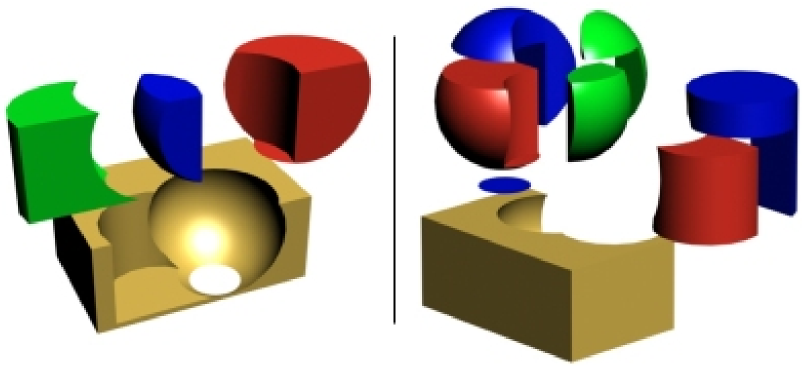
You can choose any combination of the three cutting options to get the desired result. If you have non-closed meshes, the orientation of the mesh might determine which part of the stock is considered to be inside or outside the cutter.



Stock Outside Cutter The result contains the parts of the stock outside of all of the cutters. This option gives you a similar result to a Boolean subtraction of the cutters from the stock objects. The gold part of the object in the following illustration results from this option.

Stock Inside Cutter The result contains the parts of the stock inside one or more cutters. This option gives you similar results to a Boolean intersection of the cutters and the stock objects. There is some difference because each cutter is treated individually. The green, blue and red parts of the object on the left side of the following illustration are the results of this option.

Cutters Outside Stock The result contains the parts of the cutters that are not inside the stock objects. Note that the cutters will cut each other if they intersect also. You can see in the following illustration. The parts on the right side of the following illustration that are not present on the left side are results of this option.



Cylinder and sphere as cutters and box as stock

Left: Keeping stock inside and outside cutters

Right: Keeping stock inside/outside cutters and cutters outside stock

Display Choose one of the following display modes:

- **Show Result** Displays the result of the Boolean operations.
- **Show Ops** Displays the operands that define the Boolean result. Use this mode to edit the operands and modify the result.

Apply Material Choose one of the following material application modes:

- **Apply Operand Material** New faces created by the Boolean operation acquire the material of the operand.
- **Retain Original Material** New faces created by the Boolean operation retain the material of the original object.

Sub-object Operations group

These functions operate on operands highlighted in the hierarchy view list (see following).

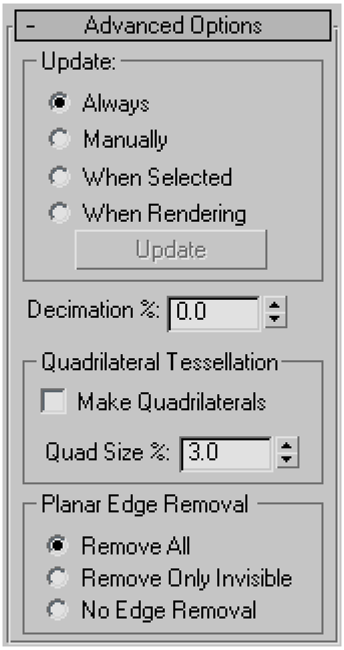
Extract Selected Based on the chosen radio button (Remove, Copy, or Inst; see following), Extract Selected applies the operation to the highlighted operand in the hierarchy view list. Three modes of extraction are available:

- **Remove** Removes the operand or operands highlighted in the hierarchy view list from the Boolean result. It essentially undoes the addition of the highlighted operand(s) to the Boolean object. Each extracted operand becomes a top-level object again.
- **Copy** Extracts a copy of the operand highlighted in the hierarchy view list. The original operand remains part of the Boolean.
- **Inst** Extracts an instance of the operand highlighted in the hierarchy view list. Subsequent modifications to this extracted operand also modify the original operand, thus the Boolean object.

Hierachy View

The hierarchy view displays a list of all operands: cutters and stock objects. You can select and edit objects using the hierarchy view, as with ProBoolean.

Advanced Options rollout



Update group

These options determine when updates are performed on the Boolean object after you make changes. Choose one of the following:

- **Always** Updates occur as soon as you make changes to the Boolean object.
- **Manually** Updates occur only when you click the Update button.
- **When Selected** Updates occur whenever the Boolean object is selected.
- **When Rendering** Updates are applied to the Boolean object only at render time, or when you click Update.

Update Applies changes to the Boolean object. Available with all options except Always.

NOTE When you first create a ProCutter object with Manually or When Rendering active, no operands, including the base object, are visible until you update at least once. Thereafter, the base object is visible, but no subsequently picked operands are until you update again.

Decimation % The percentage of edges to remove from the polygons in the Boolean object, thus reducing the number of polygons. For example, a Decimation % setting of 20.0 removes 20 percent of the polygon edges.

Quadrilateral Tessellation group

Make Quadrilaterals When on, changes the tessellation of the Boolean object from triangles to quadrilaterals. This makes the object suitable for editing subdivision surfaces on page 2008 and for smoothing meshes. It also makes the object suitable for conversion to Editable Poly format.

Quad Size % Determines the size of the quadrilaterals as a percentage of the overall Boolean object length.

Planar Edge Removal group

This option determines how the polygons on planar faces are handled. Choose one of the following:

- **Remove All** Removes all extra coplanar edges on a face such that the face itself will define the polygon.
- **Remove Only Invisible** Removes invisible edges on each face.
- **No Edge Removal** No edges are removed.

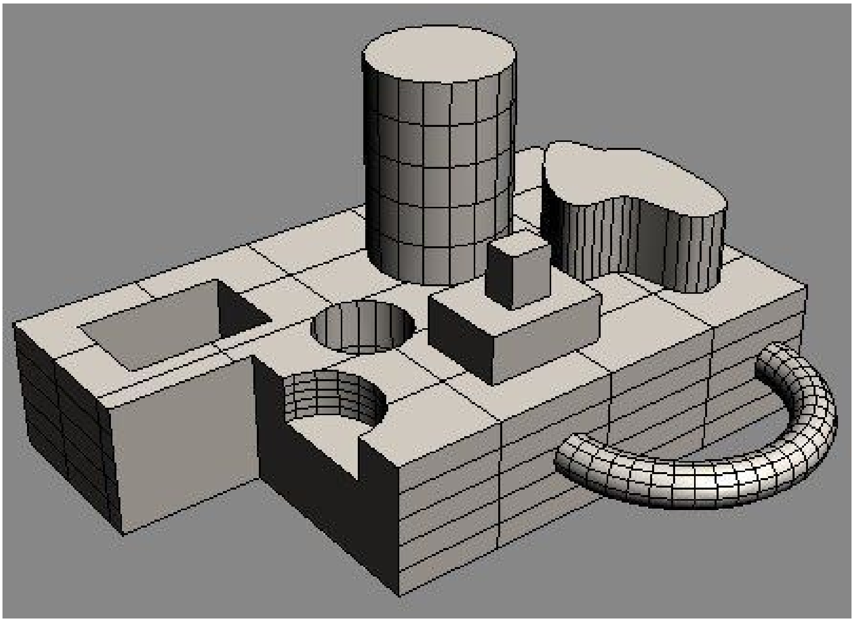
Quad Meshing and Smoothing

ProBoolean and ProCutter can re-mesh planar surfaces using a quadrilateral meshing algorithm. This capability, in combination with the subdivision surface tools in MeshSmooth, TurboSmooth, and Editable Poly, can produce dramatic results.

It does require some level of expertise to understand what is possible and how to achieve the best results using the quadrilateral tessellation. This topic shows the basics of how to use quad meshing and also contains tips and tricks about what works and how it works.

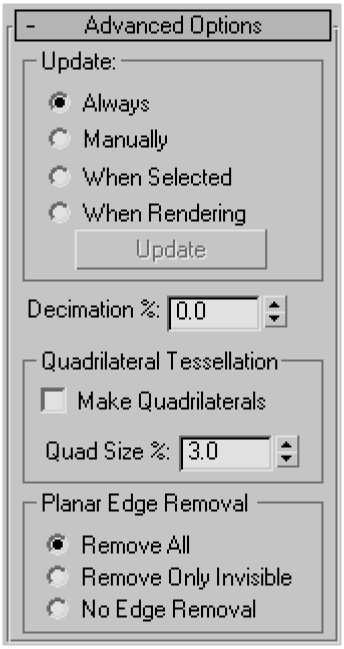
TIP You can apply quadrilateral tessellation to a mesh object without necessarily performing an actual Boolean operation on it. First create an additional object that's not touching the object you want to tessellate. Select the object to tessellate, apply ProBoolean, turn on Make Quadrilaterals, and then subtract the other object. You can then collapse the tessellated object to Editable Mesh/Poly if you like.

Quad Meshing Basics

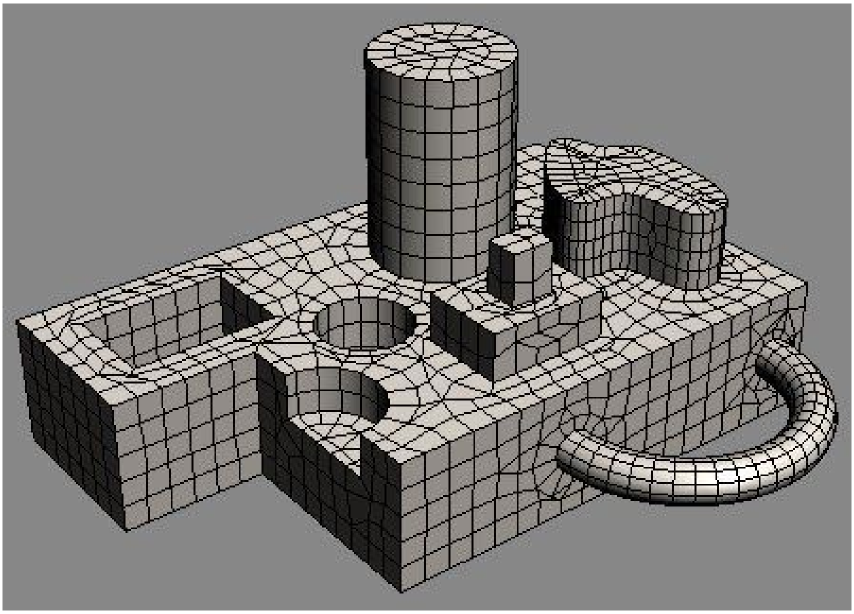


ProBoolean of an object containing several primitives

To make a quadrilateral mesh, select a ProBoolean or ProCutter object, go to the Modify panel, and expand the Advanced Options panel as shown following.

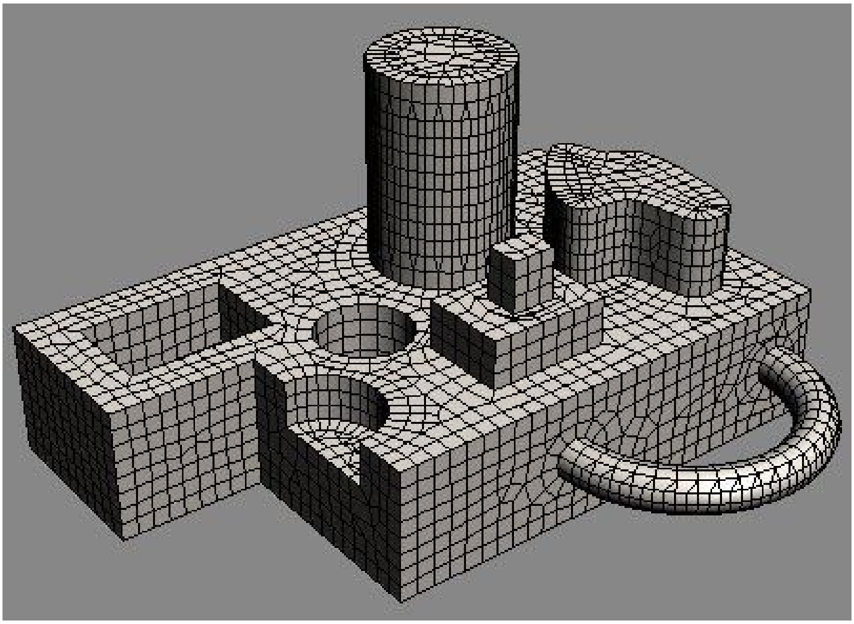


To get a result similar to the following illustration, turn on Make Quadrilaterals check box:



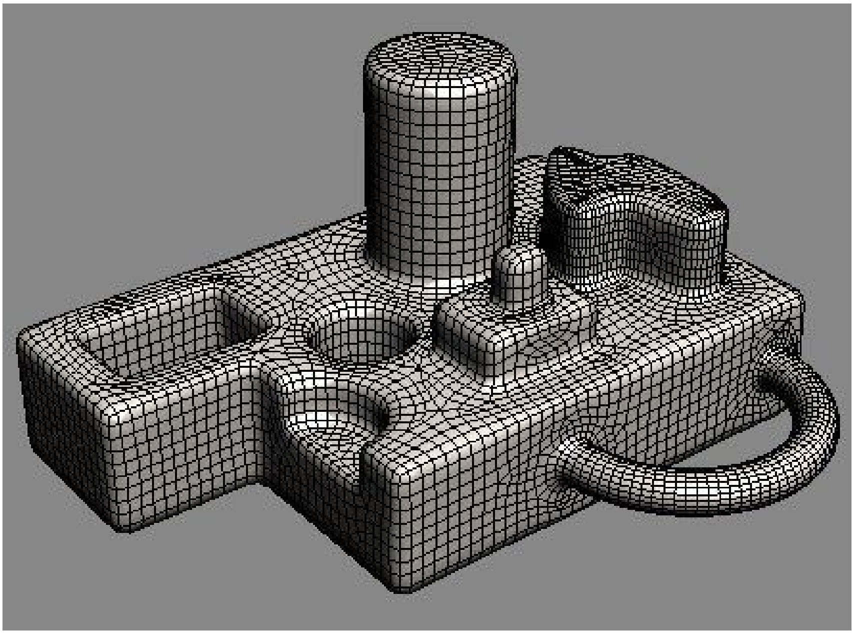
Result of a quad mesh with Quad Size %=3.0

To change the size of the individual quadrilaterals, adjust the Quad Size % parameter. Typically a value between 1 and 4 percent achieves the desired results. The smaller the quad size, the smaller the resulting fillets or blends between the surfaces when the mesh is smoothed. The default Quad Size value is 3.0 percent. A Quad Size value of 2.0 percent produces the following result:

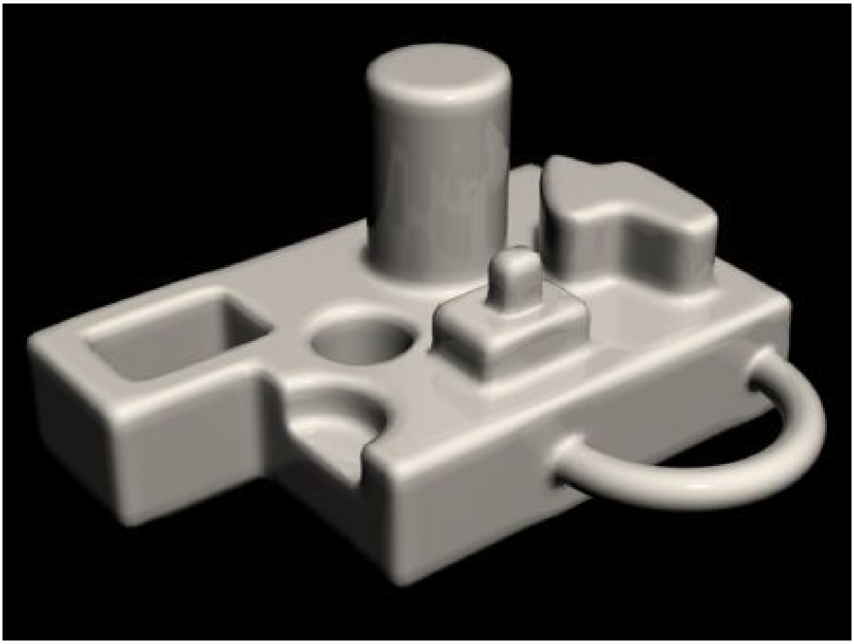


Quad meshing with Quad Size %=2.0

If you know that you have the desired result and don't plan to go back and change the quad size or the original primitives, you can convert the object to Editable Poly format and apply smoothing with the Subdivision Surface settings. If this is not the case, however, and you plan to make further adjustments, use the MeshSmooth on page 1532 or TurboSmooth on page 1822 modifier to retain the history of the ProBoolean object. The following illustration shows the result of a MeshSmooth modifier with Subdivision Amount > Iterations=1 applied to a ProBoolean object with Quad Size % set to 3.0.



MeshSmooth modifier with NURMS and Iterations=1



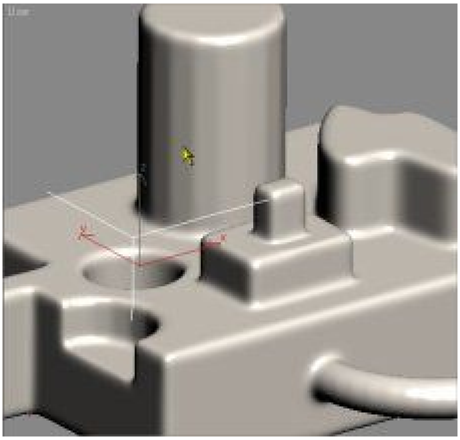
Rendered image after MeshSmooth modifier applied

Quad Meshing Tips and Tricks

Sometimes the results of quad meshing can produce undesirable results in the smoothed model.

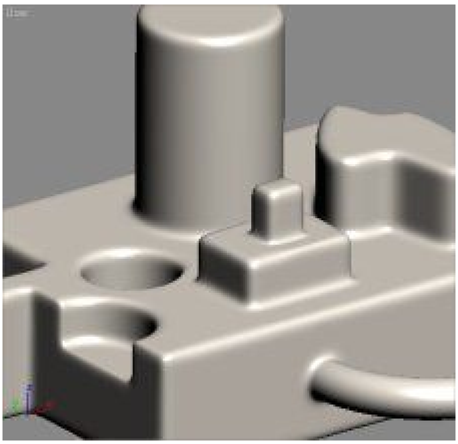
Problem #1: Stripes along cylinders or bumpiness on other surface

Solution: Increase the number of subdivisions around cylinders or along other surfaces.



Stripes and bumps with Quad Size %=2.0

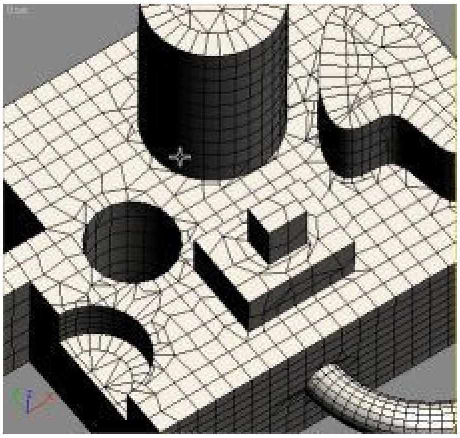
To fix the problem depicted above, the number of sides on the two cylinders was changed from 18 to 30 and the number of segments on the torus was changed from 24 to 36. The following illustration shows the improved result:



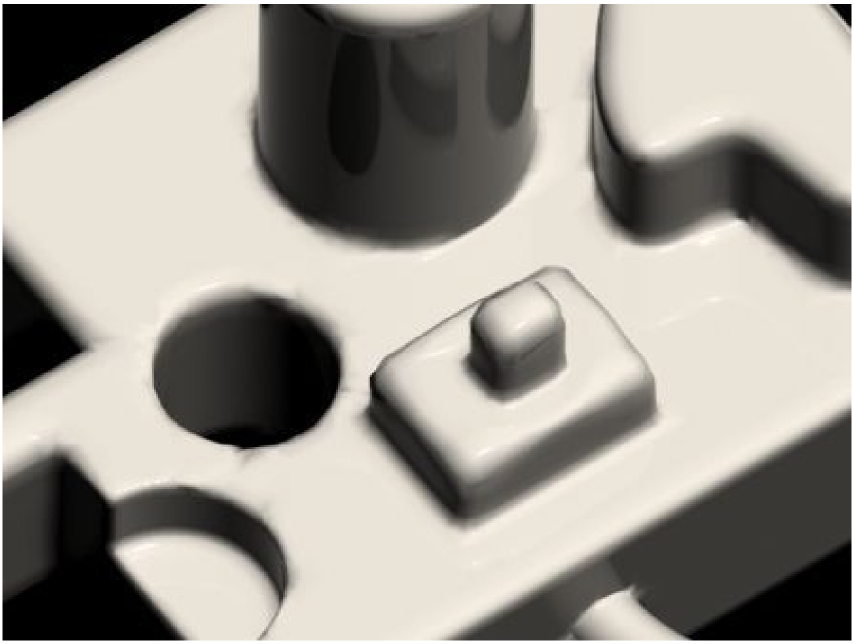
Increasing the primitives' resolution fixes the problem.

Problem #2: Triangles along boundaries caused by conflicts with visible edges from original mesh

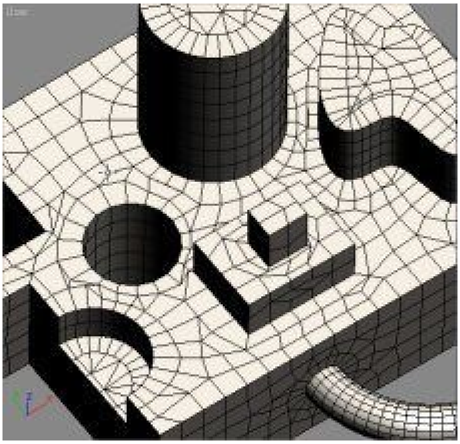
Solution: Make sure Advanced Options rollout > Planar Edge Removal is set to Remove All or create the original primitives without subdivisions on coplanar faces.



Coplanar edges not removed from original box



When rendered, misshapen geometry results from the presence of coplanar edges.

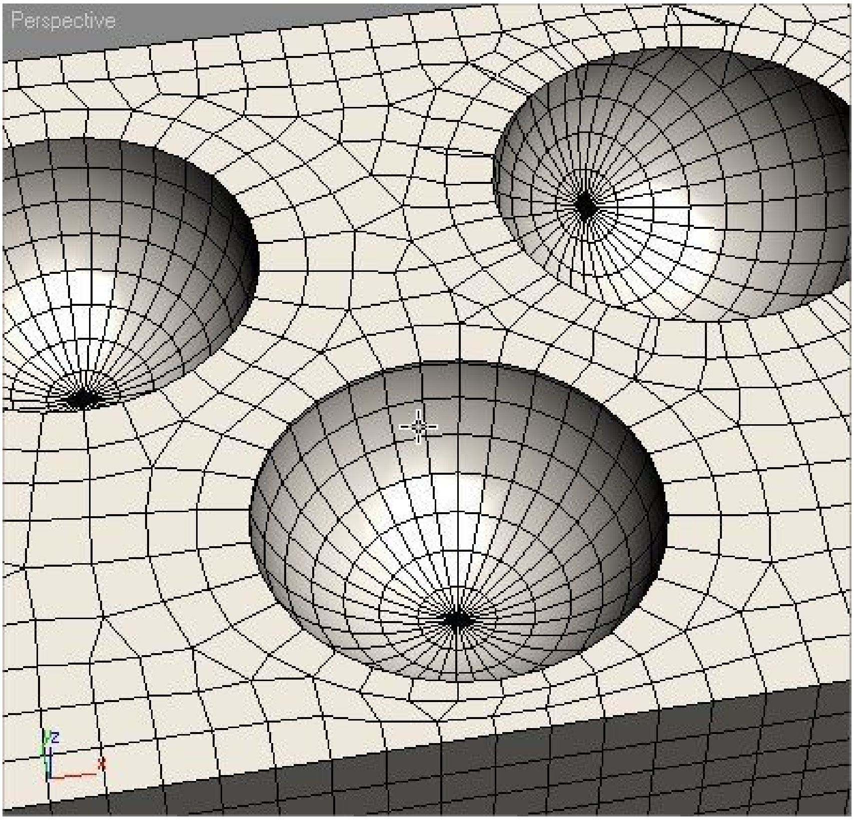


Quad mesh with all coplanar edges removed

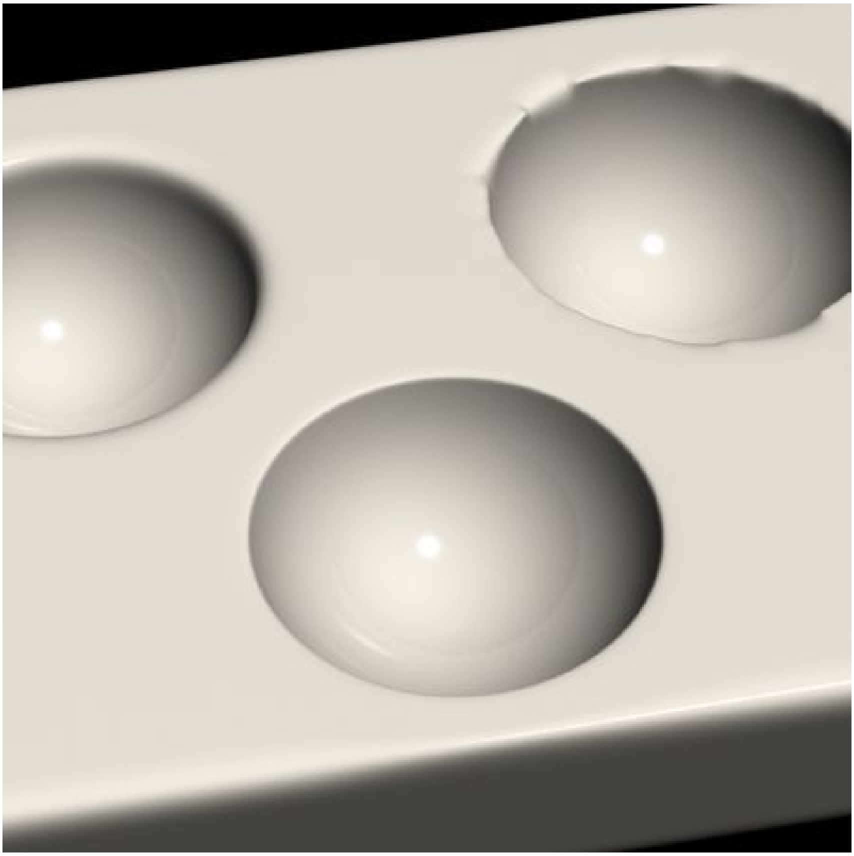
Problem #3: Poor alignment of original primitive meshes causes undesirable results.

Solution: Rotate or move original primitives into position to maximize mesh quality.

The following illustration shows the result of subtracting three spheres of the same size from a box. The left-hand sphere is aligned properly so that there are good quads along both boundaries. This should produce a good result when smoothed. The middle sphere was lifted so that there is a strip of very thin quads near the boundary. This produces very little smoothing along that edge, as you can see in the rendered image. The right-hand sphere was rotated, producing poor alignment and many triangles on the sphere as well as small quads on the plane of the box. You can see the undesirable results in the rendered image.



Quads produced by three spheres with different rotations and translations



Rendered image of smoothed three-sphere example

Dynamics Objects

Create panel > Geometry > Dynamics Objects

Create menu > Dynamics

Dynamics objects are similar to other mesh objects, except that they can be made to react to the motion of objects to which they are bound, or they can provide dynamic forces when included in a dynamics simulation on page 3852.

The following topics describe the types of dynamics objects and how to create and use them:

Spring Dynamics Object on page 890

Damper Dynamics Object on page 881

Damper Dynamics Object

Create panel > Geometry > Dynamics Objects > Object Type rollout > Damper button

Create menu > Dynamics > Damper

The Damper object provides a dynamic object that can behave as either a shock-absorber or an actuator. It consists of a base, a main housing, and a piston, with an optional boot. The piston slides within the main housing, providing different heights. The overall height can be affected by binding objects, in the same way as the Spring dynamic object.

NOTE Damper is similar to Spring in many respects. See Spring object on page 890 for more detailed descriptions of similar parameters and procedures.

Procedures

To create a damper:

1 Drag and release to specify the diameter.

2 Move the mouse and click to specify the overall height of the damper.

To use a damper in a dynamics simulation:

The following must be in place to use the damper forces in a dynamics simulation:

1 Bind two objects to the ends of the damper, and choose Bound to Object Pivots in the End Point Method group box at the top of the command panel.

2 In the dynamics simulation, add the damper to the Object List. (The damper itself is not adjusted in the dynamics simulation, so all of the dynamics parameters will be disabled for the damper object.)

3 Include at least one of the bound objects *or a parent* of one of the bound objects in the simulation. For example, you can bind two dummy objects to the ends of a damper, and one of the dummies can be the child of an

object that's included in the simulation. In this case, the dummy itself does not need to be in the simulation.

NOTE Damper is an "ideal" object with no mass. While it can be used in dynamics simulations, it cannot participate directly in collisions or effects. As a result, when you assign a damper object to a dynamics simulation, and then view it in the Edit Object dialog, all of the parameter settings are disabled.

Interface

End Point Method group

Free Damper/Actuator Choose this when using the damper as a simple object that's not bound to others or used in a dynamics simulation.

Bound to Object Pivots Choose this option when binding the damper to two objects, using the buttons described next.

Binding Objects group

Use these controls to pick the objects to which the damper is bound. To complete the binding, you must select two binding objects, and then click Bound to Object Pivots.

Piston (label) Displays the name of the object bound to the piston of the damper.

Pick Piston Object Click this button and then select the object to be bound to the piston of the damper.

Base (label) Displays the name of the object bound to the base of the damper.

Pick Base Object Click this button and then select the object to be bound to the base of the damper.

Free Damper Parameters group

Pin-to-Pin Height Use this field/spinner to specify the distance between the bottom center of the base and the top center of the piston when the damper is not bound.

Common Damper Parameters group

Renderable When on, the object appears in the rendering; when off, the object does not appear.

Material IDs are assigned to the damper object as follows:

1: Base

2: Main housing

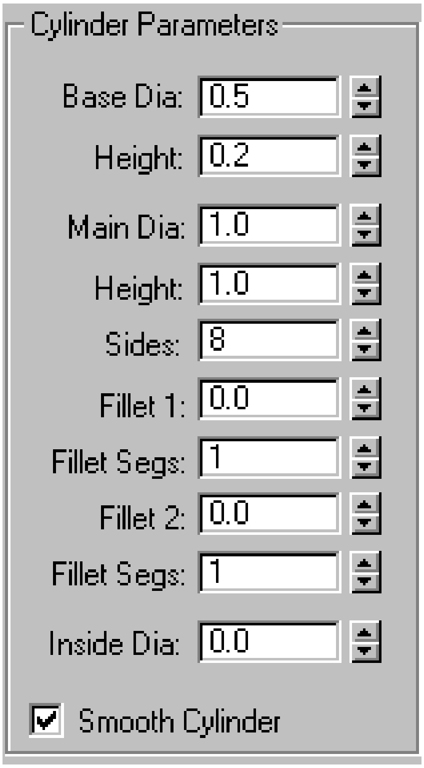
3: Piston

4: Boot Stop (appears only if you enable Boot Parameters)

5: Boot (appears only if you enable Boot Parameters)

Generate Mapping Coords Sets up the required coordinates for applying mapped materials to the object. Default=on.

Cylinder Parameters group



Provides parameters for the base and main cylinder of the damper.

Base Dia The diameter of the base, or "mount" of the damper.

Height The height of the base.

Main Dia The diameter of the main housing of the damper.

Height The height of the main housing.

Sides The number of sides of both the base and the main housing.

Fillet 1 The size of the fillet on the lower edge of the main housing.

Fillet Segs The number of segments for Fillet 1. The higher this setting, the rounder the fillet profile appears.

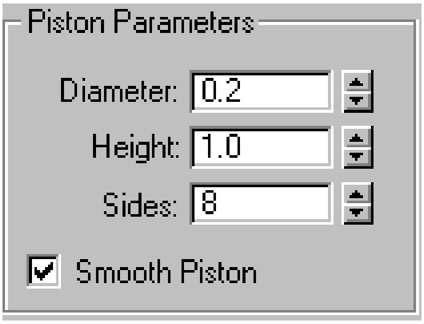
Fillet 2 The size of the fillet on the upper edge of the main housing.

Fillet Segs The number of segments for Fillet 2. The higher this setting, the rounder the fillet profile appears.

Inside Dia Specifies the inside diameter of the main housing, which is actually a tube rather than a cylinder.

Smooth Cylinder When on, smoothing is applied to both the base and the main housing.

Piston Parameters group



Provides parameters for the piston of the damper.

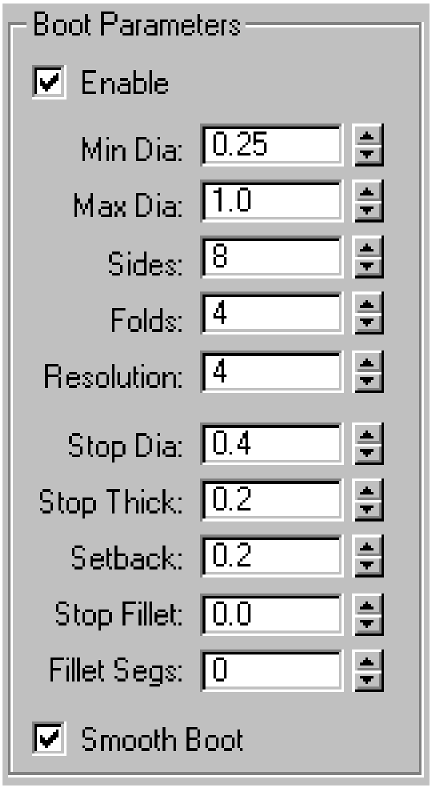
Diameter The diameter of the piston.

Height The height of the piston.

Sides The number of sides in the piston.

Smooth Piston When on, smoothing is applied to the piston.

Boot Parameters group



The boot is an optional component of the damper that's similar to the rubber "accordion" boot found on various types of dampers, such as shock absorbers. The boot acts like a bound dynamic object, in that one of its ends is bound to the main housing, while the other is bound to the piston. Thus, as the piston moves within the housing, the boot expands and contracts to follow.

Enable Turn this on to add the boot to the damper.

Min Dia The minimum diameter of the boot. This and the next parameter affect the depth of the accordion folds in the boot.

Max Dia The maximum diameter of the boot.

Sides The number of sides making up the boot.

Folds The number of accordion folds (bulges) along the height of the boot.

Resolution The number of segments in each fold.

Stop Dia The diameter of the stop, which is the ring at the top of the boot.

Stop Thick The thickness (height) of the stop ring.

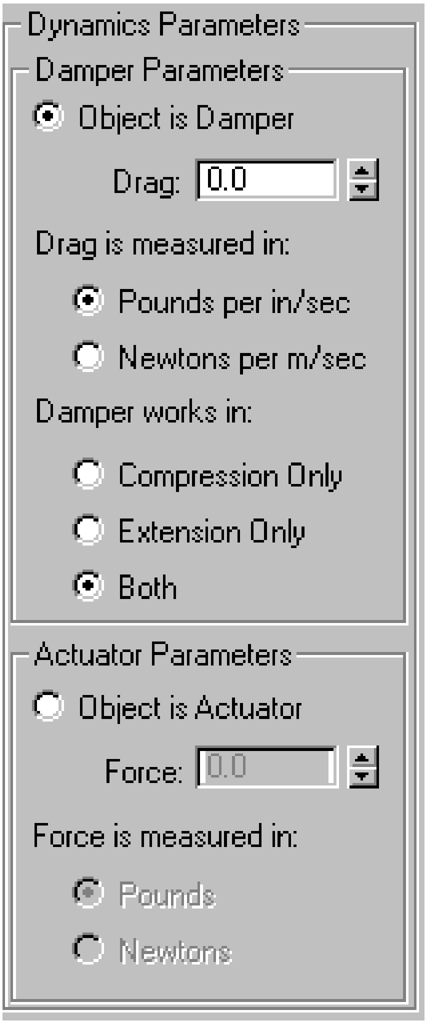
Setback The distance of the stop ring from the top of the piston.

Stop Fillet The size of the fillet on the upper edge of the stop ring.

Fillet Segs The number of segments the stop fillet. The higher this setting, the round the fillet profile appears.

Smooth Boot When on, smoothing is applied to the boot.

Dynamics Parameters group



Unlike the Spring object, the damper can also be used as an actuator. Basically, a damper absorbs force (like a shock absorber) while an actuator applies force.

The parameters in this group box, available only when End Point Method is set to Bound to Object Pivots, specify how forces are applied by the damper object in a dynamics simulation.

Damper Parameters Provides parameters for a damper type of object. Specifically, this simulates a viscous linear damper, which provides linear resistance to motion (between the two binding objects) proportional to the rate at which the damper experiences displacement. The faster it gets hit, the harder it fights back. Push it slowly, and there's almost no resistance.

- **Object is Damper** Select this option to use the damper object as a damper rather than an actuator.

Drag Specifies the force per unit linear speed, measured in one of the methods specified below.

- **Drag is measured in** Lets you specify the measurement of drag to use: Pounds per in(ch)/sec or Newtons per m(eter)/sec.
- **Damper works in** Provides directional options for the damper.

Compression Only The damper reacts only to compression forces.

Extension Only The damper reacts only to expansion forces.

Both The damper reacts to both compression and expansion forces.

Actuator Parameters Provides parameters for an actuator. When used as an actuator, the damper object exerts force between the two binding objects. A real-world example might be the thrusting piston in a log splitter. When used in a simulation, the force is applied by adjusting the value in the Force spinner. You can see the result only after solving the dynamics simulation.

- **Object is Actuator** Choose this when using the damper object as an Actuator.

Force Specifies the amount of force exerted between the two bound objects. Positive values push the objects apart, while negative values pull them together.

- **Force is measured in** Lets you specify the measurement of force to use: Pounds per inch or Newtons per meter.

Spring Dynamics Object

Create panel > Geometry > Dynamics Objects > Object Type rollout > Spring button

Create menu > Dynamics > Spring

The Spring object is a dynamics object in the shape of a coiled spring that lets you simulate a flexible spring in dynamics simulations. You can specify the overall diameter and length of the spring, the number of turns, and the diameter and shape of its “wire.” When used in a dynamics simulation, the compression and extension pressure of the spring are calculated as well.

Procedures

To create a spring:

1. Drag and release to specify the outside diameter.
2. Move the mouse and click to specify the overall length of the spring.

To use a spring in a dynamics simulation:

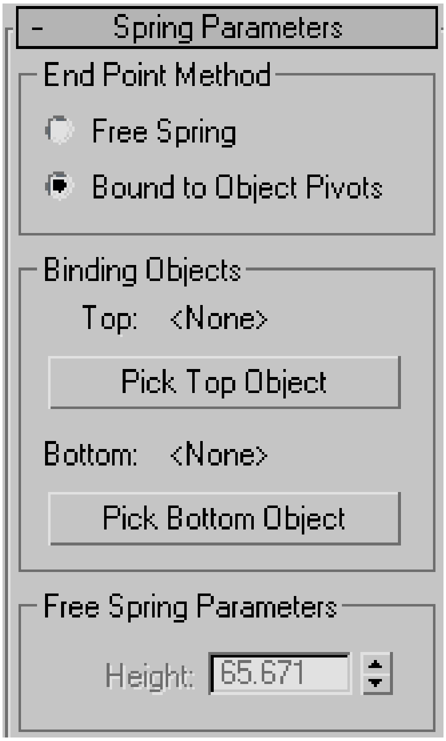
The following must be in place to use the spring forces in a dynamics simulation:

1. Bind two objects to the ends of the spring, and choose Bound to Object Pivots in the End Point Method group box at the top of the command panel.
2. In the dynamics simulation, add the spring to the Object List. (The spring itself is not adjustable in the dynamics Edit Object dialog, so all of the dynamics parameters will be disabled for the spring object.)
3. Include at least one of the bound objects *or a parent* of one of the bound objects in the simulation. For example, you can bind the ends of a spring to two dummy objects, and one of the dummies can be the child of an object that’s included in the simulation. The dummy without a parent will be stationary and the spring will pass its force through the other dummy to its parent.

   NOTE Spring is an "ideal" object with no mass. While it can be used in dynamics simulations, it cannot participate directly in collisions or effects. A spring can only exert force on other objects in simulations. As a result, when you assign a spring object to a dynamics simulation, and then view it in the Edit Object dialog, all of the parameter settings are disabled.

Interface

Spring Parameters rollout



End Point Method group

Free Spring Choose this when using the spring as a simple object that's not bound to other objects or used in a dynamics simulation.

Bound to Object Pivots Choose this when binding the spring to two objects, using the buttons described next.

Binding Objects group

Use these controls to pick the objects to which the spring is bound. "Top" and "Bottom" are arbitrary descriptors; the two bound objects can have any positional relationship to each other. To complete the binding, select two binding objects, and then click Bound to Object Pivots.

Each end point of the spring is defined by the center of the overall diameter and the center of the wire. This end point is placed at the pivot point of the object to which it is bound. You can adjust the relative position of the binding object to the spring by transforming the binding object while the Affect Object Only button is turned on in the Hierarchy > Pivot panel.

Top (label) Displays the name of the "top" binding object.

Pick Top Object Click this button and then select the "top" object.

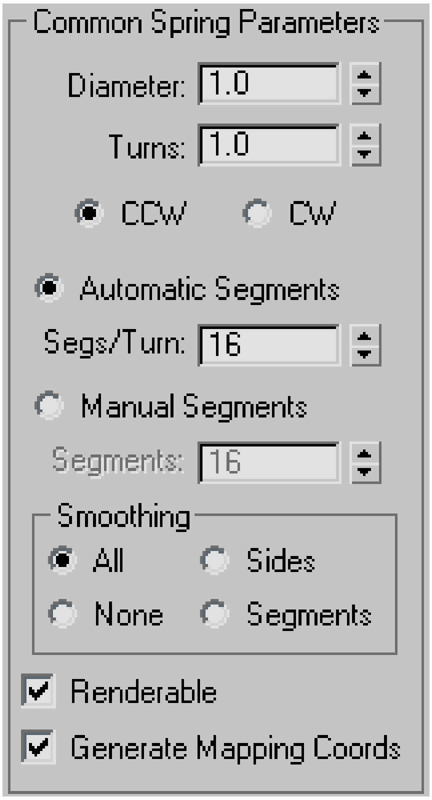
Bottom (label) Displays the name of the "bottom" binding object.

Pick Bottom Object Click this button and then select the "bottom" object.

Free Spring Parameters group

Height Use this field/spinner to set the straight-line height or length of the spring when it is not bound. This is not the actual length of the spring's wire.

Common Spring Parameters group



Diameter The overall diameter of the spring, as measured at the center of the wire. (The diameter of the wire itself has no effect on this setting.)

Turns The number of full 360-degree turns in the spring.

CCW/CW Specifies whether the coils of the spring are counterclockwise (CCW) or clockwise (CW).

Automatic Segments Choose this option to force each turn of the spring to contains the same number of segments, as specified in the Segs/Turn spinner. Thus, if you increase the number of turns, the number of segments also increases.

Segs/Turn This spinner lets you specify the number of segments in each 360-degree turn of the spring.

Manual Segments When this option is chosen, the length of the spring contains a fixed number of segments, no matter how many turns in the spring. Thus, as you increase the number of turns, you must manually increase the number of segments to maintain a smooth curve.

Segments This spinner lets you specify the total number of manual segments in the spring.

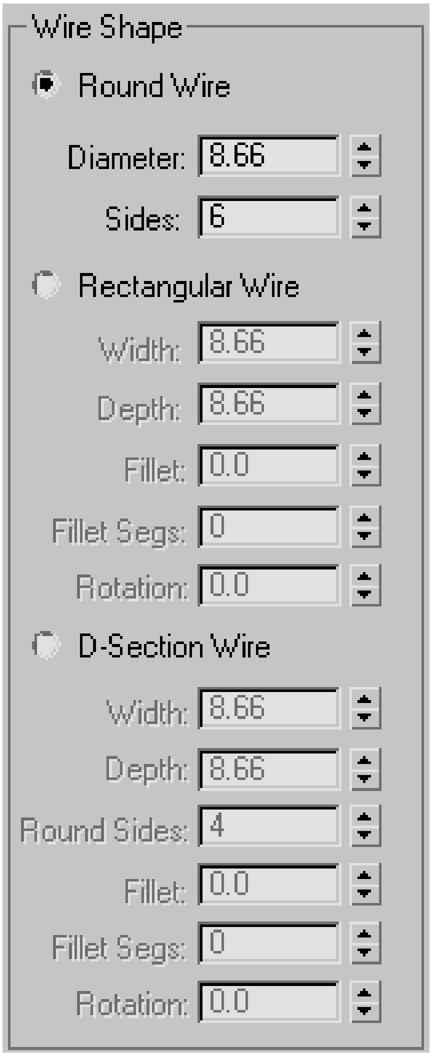
Smoothing Provides various methods of smoothing the object. The options here work the same as those in the Torus primitive on page 430.

- **All** All surfaces are smoothed.
- **Sides** Smoothing runs along the length of the wire, but not around its perimeter.
- **Segments** Smoothing runs around the perimeter of the wire, but not along its length.
- **None** No smoothing is applied.

Renderable When on, the object appears in the rendering; when off, the object does not appear.

Generate Mapping Coords Assigns mapping coordinates to the object. Default=on.

Wire Shape group



Provides three different types of wire cross-sections for the spring: round, rectangular, or D-shaped. Each type has its own set of parameters.

Round Wire Specifies a round wire for the spring.

- **Diameter** The diameter of the wire.
- **Sides** The number of sides that make up the cross section.

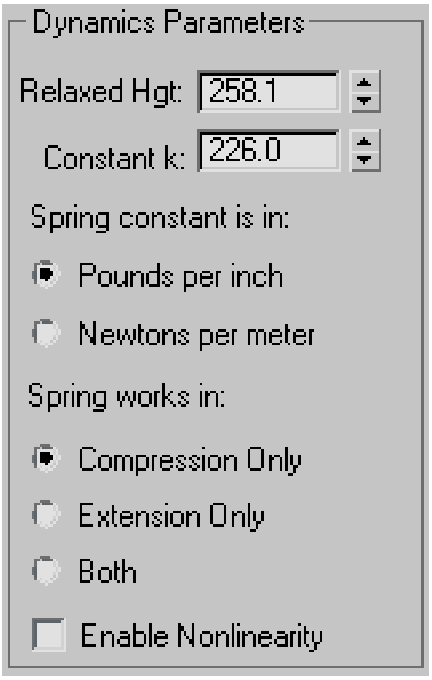
Rectangular Wire Specifies a rectangular wire.

- **Width** Determines the width of the cross section.
- **Depth** Determines the depth of the cross section.
- **Fillet** When combined with Fillet Segs (below), this lets you fillet (round) the corners of the cross section.
- **Fillet Segs** Specifies the number of segments in the fillet.
- **Rotation** Rotates the angle of the cross section along the entire length of the spring.

D-Section Wire Specifies a D-shaped wire.

- **Width** Determines the width of the cross section.
- **Depth** Determines the depth of the cross section.
- **Round Sides** Specifies the number of segments that make up the rounded side of the D-shape.
- **Fillet** When combined with Fillet Segs (below), this lets you fillet (round) the corners of the cross section.
- **Fillet Segs** Specifies the number of segments in the fillet.
- **Rotation** Rotates the angle of the cross section along the entire length of the spring.

Dynamics Parameters group



These parameters specify the forces that the spring contributes to a dynamic simulation.

Relaxed Hgt Specifies the height (or length) at which the spring is "relaxed" and therefore contributes no force--either compression or extension. For example, if the placement of the binding objects stretches the spring to a length of 50 units but the Relaxed Len is set to 30, then an extension force is in effect because the spring is stretched further than its relaxed length.

Constant k The amount of force exerted per unit change in length with respect to the Relaxed Hgt value. This could also be described as the measure of force-per-units-change in length as compared to the Relaxed Length. For example, if your spring is set to a Spring Constant of k=10 lb per in, and you stretch it to be ten inches longer than the Relaxed Hgt value, it will try to close with a force of 100 pounds. If you compress it two inches shorter than the Relaxed Hgt value, it will push back with 20 pounds of force.

Spring constant is in Lets you specify the measurement of force to use: Pounds per inch or Newtons per meter.

Spring works in Lets you specify the type of force you want the spring to exert. While most springs actually provide both compression and extension force, if your simulation requires only one, you can save calculation time by using one instead of both.

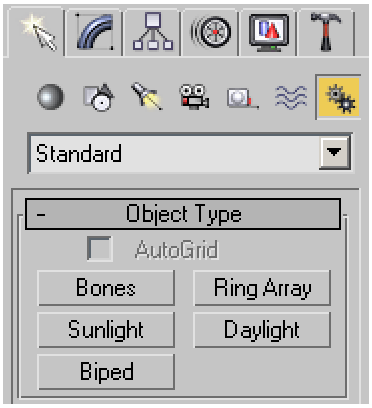
- **Compression Only** This type of spring provides only expansive force when its length is shorter than the specified Free Length.
- **Extension Only** Provides contractive force when its length is greater than the specified Free Length.
- **Both** Provides both expansive and contractive force, depending on the variation from Relaxed Hgt.

Enable Nonlinearity When on, the compression and extension of the spring are non-linear, based on the assumption that a spring has physical limits to the amount it can stretch or contract. Thus, the further the spring gets from the Relaxed Hgt setting, the less linear the feedback. The non-linear compression is calculated using the relationship between the coil dimensions, wire diameter, and length. Extension compares the relationship between the wire diameter and overall spring diameter.

Systems

Create panel > Systems

A system combines objects, linkages, and controllers to produce an object set that has behavior as well as geometry. Systems help you create animations that would be much more difficult or time-consuming to produce using features independently. Systems can range from simple object generators to full-scale subsystem programs.



■ Bones on page 901 creates a hierarchically linked set of bones and joints. You can transform and animate bones using both forward and inverse kinematics. See Animating with Forward Kinematics on page 3354 and Inverse Kinematics (IK) on page 3374.

■ Ring Array on page 923 creates a ring of boxes.

■ Sunlight on page 5139 creates and animates a directional light that follows the geographically correct angle and movement of the sun over the earth at a given location.

■ Daylight on page 5139 creates an assembly with a sky and a sun. Using the Get location function you can create and animate a light that follows the geographically correct angle and movement of the sun over the earth at a given location.

■ Biped on page 4147 creates a two-legged character skeleton designed for animation.

Systems are primarily intended for plug-in on page 8092 component software. Additional systems might be available if your configuration includes plug-in systems.

You can externally reference system objects in your scene. For more information, see XRef Objects on page 6936.

Procedures

To create a system:

1 On the Create panel, click Systems.

The Systems panel is displayed.

2 On the Object Type rollout, choose a system to create.

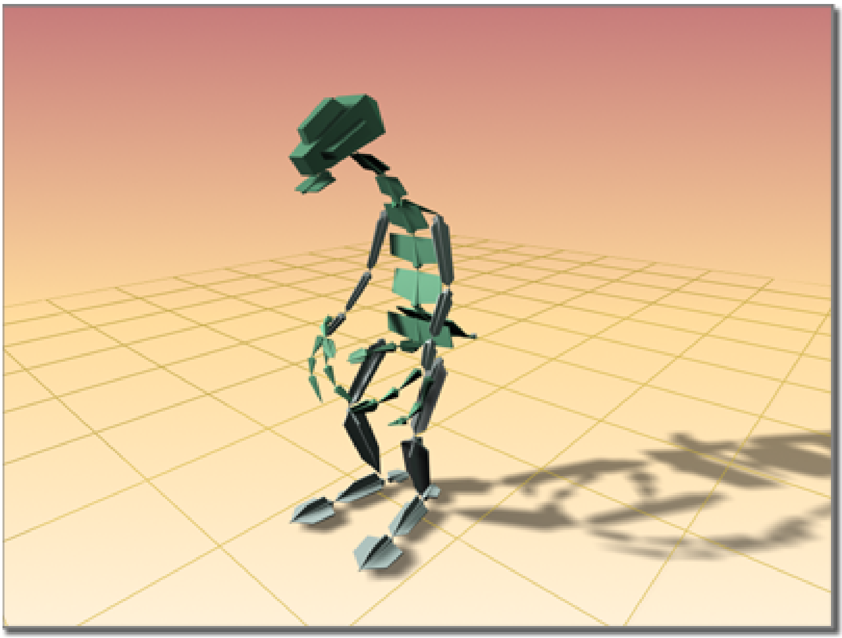
3 Drag in a viewport to create the system.

Bones System

Create panel > Systems > Bones button

Animation menu > Bone Tools > Create Bones

A Bones system is a jointed, hierarchical linkage of bone objects that can be used to animate other objects or hierarchies. Bones are especially useful for animating character models that have a continuous skin mesh. You can animate bones with forward or inverse kinematics. For inverse kinematics, bones can use any of the available IK solvers on page 3382, or through Interactive on page 3459 or applied IK on page 3462.

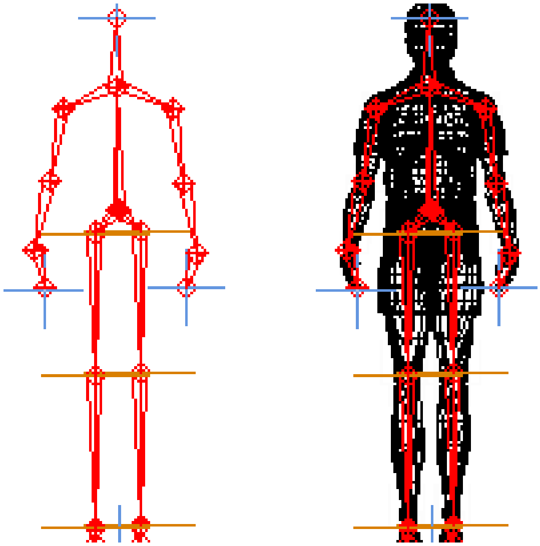


Dinosaur character modeled using bones

Bones are renderable objects. They have several parameters, such as taper and fins, that can be used to define the shape the bone represents. The fins make it easier to see how the bone is rotating.

For animation, it is very important that you understand the structure of a bone object. The bone's geometry is distinct from its link. Each link has a pivot point at its base. The bone can rotate about this pivot point. When you move a child bone, you are really rotating its parent bone.

It might be useful to think of bones as joints, because it is their pivot placements that matter, more than the actual bone geometry. Think of the geometry as a visual aid that is drawn lengthwise from the pivot point to the bone's child object. The child object is usually another bone.



Bones system seen alone and inside a wireframe model

Any hierarchy can display itself as a bone structure (see Using Objects as Bones on page 913), by simply turning on Bone On in the Bone Editing Tools rollout on page 915.

See also:

- Bone Tools on page 915

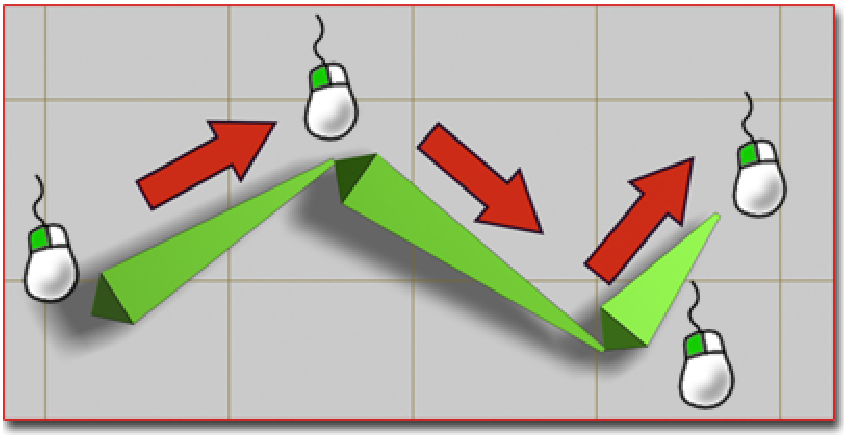
Creating Bones

You start creating bones by clicking the Create Bones button on the Bone Editing Tools rollout on page 915, or by clicking the Bones button in the Systems category on the Create panel.

To create bones, do the following.

1. Your first click in a viewport defines the start joint of the first bone.
2. The second click in a viewport defines the start joint of the next bone. Visually only one bone is drawn at this point because bones are visual aids drawn between two pivot points. It is the actual pivot point's placement that is important.
3. Each subsequent click defines a new bone as a child of the previous bone. The result of multiple clicks is a single chain of bones.
4. Right-click to exit bone creation.

   This creates a small "nub" bone at the end of the hierarchy, which is used when assigning an IK chain. If you are not going to assign an IK chain to the hierarchy, you can delete the small nub bone.



Creating a simple chain of three bones

3ds Max lets you create a branching hierarchy of bones. To create a branching hierarchy, such as legs branching from a pelvis, do the following:

1. Create a chain of bones, and then right-click to exit bone creation.

2 Click Bones (or Create Bones) again, and then click the bone where you want to begin branching. The new chain of bones branches from the bone you click.

WARNING The behavior of a branching bone hierarchy is not always intuitive.

NOTE You can also use Select And Link on page 3343 to connect one bone hierarchy to its branches. However, except for this one special case, using Select And Link with bones is *not recommended*. To edit an existing bone structure, whether branching or not, use the Bone Tools on page 915 instead.

Assigning IK Controllers to Bones

By default, bones are not assigned inverse kinematics (IK). Assigning an IK solver can be done in one of two ways. Typically, you create a bone hierarchy, then manually assign an IK solver. This allows for very precise control over where IK chains are defined.

The other way to assign an IK solver is more automatic. When you create bones, choose IK solver from the list in the IK Chain Assignment rollout, and then turn on Assign To Children. When you exit bone creation, the chosen IK solver is automatically applied to the hierarchy. The solver extends from the first bone in the hierarchy to the last.

For more information about IK, see Introduction to Inverse Kinematics on page 3374.

Setting the Initial Position of Bones

When you first create a bones system, the position of the bones is the *initial state*. Before you assign an IK solver or method, you can change the initial state of the bones by moving, rotating, or stretching the bones individually.

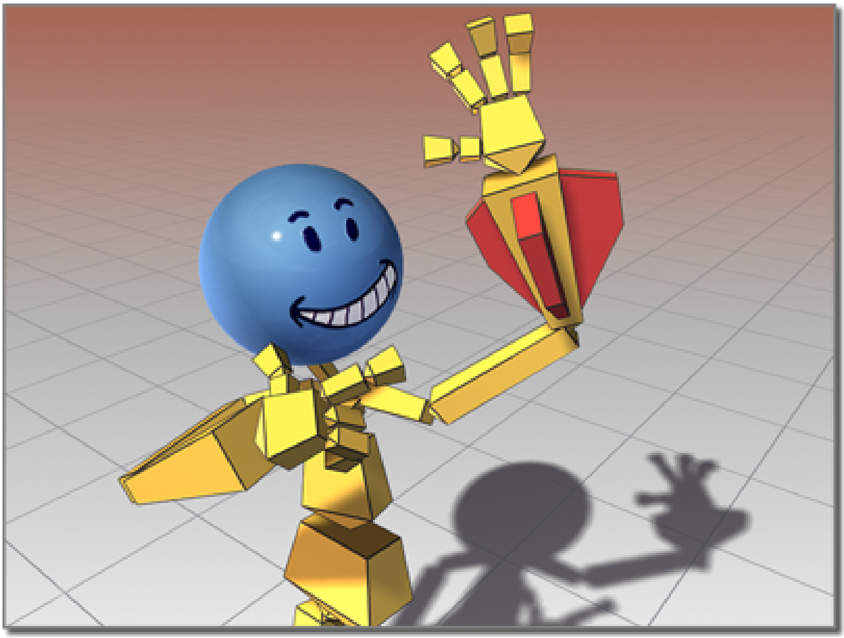
Bone Color

By default, bones are assigned the color specified for Bones in the Colors panel on page 7712 of the Customize User Interface dialog on page 7697. Choose Object as the Element and then choose Bones in the list. You can change the color of individual bones by selecting the bone, clicking the active color swatch next to the bone's name in the Create panel or Modify panel, and then selecting a color in the Object Color dialog on page 387.

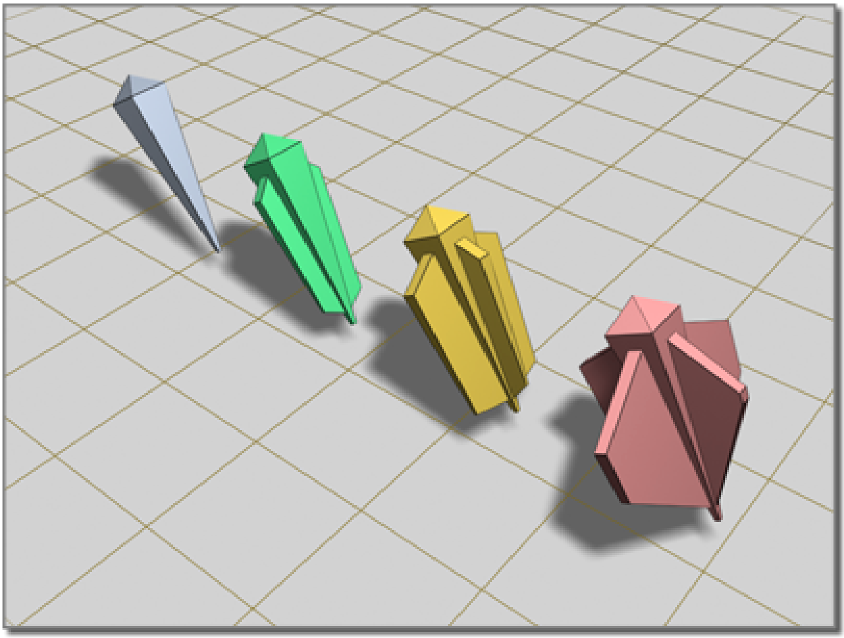
You can also use the Bone Tools on page 915 to assign bone colors, or to assign a color gradient to a bone hierarchy.

Bone Fins

Fins are visual aids that help you clearly see a bone’s orientation. Fins can also be used to approximate a character's shape. Bones have three sets of fins: side, front, and back. By default, fins are turned off.



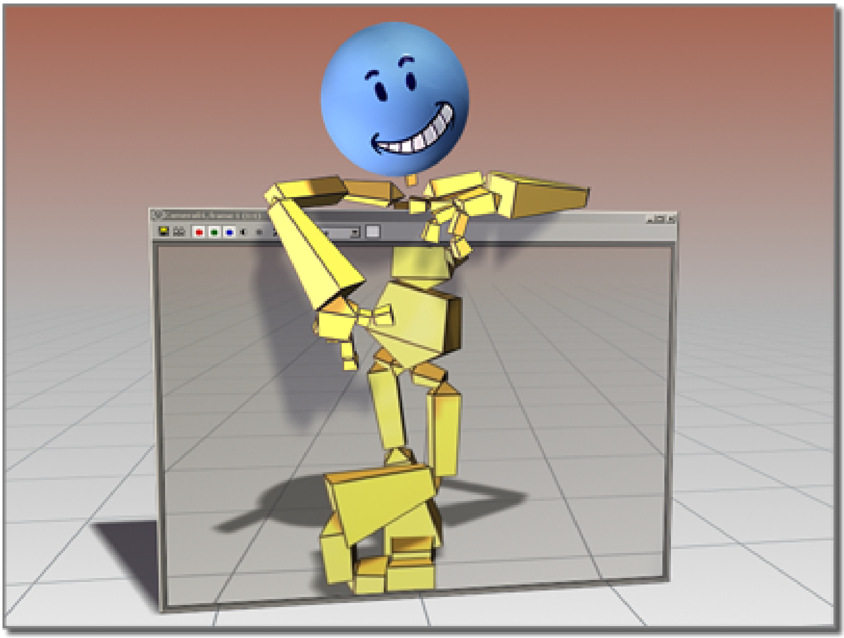
Bones can have fins.



Bones with various fin configurations

Renderable Bones

Bones can be renderable, though by default, they are not. To make a bone renderable, turn on the Renderable check box in the bone's Object Properties dialog on page 305.



Bones can be renderable.

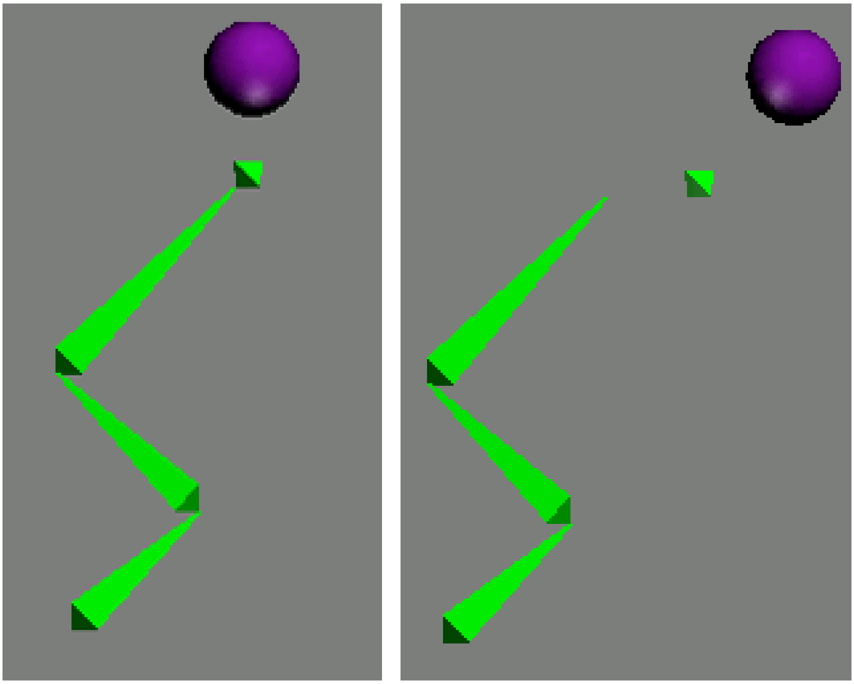
Object Properties for Bones

In addition to visual properties, bones have behavioral properties. The controls for these are located on the Bone Tools floater on page 915.

You can use these controls to turn other kinds of objects into bones.

Using Constraints with Bones

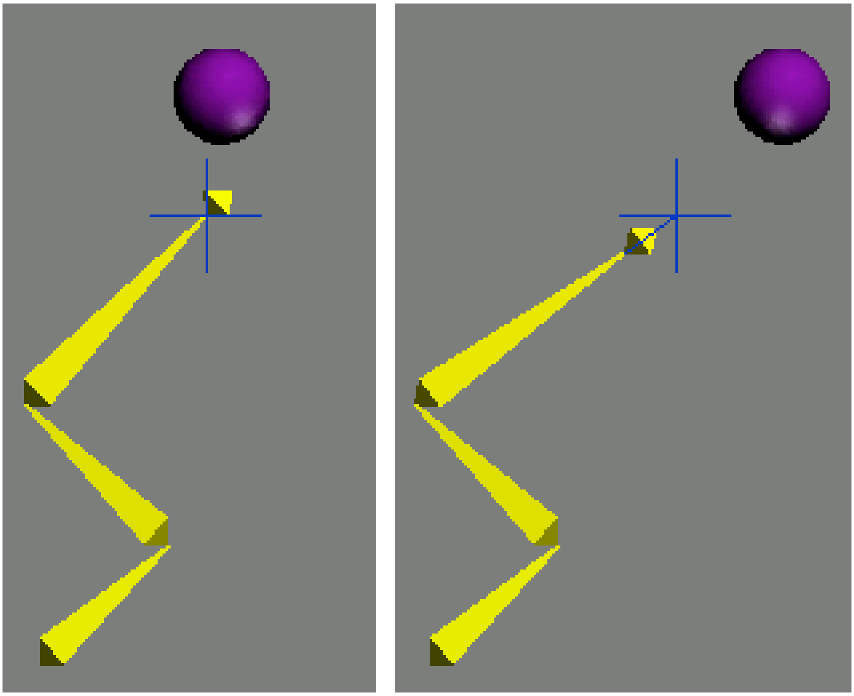
You can apply constraints on page 3287 to bones as long as an IK solver or method is not controlling the bones. If the bones have an assigned IK controller, you can constrain only the root of the hierarchy or chain. However, applying position controllers or constraints to a linked bone can cause undesirable effects, such as breaking of the bone chain.



The "nub" bone at the end of the chain has a Spring controller applied to it. The Spring controller is connected to an animated sphere.

Right: The sphere's movement breaks the bone chain.

To avoid this problem, don't apply position controllers directly to child bones. Instead, create an IK chain and apply the controller to the IK chain's end effector.



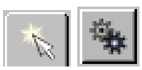
A IK chain has been applied, connecting the end nub to its parent bone. The IK chain's end effector is connected to the ball by a Spring controller.

Right: Now when the sphere moves, the IK chain prevents the bones from breaking.

Constraints and controllers that affect orientation only, such as Orientation or Look At, do not present this problem when applied to child bones.

Procedures

To create a bones system:

1.  On the Create panel, click Systems, and then click Bones.
   You can also access Create Bones through the Bone Tools rollout.
2. Click in a viewport.
   This creates a joint that is the base of the bone's hierarchy.
3. Drag to define the length of the second bone.

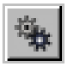
4 Click to set the length of the second bone, and then drag to create the third bone. Drag and click to continue creating new bones.

5 Right-click to end creation.

   3ds Max creates a small "nub" bone at the end of the hierarchy. This bone is used when assigning an IK chain.

   The first bone you create is at the top of the hierarchy. The last bone you create is at the bottom. For more about linked objects, see the Hierarchy Panel on page 7661.

To create a bones hierarchy with an IK solver automatically applied:

1   In the Create panel, click Systems, and then click Bones.

2 In the IK Chain Assignment rollout, select an IK solver from the list.

3 Turn on Assign To Children.

4 In a viewport, click and drag to create the bones. Right-click to end bone creation.

   After the bones are created, the chosen IK solver is applied to them.

To edit the appearance of a bone:

1 Select a bone.

2  Click the Modify tab on the command panel.

3 Change settings in the Bone Parameters rollout.

To change the length of bones after they've been created:

IMPORTANT Repositioning a bone affects its length visually. More important, it affects the bone's pivot position. The length of the bone is only a visual aid drawn between each bone's pivot point. A bone has only one pivot. The bone you see visually is connecting its pivot point to the next bone's pivot point.

---

1 Choose Animation menu > Bone Tools.

2 On the Bone Tools dialog, click Bone Edit Mode.

3 Move the child of the bone you want to change. The length of its immediate parent changes to reach the child bone.

4 Turn off Bone Edit Mode when you are finished editing the bones.

To add fins to bones:

1 Select the bone.

2 Choose Animation menu > Bone Tools.

3 Select the bones to which you want to add fins.

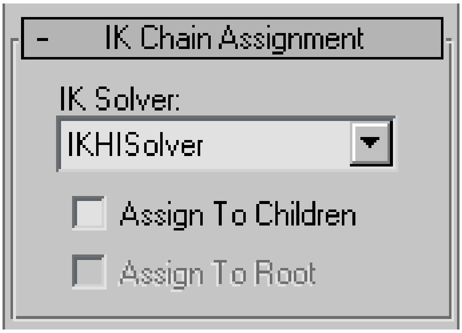
4 In the Fin Adjustment Tools rollout, turn on Side Fins, Front Fin or Back Fin.

5 Adjust the size and appearance of the fins with the appropriate spinners.

NOTE You can also add fins to an individual bone on the Modify panel.

Interface

IK Chain Assignment rollout (creation time only)



Provides the tools to quickly create a bone chain with an IK solver automatically applied. Also allows for bone creation with no IK solver.

IK Solver drop-down list Specifies the type of IK solver to be automatically applied if Assign To Children is turned on.

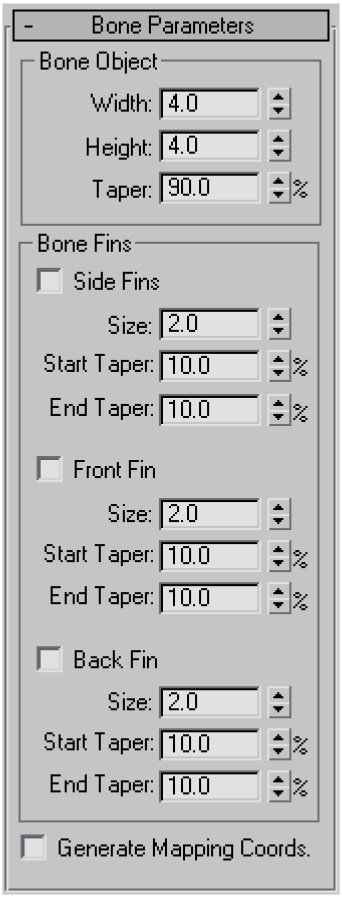
Assign To Children When on, assigns the IK solver named in the IK solver list to all the newly created bones except the first (root) bone. When off, assigns a standard PRS Transform controller on page 3220 to the bones. Default=off.

NOTE Choosing the SplineIKSolver and turning on Assign To Children causes the Spline IK Solver dialog on page 3452 to appear after bones have been created.

Assign To Root When on, assigns an IK solver to all the newly created bones including the first (root) bone.

Turning on Assign To Children also automatically turns on Assign To Root.

Bone Parameters rollout (creation and modification time)



These controls change the appearance of the bones.

Bone Object group

Width Sets the width of the bone to be made.

Height Sets the height of the bone to be made.

Taper Adjusts the taper of the bone shape. A Taper of 0 produces a box-shaped bone.

Bone Fins group

Side Fins Lets you add a set of fins to the sides of the bones you create.

- **Size** Controls the size of the fin.
- **Start Taper** Controls the start taper of the fin.
- **End Taper** Controls the end taper of the fin.

Front Fin Lets you add a fin to the front of the bone you create.

- **Size** Controls the size of the fin.
- **Start Taper** Controls the start taper of the fin.
- **End Taper** Controls the end taper of the fin.

Back Fin Lets you add a fin to the back of the bone you create.

- **Size** Controls the size of the fin.
- **Start Taper** Controls the start taper of the fin.
- **End Taper** Controls the end taper of the fin.

Generate Mapping Coords Creates mapping coordinates on the bones. Since the bones are renderable, they can also have materials applied, which can use these mapping coordinates.

Using Objects as Bones

Select a linked object or multiple objects linked to each other. > Animation menu > Bone Tools > Object Properties rollout > Bone On toggle

You can use arbitrary objects such as cylinders or boxes as bones, controlling their animation as if they were bones in a bones system on page 901. You can apply an IK solver on page 3382 to the boned objects.

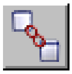
To use objects as bones, select them and then turn on the Bone On toggle on the Object Properties rollout of the Bone Tools dialog on page 915.

WARNING Turning on Freeze Length has no visible effect unless you transform the *child* of the object to which Freeze Length is applied.

Once you've set objects to function as bones, applying an IK solution behaves as it does for standard bone objects. The geometry of the boned objects can stretch or squash during animation.

Procedures

To use objects as bones:

1  Link the objects you want to display as bones.

2  Select all of these objects.

NOTE You can set a single object to work as a bone, but this doesn't have much use.

---

3 Choose Animation menu > Bone Tools.

The floating Bone Tools dialog is displayed.

4 On the Object Properties rollout, turn on Bone On.

3ds Max now treats the selected objects as bones.

5  Select the object to use as the start of the IK chain.

6 Choose Animation > IK Solvers > HI Solver.

You can choose a different IK solver, but the HI Solver is the preferred choice.

7 Click to select the end of the IK chain.

Now when you transform the boned objects, their movement is governed by the IK solver.

Bone Tools

Animation menu > Bone Tools

This command opens the Bone Tools floater, which provides functions for working with bones. The floater contains three rollouts, described in these topics:

Bone Editing Tools Rollout on page 915

Fin Adjustment Tools Rollout on page 919

Object Properties Rollout (Bone Tools) on page 921

Bone Tools Rollouts

Bone Editing Tools Rollout

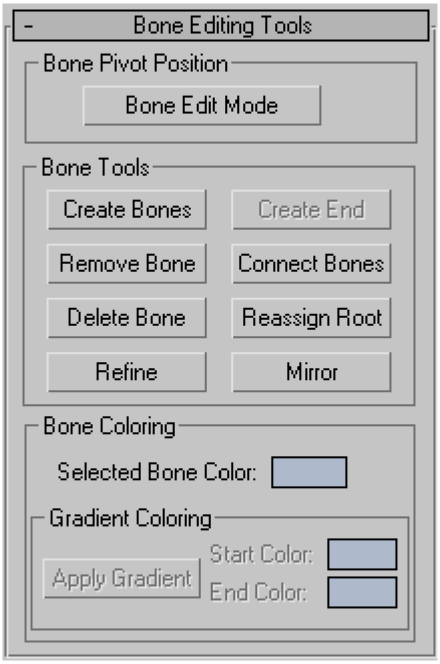
Animation menu > Bone Tools > Bone Tools floater > Bone Editing Tools rollout

Controls on the Bone Editing Tools rollout let you create and modify bone geometry and structure, and set bone color for one or more bones.

See also:

- Bones System on page 901

Interface



Bone Pivot Position group

Bone Edit Mode Lets you change the lengths of bones and their positions relative to one another.

When this button is on, you can change the length of a bone by moving its child bone. In effect, you can scale or stretch a bone by moving its child bone while in this mode. You can use this tool both before and after assigning an IK chain to the bone structure.

When Bone Edit Mode is on, you cannot animate, and when Auto Key or Set Key is on, Bone Edit Mode is unavailable. Turn off Auto/Set Key to edit bones.

NOTE Moving a bone in Bone Edit mode affects the length of *both the child and its parent*. If the bones aren't spatially aligned in the usual way (for example, if you are using other objects as bones), this might have unexpected results.

Bone Tools group

Create Bones Begins the bone-creation process. Clicking this button is the same as clicking Create panel > Systems > Bones on page 901.

Create End Creates a nub bone at the end of the currently selected bone. If the selected bone is not at the end of a chain, the nub is linked in sequence between the currently selected bone and the next bone in the chain.

Remove Bone Removes the currently selected bone. The bone's parent bone is stretched to reach the removed bone's pivot point, and any children of the removed bone are linked to its parent. Any IK chains that included the removed bone will remain intact.

Connect Bones Creates a connecting bone between the currently selected bone and another bone. When you click this button, a dotted line appears in the active viewport from the first selected bone. Move the cursor to another bone to create a new connecting bone. The first selected bone will become a parent to the connecting bone, which is in turn a parent to the second selected bone.

Delete Bone Deletes the currently selected bone, removing all its parent/child associations. A nub is placed at the end of the deleted bone's parent. Any IK chains that included this bone become invalid.

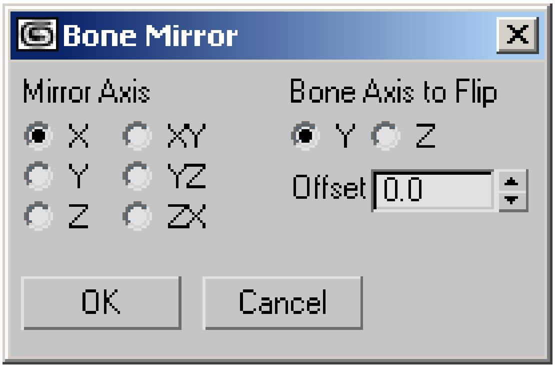
Reassign Root Makes the currently selected bone the root (parent) of the bone structure.

If the current bone is the root, clicking this has no effect. If the current bone is the end of the chain, the chain is completely reversed. If the current bone is in the middle of the chain, the chain becomes a branching hierarchy.

Refine Splits a bone in two. Click Refine, and then click a bone where you want it to split.

Mirror Opens the Bone Mirror dialog (see following), which lets you create mirror copies of selected bones without changing the sign of the bones' scale. Instead, Mirror flips one of the bone axes: Y or Z. You can specify the mirroring axis and the flip axis with the dialog controls.

Bone Mirror Dialog



Opens when you click the Mirror button. Use it to specify the mirroring axis, the flip axis, and an offset value.

While the dialog is open, you can see a preview of the mirrored bone(s) in the viewports. Click OK to create the bones, or Cancel to prevent creation.

Mirror Axis Choose an axis or plane about which the bones will be mirrored: X/Y/Z or XY/YZ/ZX.

Bone Axis to Flip To avoid creating a negative scale, choose the bone axis to flip: Y or Z.

Offset The distance between the original bones and the mirrored bones. Use this to move the mirrored bones to the other side of the character.

Bone Coloring group

Selected Bone Color Sets the color for selected bones.

Apply Gradient Applies a gradient color across several bones based on the Start Color and End Color values. This option is available only when two or more bones are selected. The Start Color is applied to the highest parent bone in the selected chain, while the End Color is applied to the last child object in the selected chain. Intermediate colors in the gradient are applied to bones in between.

Start Color Sets the starting color for the gradient.

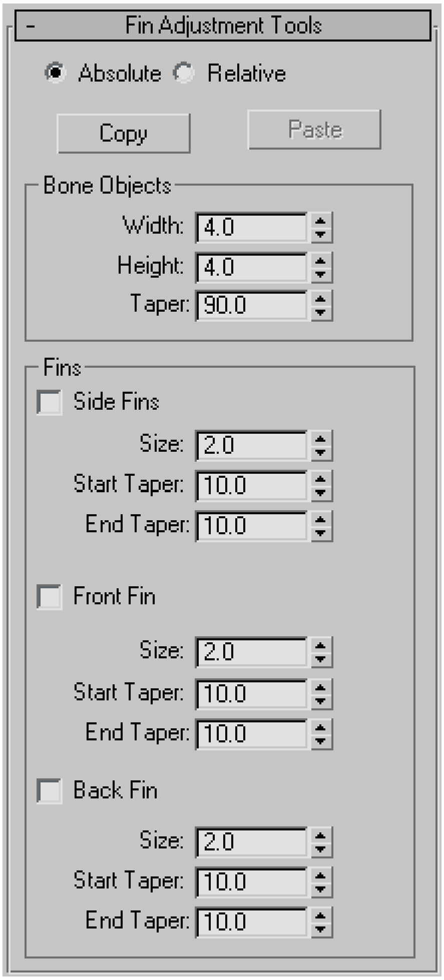
End Color Sets the ending color for the gradient.

Fin Adjustment Tools Rollout

Animation menu > Bone Tools > Bone Tools floater > Fin Adjustment Tools rollout

Controls on the Fin Adjustment Tools rollout are for adjusting some aspects of bone geometry, including fins.

Interface



Absolute Sets the fin parameters as absolute values. Use this option to set the same fin values for all selected bones.

Relative Sets the fin parameters relative to their current values. Use this option to retain size relationships between bones with different-sized fins.

Copy Copies the bone and fin settings for the currently selected bone, in preparation for pasting to another bone.

Paste Pastes the copied bone and fin settings to the currently selected bone.

Bone Objects group

Width Sets the width of the bone.

Height Sets the height of the bone.

Taper Adjusts the taper of the bone shape. A Taper with a 0 value produces a box-shaped bone. Higher values pinch the bone where it joins its child bone, while lower values expand that end of the bone.

Fins group

Side Fins Adds side fins to selected bones.

- **Size** Controls the size of the fin.
- **Start Taper** Controls the start taper of the fin.
- **End Taper** Controls the end taper of the fin.

Front Fin Adds front fins to selected bones.

- **Size** Controls the size of the fin.
- **Start Taper** Controls the start taper of the fin.
- **End Taper** Controls the end taper of the fin.

Back Fin Adds fins to the backs of selected bones.

- **Size** Controls the size of the fin.
- **Start Taper** Controls the start taper of the fin.
- **End Taper** Controls the end taper of the fin.

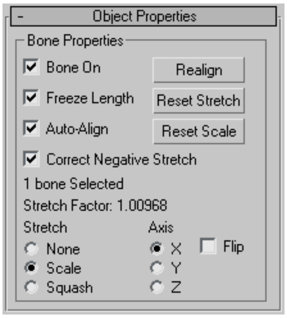
Object Properties Rollout (Bone Tools)

Animation menu > Bone Tools > Bone Tools floater > Object Properties rollout

Controls on the Object Properties rollout for bones let you turn other objects into bones. They also control bone rigidity and alignment.

NOTE You can reset the scale of bones with the Reset Scale option.

Interface



Bone On/Off When turned on, the selected bone or object behaves as a bone. Turning this option off causes the object to stop behaving like a bone: there is no auto alignment or stretching. Default=on for bone objects, off for other kinds of objects.

NOTE Turning this option on doesn't immediately cause objects to align or stretch. However, future transforms of children can cause rotation and stretching.

Freeze Length When turned on, the bone maintains its length. When turned off, the bone's length is based on the translation of its child bone. This option is available only if Bone On is on. Default=on.

WARNING When you turn on Freeze Length, this has no visible effect unless you transform the *child* of the object to which Freeze Length is applied.

Auto-Align When turned off, the bone's pivot point doesn't align to its child object. The translation of a child bone will not be converted into rotation of the parent. Instead, the child is allowed to move away from the parent's X axis. This option is available only if Bone On is on. Default=on.

NOTE Changing the Auto-Align state does not have an immediate visual effect on the skeleton. It affects future behavior when bones are moved.

Correct Negative Stretch When turned on, any stretching of the bone that results in a negative scale factor will be corrected to a positive number. This option is available only if Bone On is on. Default=on.

Realign Causes the bone's X axis to realign and point at the child bone (or the average pivot of multiple children). Normally this alignment is maintained, and there is no need to use this option. However, it is possible for the bones to come out of alignment by turning off Auto-Align and moving a child bone. Use Realign to align the bone back to its child. This option is available only if Bone On is on.

Reset Stretch Stretches the bone to reach its child object if the child has been moved away from the bone. This option is available only if Bone On is on.

Reset Scale Resets a stretched bone's internally calculated scale to 100% on each axis. Using this option avoids unexpected behavior due to objects which are both linked and scaled. This option has no visual effect on the bone. This option is available only if Bone On is on.

Stretch Factor Information

Under the Correct Negative Stretch options is a text display giving information about the number of bones selected and the respective stretch factor for all three axes. If more than one bone is selected, the Stretch Factor text displays *undefined*.

NOTE The Stretch Factor text only updates when you are in Bone Edit Mode on page 915.

Stretch and Axis Options

Stretch Determines what kind of stretch takes place when the child bone is transformed and Freeze Length is off. Default=Scale.

- **None** No stretch takes place.
- **Scale** Lets the bone scale. The stretch happens along one axis.
- **Squash** Lets the bone squash. The bone gets fatter as it gets shorter, and thinner as it gets longer.

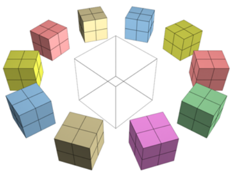
Axis Determines the axis used for the stretch.

- **X/Y/Z** Choose the axis for scaling or squashing.
- **Flip** Flips the stretch along the selected axis.

Ring Array System

Create panel > Systems > Ring Array button

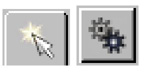
The Ring Array object consists of a dummy object on page 2615 surrounded by a ring of boxes. You can arrange the boxes in the ring along a sine curve, vary their number, and animate the ring array's parameters. You can also replace the boxes with other objects using Track View, as described in the Procedures section, below.



Example of ring array

Procedures

To create the ring array system:

1.  On the Create panel, click Systems, and then click Ring Array.
2. Drag in a viewport to set the center and radius of the array.

   A dummy object appears at the center. By default, four boxes are evenly spaced in a circle around it.

To animate the ring array:

1. 

Turn on Auto Key.
2. Move to a nonzero frame.

3 Adjust the ring array parameters.

   You can't animate the number of boxes in the ring.

4 Repeat steps 2 and 3 for additional keyframes.

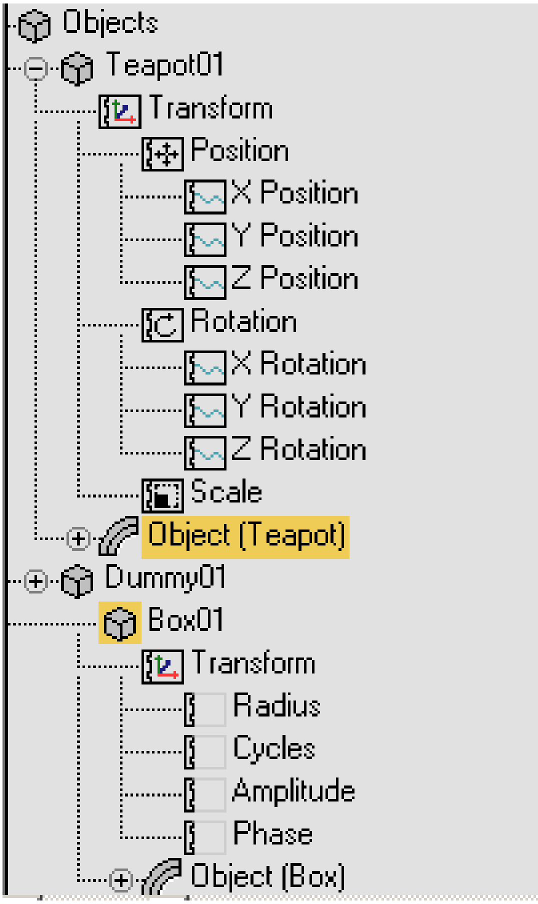
NOTE To animate the ring array after creation, use the Motion panel, not the Modify panel.

To put other kinds of objects in the ring:

You can use either version of Track View: Curve Editor or Dope Sheet.

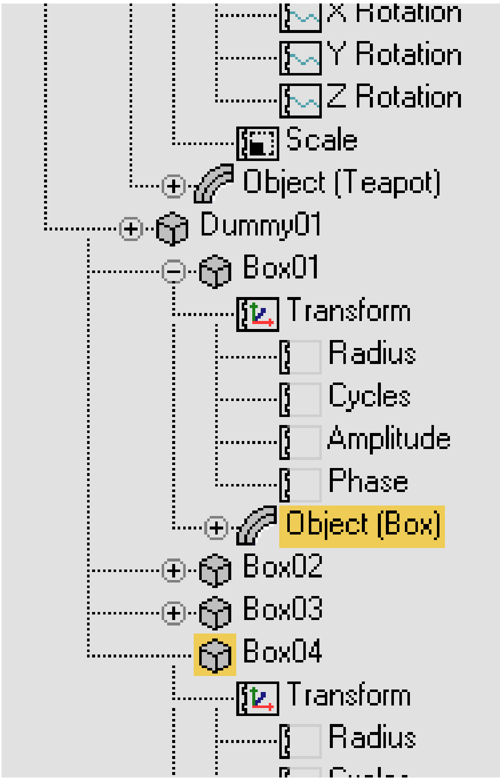
1 In the Track View Controller window, click the name of object container of the object to put in the ring.

   The name highlights.



2 Still in the Controller window, right-click and choose Copy from the menu.

3 Highlight the object container of one of the ring array boxes.

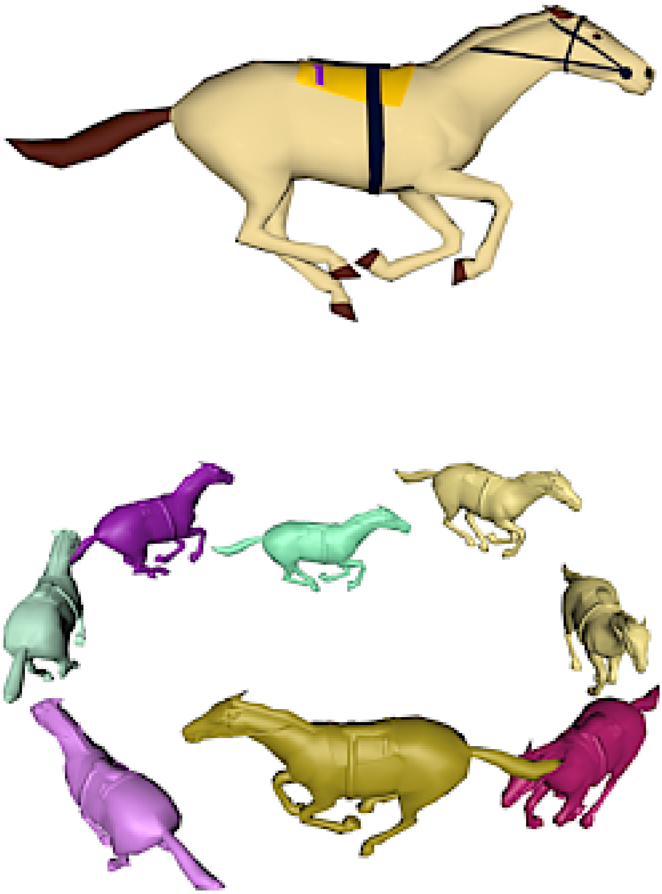


4 Right-click and choose Paste.

5 In the Paste dialog, choose Copy or Instance. Optionally, to replace all the boxes with the copied object, turn on Replace All Instances. Click OK.

The box or boxes are replaced with the copied object.

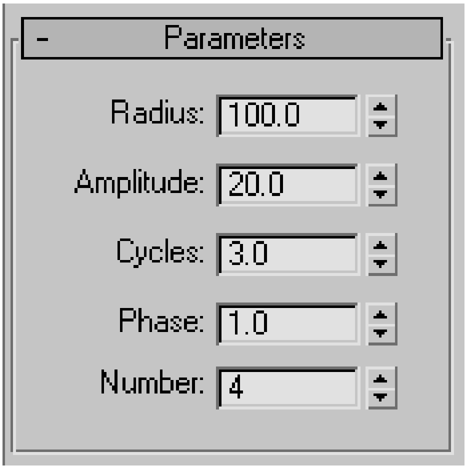
TIP To see the replacement objects, you might need to refresh the viewports.



Top: Object substituted for boxes in array

Bottom: The result

Interface



These parameters control ring arrays. To adjust and animate the ring array after creation, select one of the array objects (not the dummy), and then go to the Motion panel, not the Modify panel.

Radius Sets the radius of the ring. You set the initial Radius value when you drag to create the ring array.

Amplitude Sets the amplitude of the ring's sine curve, in active units. Amplitude is a height offset from the local origin of the center dummy object.

Cycles Sets the number of cycles in the ring's sine curve. When Cycles is 0.0, the ring is flat. When Cycles is 1.0, the ring is tilted. Greater values increase the number of peaks in the curve.

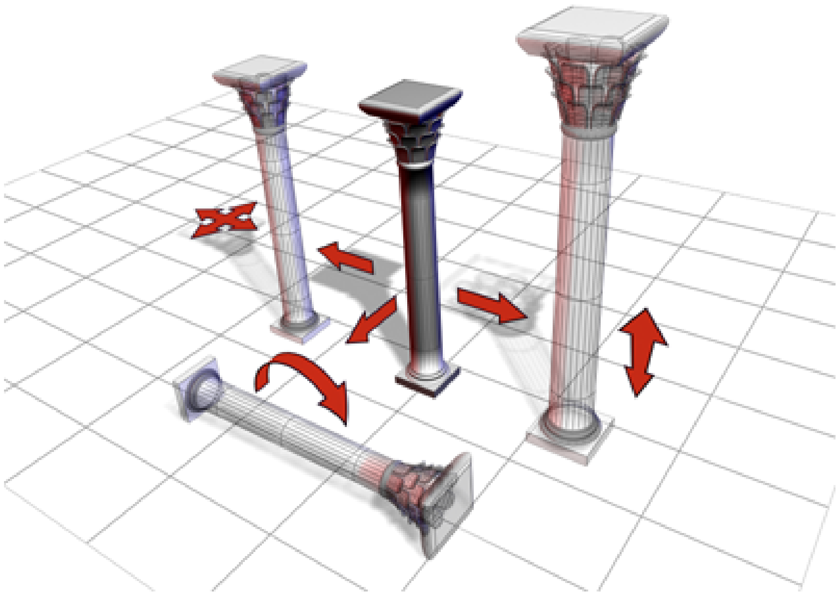
Phase Offsets the phase of the wave. That is, it has the effect of moving the wave along the circumference of the ring. Whole values have no effect; only fractional values do.

Number Sets the number of boxes in the ring.

Moving, Rotating, and Scaling Objects

9

To change an object's position, orientation, or scale, click one of the three transform buttons on the main toolbar or choose a transform from a shortcut menu. Apply the transform to a selected object using the mouse, the status bar Coordinate Display fields, a type-in dialog, or any combination of the above.



The column can be moved, rotated, and scaled.

These topics describe how to use and animate transforms, with procedures:

Using Transforms on page 931

Using Transform Gizmos on page 934

Transform Type-In on page 944

Animating Transforms on page 947

Transform Managers on page 949

Specifying a Reference Coordinate System on page 951

Choosing a Transform Center on page 952

Using the Axis Constraints on page 955

Transform Commands on page 958

Transform Coordinates and Coordinate Center on page 967

Transform Tools on page 980

Scaling and Dimensions

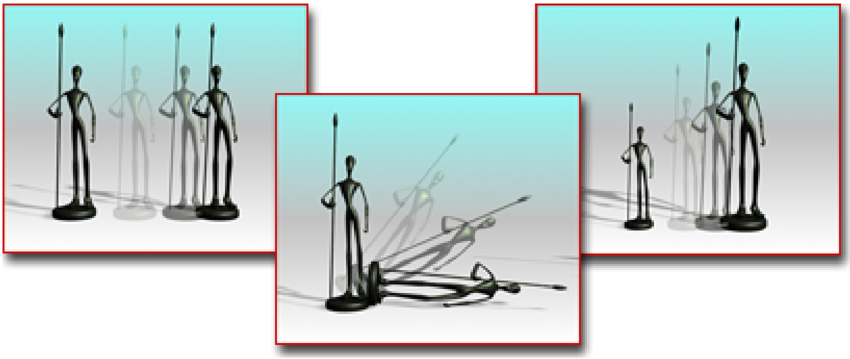
If you scale an object and later check its base parameters in the Modify panel, you see the dimensions of the object before it was scaled. The base object exists independently of the scaled object that is visible in your scene.

You can use the Measure utility on page 2680 to measure the current dimensions of an object that has been scaled or changed by a modifier.

See also:

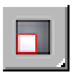
- Creating Copies and Arrays on page 1023
- Using +Clone on page 1037

Using Transforms



Changing a model by changing its position, rotation, or scale

A transform is an adjustment of an object's position, orientation, or scale, relative to the 3D world (or world space) in which you're working. You can apply three types of transform to an object:

-  Position on page 959
-  Rotation on page 960
-  Scale on page 962

This section presents these brief topics designed to help you quickly start learning how to transform objects, and how to animate your transforms:

Using Transform Gizmos on page 934

Animating Transforms on page 947

Transform Managers on page 949

Specifying a Reference Coordinate System on page 951

Choosing a Transform Center on page 952

Using the Axis Constraints on page 955

Failure to Move or Rotate

In some cases, an object might fail to move or rotate, even when the proper button is on and the object is selected. This could be due to one of the following reasons:

- The object is frozen on page 212.
- A transform controller has been assigned to the object. See Animation Controllers on page 3133.
- Inverse Kinematics mode is on and the preference called Always Transform Children of the World is off. See Introduction to Inverse Kinematics (IK) on page 3374.

Procedures

To transform an object using the main toolbar:

1. On the main toolbar, click one of the three transform buttons: Select And Move on page 959, Select And Rotate on page 960, or Select And Scale on page 962. These buttons are usually referred to as Move, Rotate, and Scale.
2. Position the mouse over the object you want to transform.
   - If the object is already selected, the cursor changes to indicate the transform.
   - If the object is not selected, the cursor changes to a crosshairs to show that the object can be selected.
3. Drag the mouse to apply the transform.

   If you drag the mouse over an unselected object, it becomes selected and is also transformed.

   You can restrict transforms to one or two axes easily with the transform gizmo on page 934.

To cancel a transform:

- Right-click while you're dragging the mouse.

To transform an object from the quad menu:

1 Right-click a selected object. The quad menu on page 7516 lists the three transforms.

2 Choose one of the transforms. The equivalent transform button is selected on the main toolbar.

3 Drag the object to apply the transform.

To use transform type-in:

1 Choose Tools menu > Transform Type-In to display the dialog.

2 Apply a transform to a selected object.

3 You can do any of the following, switching from one to the other as required.

- Type a value in an axis field and press Enter to apply the transform change to the selection.
- Drag a spinner in an axis field to update the selection.
- Drag the object to apply the transform and read the resulting change in the dialog.

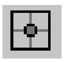
For example, if Move is active, the dialog fields read out both the absolute and offset positions of the selected object in world space. If no object is selected, the fields turn gray.

To use transform type-in on the status bar:

1 Select an object or a group of objects.

2 On the main toolbar, choose a transform (Move, Rotate, or Scale) to perform on the objects.

3 On the status bar, you can do any of the following, switching from one to another as required:

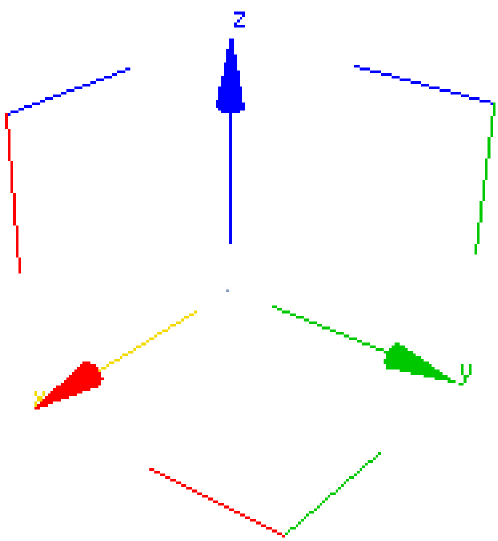
-  Type a value in an axis field and press Enter to apply the transform change to the selection. The Absolute/Offset toggle, to the right of the X, Y, and Z fields, lets you switch between entering values that are absolute (in world space) or offset (relative to the selection's present position, orientation, and dimensions).

- Drag a spinner in an axis field to update the selection.
- Drag the object to apply the transform and read the resulting change in the X, Y, and Z fields.

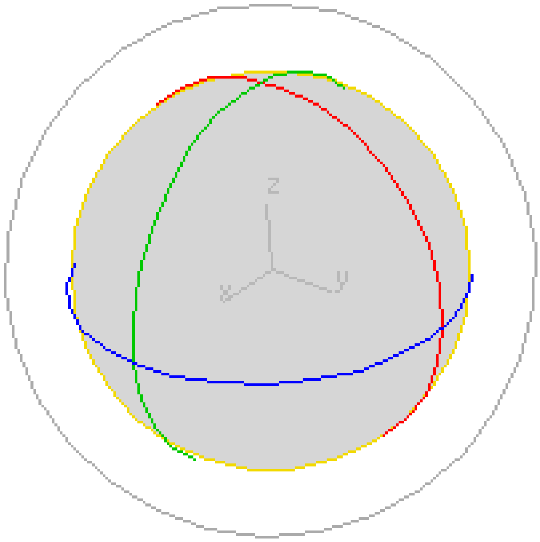
TIP To see the Z field, drag the transform type-in portion of the toolbar while a pan hand is visible.

Using Transform Gizmos

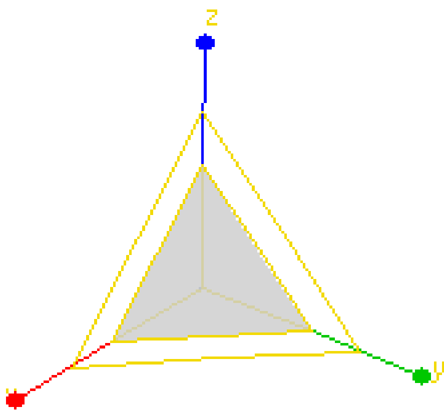
Select an object. > main toolbar > Click any transform button to display the object's Transform Gizmo icon.



Move gizmo



Rotate gizmo



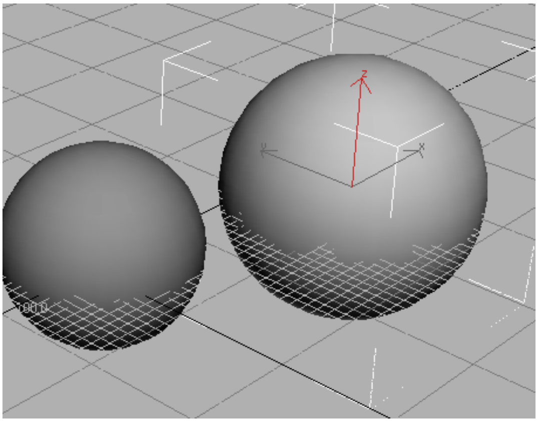
Scale gizmo

The Transform gizmos are viewport icons that let you quickly choose one or two axes when transforming a selection with the mouse. You choose an axis by placing the mouse over any axis of the icon, and then drag the mouse to transform the selection along that axis. In addition, when moving or scaling an object, you can use other areas of the gizmo to perform transforms along any two axes simultaneously. Using a gizmo avoids the need to first specify a transform axis or axes on the Axis Constraints toolbar on page 7503, and also lets you switch quickly and easily between different transform axes and planes.

A Transform gizmo appears when one or more objects are selected and one of the transform buttons (Select And Move on page 959, Select And Rotate on page 960, or Select And Scale on page 962) is active on the main toolbar. Each transform type uses a different gizmo. By default, each axis is assigned one of three colors: X is red, Y is green, and Z is blue. The corners of the Move gizmo are assigned the two colors of the related axes; for example, the corner for the XZ plane is red and blue.

When you position the mouse over any axis, it turns yellow to indicate that it's active. Similarly, position the mouse over one of the plane handles, and both associated axes turn yellow. You can now drag the selection along the indicated axis or axes. Doing so changes the Axis Constraints toolbar "Restrict to ..." setting on page 955.

NOTE When no transform tool is active and you select one or more objects, axis tripods appear in the viewport.



The axis tripod appears when the transform gizmo is inactive.

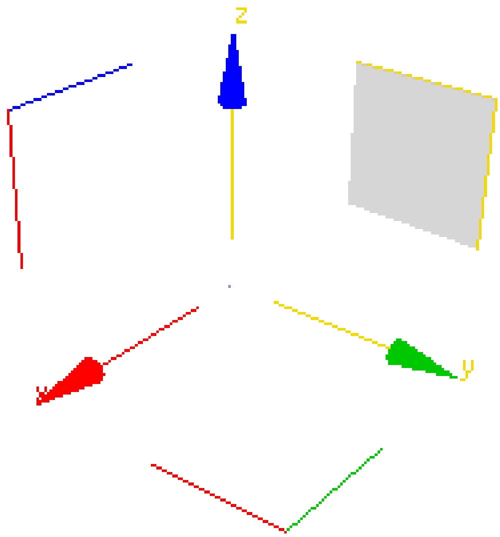
Each axis tripod consists of three lines, labeled X, Y, and Z, and shows you three things:

- The orientation of the tripod reveals the orientation of the current reference coordinate system.
- The location of the junction of the three axis lines shows where the current transform center is.
- The highlighted red axis lines show the current axis constraints on page 955.

Move Gizmo

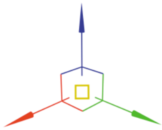
The Move gizmo includes plane handles, and the option to use a center box handle.

You can select any of the axis handles to constrain movement to that axis. In addition, the plane handles allow you to constrain movement to the XY, YZ, or XZ planes. The selection hotspot is within the square formed by the plane handles. You can change the size and offset of the handles and other settings on the Gizmos panel on page 7778 of the Preferences dialog on page 7743.



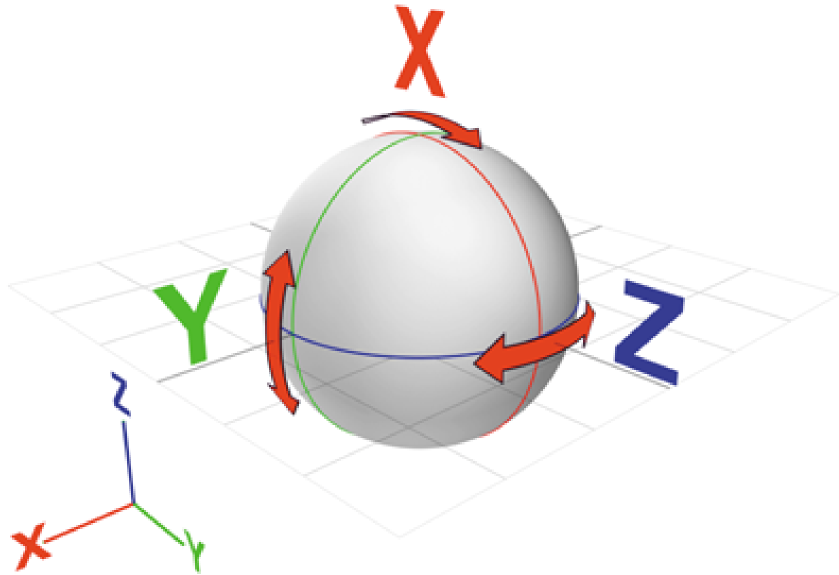
The Move gizmo with the YZ axes selected.

You can constrain translation to the viewport plane by dragging the center box. To use this optional control, turn on Move In Screen Space on page 7780.

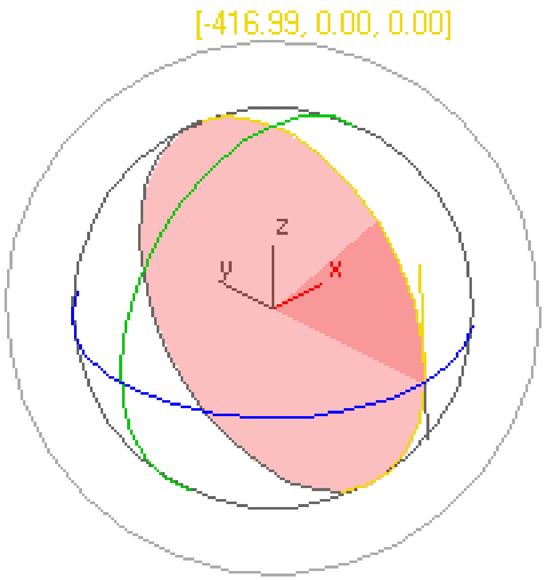


Rotate Gizmo

The Rotate gizmo is built around the concept of a virtual trackball. You can rotate an object freely, about the X, Y, or Z axis, or about an axis perpendicular to the viewport.



The axis handles are circles around the trackball. Drag anywhere on one of them to rotate the object about that axis. As you rotate about the X, Y, or Z axis a transparent slice provides a visual representation of the direction and amount of rotation. If you rotate more than 360°, the slice overlaps and the shading becomes increasingly opaque. The software also displays numerical data to indicate a precise rotational measurement.



In addition to XYZ rotation, you can also use free rotation or the viewport handle to rotate objects.

Drag inside the Rotate gizmo (or the outer edge of the gizmo) to perform free rotation. Rotation should behave as if you were actually spinning the trackball.

The outermost circle around the Rotate gizmo is the Screen handle, which lets you rotate the object on a plane parallel to the viewport.

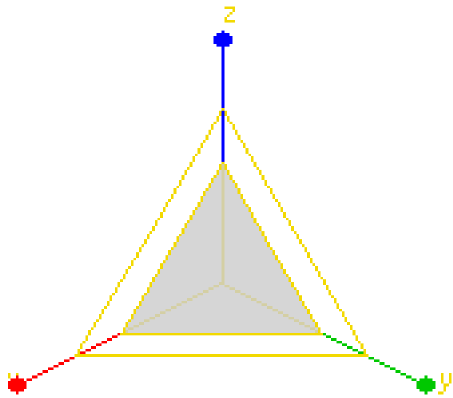
You can adjust settings for the Rotate gizmo on the Gizmos panel on page 7778 of the Preferences dialog on page 7743

Scale Gizmo

The Scale gizmo includes plane handles and scaling feedback through the stretching of the gizmo itself.

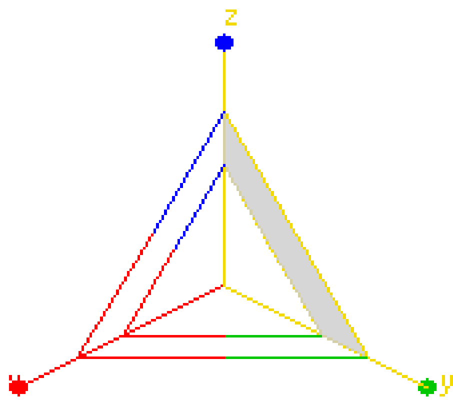
The plane handles let you perform uniform and non-uniform scaling without changing your selection on the main toolbar:

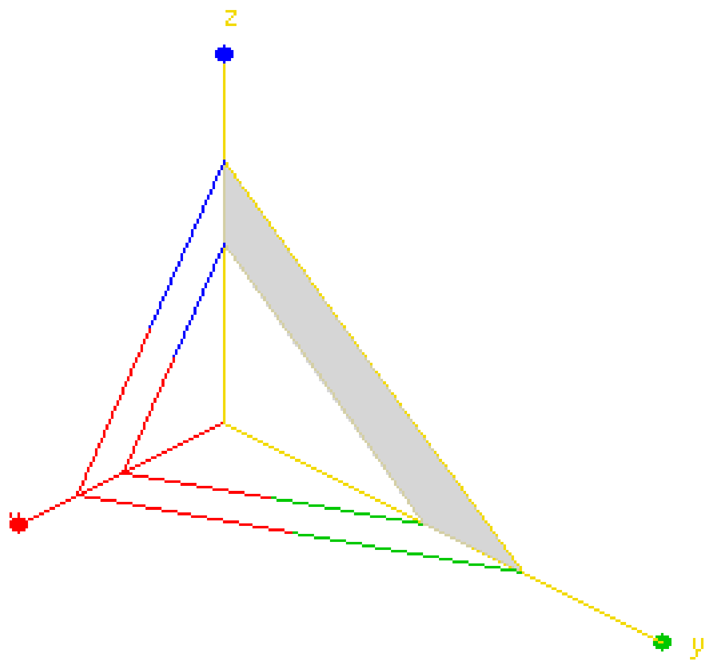
- To perform Uniform scaling, drag in the center of the gizmo.



The Transform gizmo with Uniform scaling selected.

- To perform non-uniform scaling, drag on a single axis or a plane handle.





Top: The Scale gizmo with the YZ plane handle selected

Bottom: Non-uniform scaling on the YZ plane

NOTE To perform a Squash operation, you must choose Select and Squash on page 965 on the main toolbar.

The Scale gizmo provides feedback by changing its size and shape; in the case of a uniform scale operation, it will grow or shrink as the mouse moves, and during non-uniform scaling, the gizmo will stretch and deform while dragging. However, once the mouse button is released, the gizmo returns to its original size and shape.

You can adjust settings for the Scale gizmo on the Gizmos panel on page 7778 of the Preferences dialog on page 7743

Notes

Using a Transform gizmo sets the default axis constraint to the last axis or plane you used.

If Lock Selection Set is on, you can drag anywhere in the viewport to transform the object. Dragging an axis, however, still applies the constraint along that axis.

See also:

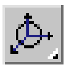
- Gizmos Preferences on page 7778

Procedures

Example: To explore use of the transform gizmo:

1.  Reset the program, then create a sphere, and then click the Select and Move button.

   The Transform gizmo appears at the center of the sphere. Because the default axis constraint on the Axis Constraints toolbar is XY Plane, the X and Y shafts of the Transform gizmo are yellow (active), while the Z shaft is blue.

2.  Use Orbit on page 7602 to adjust the Perspective view for a better view of the Transform gizmo. When you're done, right-click to return to Select And Move .

3. Point to any part of the sphere away from the Transform gizmo, and drag to confirm that the sphere is locked to the XY plane.

4. Point to the Z-axis shaft, and drag.

   The Z shaft turns yellow, the X and Y shafts turn red and green, respectively, and the sphere moves along the Z axis.

5. Point to the Y shaft, and drag.

   The Y shaft turns yellow, and the sphere moves along only the Y axis.

6. Point to the red-and-green corner mark opposite the ends of the X and Y axes, and drag.

   The sphere moves along the XY plane.

7. Press the Spacebar to turn on Selection Lock on page 7539.

8. Drag the mouse anywhere in a viewport away from the selection.

   The sphere moves along the XY plane.

9 Point to the X shaft, and drag.

The sphere moves along only the X axis.

Experiment with other transformations, such as rotation and scale. Try different reference coordinate systems. Experiment with sub-object transformations.

Interface

Change default colors Customize menu > Customize User Interface dialog > Colors panel on page 7712 > Gizmos Element > Active Transform Gizmo and Transform Gizmo X/Y/Z.

Enable/disable Transform Gizmo Customize menu > Preferences > Gizmos panel on page 7778 > On check box.

NOTE When you turn off the Transform gizmo in Preferences, the standard axis tripod appears instead. To toggle display of either the gizmo or the tripod, press the X key or use Views menu > Show Transform Gizmo.

There are additional controls for each Gizmo in the Gizmos panel on page 7778 of the Preferences dialog.

Transform Type-In

Status bar > Transform Type-In

Edit menu > Transform Type-In

F12

Main toolbar > Right-click Select And Move, Select And Rotate, or one of the Select And Scale buttons.

Transform Type-In is a dialog that lets you enter precise values for move, rotate, and scale transforms on page 8157. You can use Transform Type-In with anything that can display an axis tripod or Transform gizmo.

You can also use the Transform Type-In boxes on the status bar on page 7524. To use the Transform Type-In boxes on the status bar, simply enter the appropriate values in the boxes and press Enter to apply the transformation. You can alternate between entering absolute transform values or offset values by clicking the Relative/Absolute Transform Type-In button to the left of the transform boxes.

If you choose Transform Type-In from the Edit menu, press F12, or right-click one of the transform toolbar buttons, Transform Type-In pops up as a dialog. The title of the dialog reflects the active transform. If Rotate is active, the dialog's title is Rotate Transform Type-In and its controls affect rotation. If Scale is active, its title is Scale Transform Type-In, and so on. You can enter either absolute transform values or offset values.

In most cases, both absolute and offset transforms use the active reference coordinate system on page 967. The exceptions are View, which uses the World coordinate system, and Screen, which uses World for absolute moves and rotations. Also, absolute scaling always uses the Local coordinate system. The dialog labels change to show the reference coordinate system being used.

When you use the Transform Type-In at a sub-object level, you transform the transform gizmo of the sub-object selection. So, for example, the absolute position values represent the absolute world position of the transform gizmo. If you've selected a single vertex, it's the absolute world position of the vertex.

If multiple vertices are selected, the Transform gizmo is placed at the center of the selection, so the position you specify in the Transform Type-In sets the absolute position of the center of the selected vertices.

When multiple vertices are selected in Local transform mode, you end up with multiple transform gizmos. In this case, only the Offset control is available.

Because the axis tripods are not scaled, the Absolute Scale control is not available at the sub-object level. Only Offset is available.

When you use the Transform Type-In for Absolute rotation, the state of the Center flyout is respected. You can perform absolute rotations about the pivot point of the object, the selection center, or transform coordinate center. See Choosing a Transform Center on page 952.

Using Type-In with Sub-Object Selection

You can use Transform Type-In with any sub-object selection or gizmo. The transform affects the axis tripod for the selection.

Absolute and offset world coordinates are those of the object's or selection's coordinate system, whose origin is indicated by the axis tripod. If multiple vertices are selected, the tripod is at the center of the selection and its location is given in world coordinates.

Because axis tripods cannot be scaled, Absolute Scale fields are unavailable when you are at a sub-object level.

See Basics of Creating and Modifying Objects on page 377 for information on sub-object selection and gizmos.

Procedures

To use transform type-in:

1 Select an object or a group of objects.

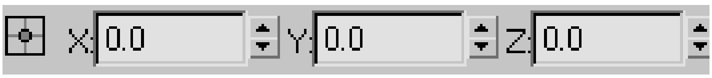
2 Choose a transform to perform on the objects (Move, Rotate, or Scale).

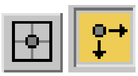
3 You can do any of the following, switching from one to another as required:

- Type a value in an axis field and press Enter to apply the transform change to the object in the viewport.
- Drag a spinner in an axis field to update the object in the viewport.
- Drag the object to apply the transform and read the resulting change in the axis fields.
  For example, if Move is active, the fields read out both the absolute positions of the selected object in world space. If no object is selected, the fields turn gray.

Interface

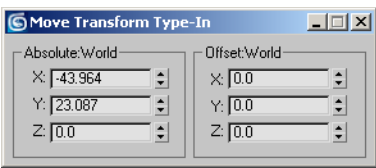
Status bar



 **Absolute/Offset Mode Transform Type-In** When this is off, the software treats values you enter into the X, Y, and Z fields as absolutes. When this is on, the software applies transform values you enter as relative to current values; that is, as an offset. Default=off.

X, Y, and Z Display and accept entry for values of position, rotation, and scale along each axis.

Absolute group (Dialog)



X, Y, and Z Display and accept entry for absolute values of position, rotation, and scale along each axis. Position and rotation are always displayed, as world scale is always local.

Offset group (Dialog)

X, Y, and Z Display and accept entry for offsets of the position, rotation, and scale values along each axis.

Displayed offset values revert to 0.0 after each operation. For example, if you enter 45 degrees in a Rotate Offset field, when you press Enter, the software rotates the object 45 degrees from its previous position, increases the Absolute field value by 45 degrees, and resets the Offset field to 0.0.

Offset labels reflect the active reference coordinate system. The Offset can be Offset: Local, Offset: Parent, and so on. If you use Pick to select the reference coordinate system of a particular object, the Offset will be named with that object.

Animating Transforms

You can animate changes in position, rotation, and scale (transforms) by turning on the Auto Key button and then performing the transform at any frame other than frame 0. This creates a key for that transform at the current frame.

Example: To animate an object moving among three points:

1. 

Turn on the Auto Key button on page 3083.

   The Auto Key button and the highlight border around the active viewport both turn red.

2. Drag the time slider to frame 25.

3. Move the object from its current position (point A) to another location (point B).

   The software creates Move keys at frames 0 and 25. These appear on the track bar on page 7531. The establishing key at frame 0 describes the object's original position, at point A. The key at frame 25 describes the object's position at point B.

4. Drag the time slider to frame 50.

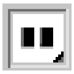
5 Move the object from point B to a third location (point C).

The software creates a Move key at frame 50 that describes the object's position at point C.

6 Click the Auto Key button to stop recording.

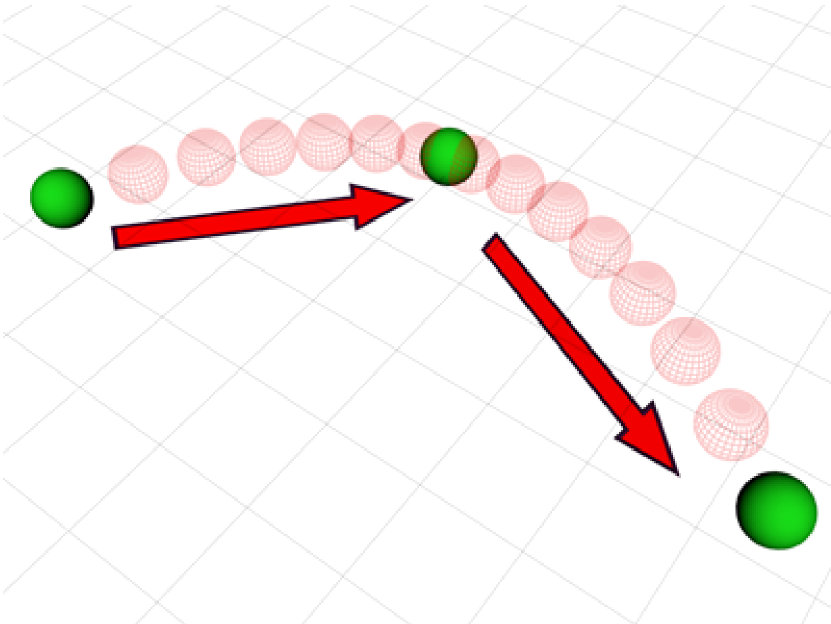
7  Click the Play button on page 7561.

The object moves from point A to point B over frames 0 to 25, and then proceeds to point C over frames 26 to 50.

8  The Play button has turned into a Stop button; click Stop to stop playback.

You can combine different transforms in a single animation sequence, so that an object appears to move as it rotates and changes in size.

See Animation Concepts and Methods on page 3078 for more information on animation techniques.



An object animated among three points

Transform Managers

3ds Max provides three controls, collectively referred to as the transform managers, for modifying the action of the transform tools.

The transform manager controls are as follows:

- The Reference Coordinate System drop-down list on page 967, which controls the orientation of the transform axes, is found to the right of the Move, Rotate, and Scale transform buttons on the main toolbar.
- The Transform Center flyout on page 975, which controls the center about which the software applies the transform, is found to the right of the Reference Coordinate System drop-down list.
- The Axis Constraint setting on page 955 lets you restrict the transform to a single axis or two axes (that is, a plane). The axis constraint tools appear on the Axis Constraints toolbar, which is off by default. You can open the toolbar by right-clicking an empty spot on the main toolbar and choosing Axis Constraints from the menu.

  TIP You can also restrict transforms with the transform gizmos on page 934.

Definitions

Certain terms are used in the description of transforms and the transform managers.

- An *axis* is a straight line along which an object is moved or scaled, or about which an object is rotated. When you work in 3D, you use three axes, labeled X, Y, and Z, which are oriented 90 degrees from each other.
- A *coordinate system* specifies the orientation of the X, Y, and Z axes used by a transform.
  For example, in the World coordinate system, as seen from the Front view, the X axis runs horizontally from left to right, the Y axis runs from back to front, and the Z axis runs vertically, from bottom to top.

  On the other hand, each object carries its own Local coordinate system. If the object has been rotated, its Local coordinate system might be different from the world coordinate system.
- The *transform center*, or pivot point, is the spot about which a rotation takes place, or to and from which scaling occurs.

Using the transform managers, you can specify any combination of axes, transform coordinate systems, and transform centers.

Axis Tripod Icon

An axis tripod appears in the viewports when you select one or more objects, to assist you visually in your transforms. This tripod consists of three lines, labeled X, Y, and Z, and shows you three things:

- The orientation of the tripod reveals the orientation of your coordinate system.
- The location of the junction of the three axis lines shows you where your transform center is.
- The highlighted red axis lines show you the axis or axes to which the transform is constrained. For example, if only the X axis line is red, you can move objects only along the X axis.

NOTE The Transform gizmo supplants the axis tripod for selections when a transform mode is active. Besides providing all of the above functions, it lets you specify the transform axis or axes without explicitly setting constraints; see Using the Axis Constraints on page 955. For more on the Transform gizmo, see Using Transform Gizmos on page 934.

You can toggle the display of the axis tripod in all viewports by choosing Views menu > Show Transform Gizmo, or by pressing the X key.

Transform Manager Settings

The state of the three transform managers (coordinate system, center, and axis constraints) is stored with each type of transform. When you switch from Move to Rotate to Scale, the transform managers change to whatever combination they were in when you last used that transform.

For example, if you click Rotate and set the transform managers to Local, Selection Center, and Y constraint, when you click Move, the controls might shift to View, Pivot Point, and XY constraint (whichever combination was set the last time you used Move). When you go back to Rotate, the controls revert to Local, Selection Center, and Y constraint.

TIP To avoid surprises, always click the transform button first, and then set the transform managers. If, instead, you first set the transform managers, their settings are likely to change as soon as you choose a new transform button. One way to remember this is always to set the transform and managers by working from left to right on the toolbar. Alternatively, you can turn on Customize menu > Preferences > General tab > Reference Coordinate System group > Constant, which keeps the transform manager settings the same for all transforms.

Specifying a Reference Coordinate System

The reference coordinate system determines the orientation of the X, Y, and Z axes used by the transform. The type of transform system you use affects all transform operations.

You specify the transform coordinate system using the Reference Coordinate System list on page 967.

Creating a Local Axis

While modeling, it's often helpful to have a temporary, movable local axis so you can rotate or scale about an arbitrary center.

NOTE This technique does not work for animation. See Choosing a Transform Center on page 952 for animation tips.

TIP As an alternative to this method, you can use the Working Pivot on page 3479.

To create an adjustable local axis:

1. Create a Point helper object on page 2628.
2. From the Transform Coordinate System list, choose Pick, and then click the point object.

   The name of the point object appears in the list as the active coordinate system.

Now you can use the point object's coordinate system as an adjustable axis.

To use the adjustable axis:

1. Place the point object where you want the rotate or scale transform to be centered.

2 Select the object you want to transform.

3 Choose the point object's name in the Transform Coordinate System drop-down list.

4 From the Use Center flyout on page 975 choose Use Transform Coordinate Center. For more information, see Choosing a Transform Center on page 952.

5 Proceed with the transform.

Choosing a Transform Center

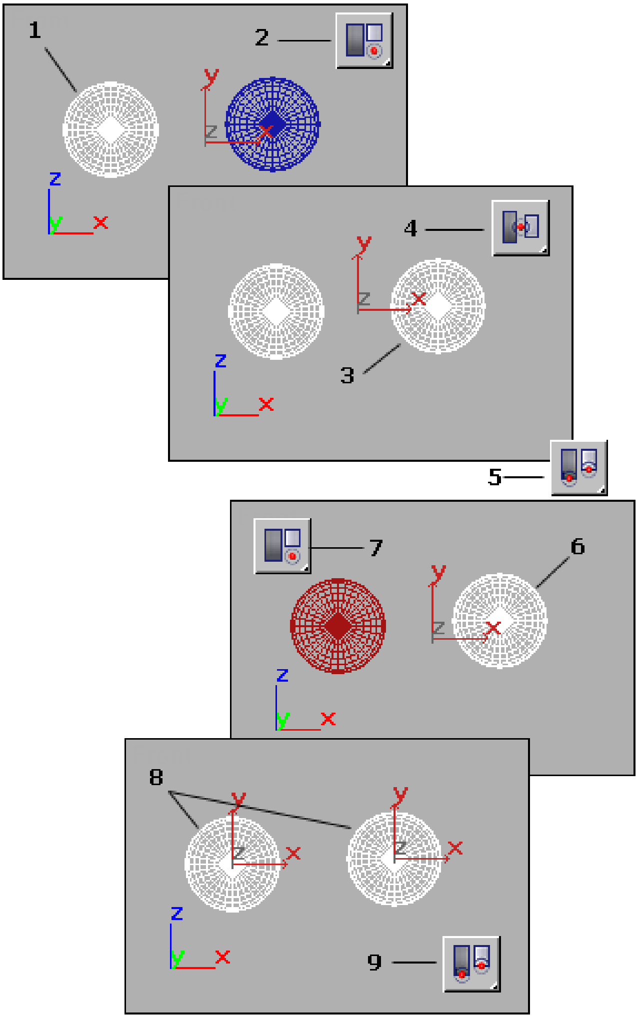
The transform center affects scale and rotation transforms, but has no effect on position transforms.

The software lets you choose from three types of transform center using the Use Center flyout on page 975 on the main toolbar. When you change the transform center, the junction of the axis tripod icon moves to the location you specify.

By default, 3ds Max sets the transform center to Use Pivot Point center for single objects. When you select multiple objects, the default transform center changes to Use Selection Center, because selection sets have no pivot point. You can change the transform center in either case, and the program remembers and restores the transform center setting separately for selections of single and multiple objects (during the current session).

TIP You can transform an object's pivot with the Hierarchy panel > Adjust Pivot controls on page 3476. Alternatively, you can transform objects using the working pivot on page 3479.

For example, you might select a single object and choose Use Transform Coordinate Center, and then select multiple objects and choose Use Pivot Point Center. When you next select a single object, the program switches back to Use Transform Coordinate Center. Then, when you select multiple objects, the center switches back to Pivot Point.



1. User selects single object.

2. User clicks Use Transform Coordinate Center from Use Center flyout on Main toolbar.

3. User adds second object to selection.

4. Transform center changes to Use Selection Center when selection set contains more than one object.

5. User clicks Use Pivot Point Center while multiple objects still selected.

6. User selects single object.

7. Transform center returns to Use Transform Coordinate Center (see step 2).

8. User selects multiple objects.

9. Transform center returns to Use Pivot Point Center (see step 5).

Transforming About Snapped Points

While the transform center choices are often useful at the object level, they are not usually convenient when transforming sub-object selections. You can override the active transform center and perform the current transform about a temporary point by using snaps. When Snaps is active, and your selection is locked, the point you snap to will set the point about which the transform is performed. Using this technique, you can:

- Move relative to two snap points.
- Rotate about a snapped point.
- Scale about a snapped point.

For more details, see Snap Settings on page 2662.

Animation and the Transform Center

Because of the nature of keyframing, you can animate rotation and scale transforms properly only by using an object's local pivot point. For example, while modeling, you can rotate an object that's offset from the world origin around the world center coordinate system. The object sweeps around the origin in a large arc. However, if you attempt to animate this, the object rotates about its local axis and moves in a straight line from one end of the arc to the other.

To avoid this discrepancy, when Auto Key on page 7549 is on and either the Rotate or Scale button is active, the Use Center flyout is unavailable and set to Use Pivot Point. When Auto Key button is off, all transforms use the center settings previously described.

You can override this behavior by turning off Local Center During Animate on page 7775 in the Animation Preferences settings.

Keep in mind that this affects only the center of the transform. The orientation of the selected transform coordinate system is still in effect.

Animating "Off-Center"

You can animate a rotation or scale about an off-center point by linking your object as the child of a dummy helper object, and then rotating or scaling the dummy.

Another technique is to offset the pivot point of your object using the Hierarchy panel.

For information about linking, dummy objects, and the Hierarchy panel, see Hierarchies on page 3331.

Using the Axis Constraints

Axis Constraints toolbar > Restrict to X, Y, Z, or a plane

Keyboard >

F5 restricts to X

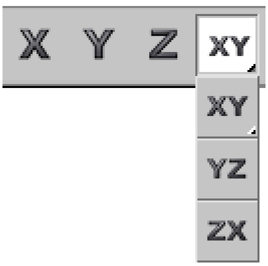
F6 restricts to Y

F7 restricts to Z

F8 cycles through the three plane restrictions

The Restrict to ... buttons, also called the Axis Constraint buttons, are located on the Axis Constraints toolbar on page 7503, which is off by default. You can turn it on by right-clicking an empty spot on the main toolbar and choosing Axis Constraints from the pop-up menu. These buttons let you specify one or two axes about or along which the transform takes place. They help you avoid transforming an object in a direction you didn't intend.

NOTE It's generally easier to use the Transform gizmos than these buttons; see Using Transform Gizmos on page 934. However, it is helpful to understand the concepts explained below.



Axis Constraint buttons

Only one axis constraint can be active at a time. When a button is turned on, transforms are constrained to the specified axis (or plane). For example, if you turn on the Restrict To X button, you can rotate an object only about the X axis of the current transform coordinate system.

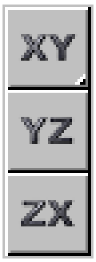
The axis or axes to which you're constrained are highlighted in red on the axis tripod icon in viewports, or in yellow on the Transform gizmo.

NOTE By default, axis constraints don't apply when using Snap. You can override this by turning on Snaps Use Axis Constraint Toggle on the Axis Constraints toolbar on page 7503, or by turning on Use Axis Constraints in Snap Options on page 2671.

NOTE Constraints are set on a transform-by-transform basis, so select the transform before you select the axis constraint. If you do not want the constraints to change, turn on Customize menu > Preferences > General tab > Reference Coordinate System group > Constant.

The axis constraints are stored separately at object and sub-object levels. If you set these three controls one way while in sub-object mode and another way while in object selection level, when you return to sub-object mode, they're restored to the way they were previously set. For example, if you're using XY constraints at object level, then switch to sub-object level and use Z constraint, when you return to object level, XY will be restored.

Restrict to Plane Flyout



The Restrict To Plane flyout, available from the Axis Constraints toolbar, lets you limit all transformations (move, rotate, scale) to the XY, YZ, or ZX planes (by default, parallel with the Top view).

You can also select planar constraint by using the Move Transform Gizmo on page 934. Instead of dragging one of the axis indicators, drag one of the plane indicators near the center of the gizmo.

When you move an object along a plane that is head-on to your view, the object moves along the single available axis shown in the view.

Reset XForm Utility

Utilities panel > Utilities rollout > Reset XForm button

Use the Reset XForm (Transform) utility to push object rotation and scaling values onto the modifier stack and align object pivot points and bounding boxes with the World coordinate system. Reset XForm removes all Rotation and Scale values from selected objects and places those transforms in an XForm modifier.

To reset the transform of a group, use the Transform button in the Reset group box of the Hierarchy > Pivot command panel.

Procedures

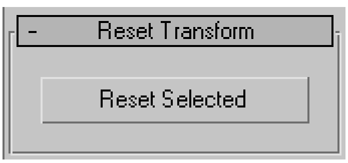
To reset an object's transform:

1. Select an object.
2. On the Utilities panel, click Reset XForm.
3. On the Utilities panel > Utilities rollout, click the More button and choose Reset XForm.
4. On the Reset Transform rollout, click Reset Selected.

   Object rotation and scaling are now carried by an XForm modifier placed at the top of the modifier stack.

   When you apply the Reset Transform utility, an XForm modifier on page 2004 that carries the rotation and scale values is placed at the top of the Modifier Stack display. You can apply other modifiers above and below the XForm modifier. You can select the XForm modifier and add other Move, Rotate, and Scale transforms. You can delete the XForm modifier to completely remove the transforms from the object. You can collapse the object to absorb the rotation and scale values into the object mesh.

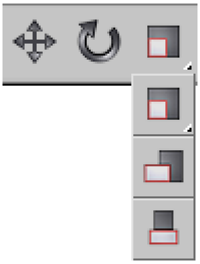
Interface



Reset selected Removes all Rotation and Scale values from selected objects and places those transforms in an XForm modifier.

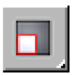
Transform Commands

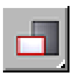
The basic transform commands are the most straightforward way to change an object's position, rotation, or scale. These commands appear on the default main toolbar on page 7499. They are also available from the default quad menu on page 7516.

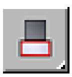


 Select and Move on page 959

 Select and Rotate on page 960

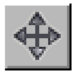
 Select and Uniform Scale on page 963

 Select and Non-Uniform Scale on page 964

 Select and Squash on page 965

See also:

- Moving, Rotating, and Scaling Objects on page 929
- Using +Clone on page 1037
- Transform Type-In on page 944

Select and Move 

Main toolbar > Select and Move

Right-click an object. > quad menu > Transform quadrant > Move

Edit menu > Select and Move

Use the Select And Move button or the Move command on the Edit or quad menu to select and move objects.

To move a single object, you do not need to select it first. When this button is active, clicking an object selects it and dragging the mouse moves it.

The direction of the movement is determined both by your mouse and by the current reference coordinate system. To restrict object movement to the X, Y, or Z axis, or to any two axes, click the appropriate button on the Axis Constraints toolbar on page 7503, use the Transform gizmo on page 934, or right-click the object, and select the constraint from the Transform submenu.

Moving an object

See also:

- Move Gizmo on page 937

Select and Rotate 

Main toolbar > Select and Rotate

Right-click an object. > quad menu > Transform quadrant > Rotate

Edit menu > Select and Rotate

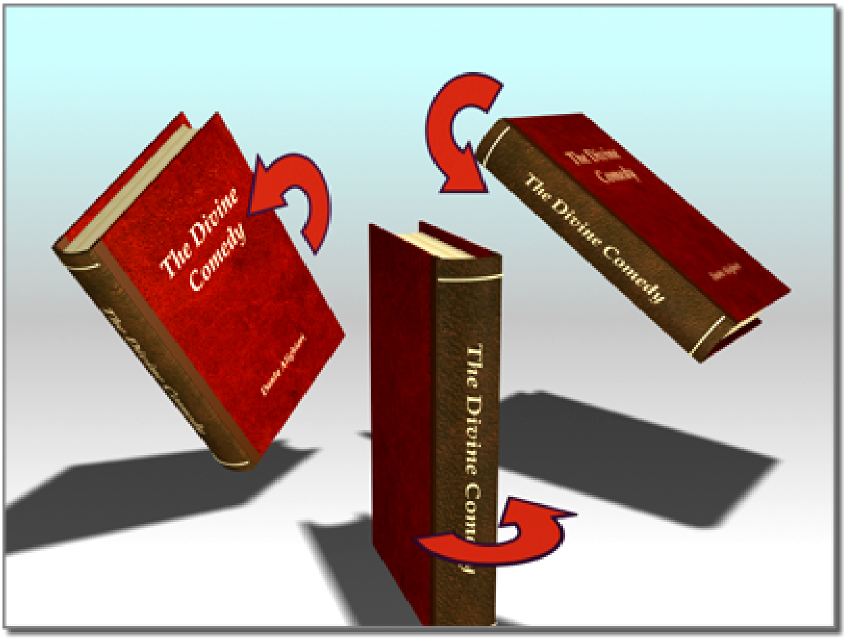
Use the Select and Rotate button or the Rotate command on the Edit or quad menu to select and rotate objects.

To rotate a single object, you don't need to select it first. When this button is active, clicking an object selects it and dragging the mouse rotates it.

When you are rotating an object about a single axis (as is usually the case), don't rotate the mouse, expecting the object to follow the mouse movement. Just move the mouse straight up and straight down. Up rotates the object one way, down rotates it the opposite way.

The center of rotation is determined by the Transform Center setting on page 967.

To restrict rotation about the X, Y, or Z axis, or to any two axes, click the appropriate button on the Axis Constraints toolbar on page 7503, use the Transform gizmo on page 934, or right-click the object, and select the constraint from the Transform submenu.



Rotating an object

See also:

- Rotate Gizmo on page 938

Procedures

This procedure illustrates the intuitive usage of the default Euler XYZ rotation controller on page 3151.

To animate object rotation interactively:

1. Add an object.
2. 

Move the time slider on page 7528 to a frame other than 0 and turn on Auto Key on page 7549.
3.  Choose Select And Rotate.
4. Rotate the object on any axis by any amount.

5 Move the time slider to a later frame.

6 Rotate the object on the same axis by an amount greater than 180 degrees.

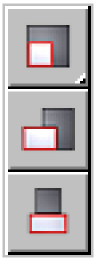
7 Play back the animation.
The rotation plays back exactly as you recorded it.

Select and Scale

Main toolbar > Select and Scale flyout

Right-click an object. > quad menu > Transform quadrant > Scale

Edit menu > Select and Scale



The Select And Scale flyout on the main toolbar provides access to three tools you can use to change object size. These are, from top to bottom:

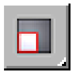
Select and Uniform Scale on page 963

Select and Non-Uniform Scale on page 964

Select and Squash on page 965

In addition, the Scale command is available on the Edit menu and the Transform quadrant of the quad (right-click) menu; this activates whichever scale tool is currently chosen in the flyout.

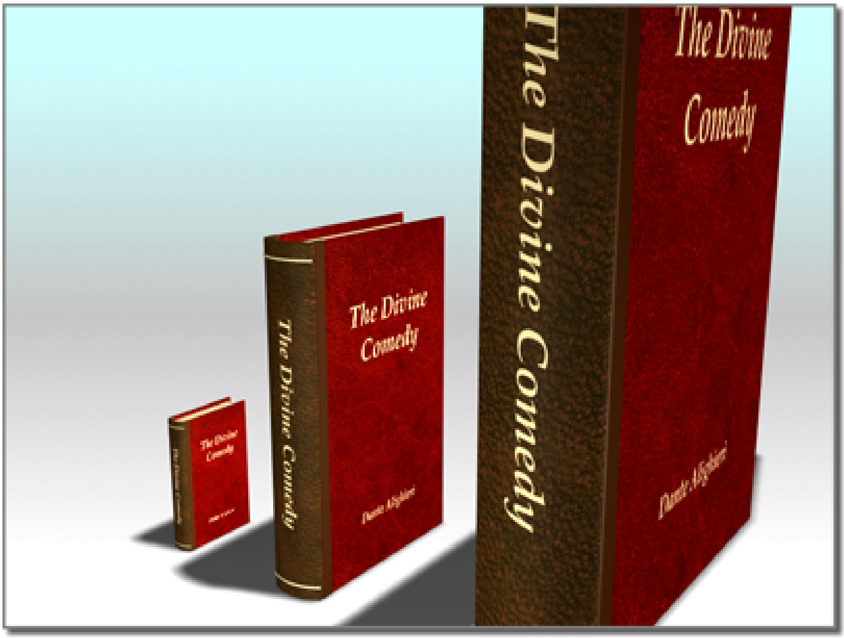
NOTE The Smart Scale command activates the Select And Scale function and, with repeated invocations, cycles through the available scaling methods. By default, Smart Scale is assigned to the R key; you can use Customize User Interface on page 7697 to assign it to a different keyboard shortcut, a menu, etc.

Select and Uniform Scale 

Main toolbar > Select and Uniform Scale (on Select And Scale flyout)

Right-click an object. > Scale (selects current toolbar Scale mode)

The Select And Uniform Scale button, available from the Select And Scale flyout on page 962, lets you scale objects by the same amount along all three axes, maintaining the object's original proportions.

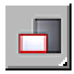


Uniform scale does not change an object's proportions.

To scale a single object, you don't need to select it first. When this tool is active, clicking an object selects it and dragging the mouse scales it.

See also:

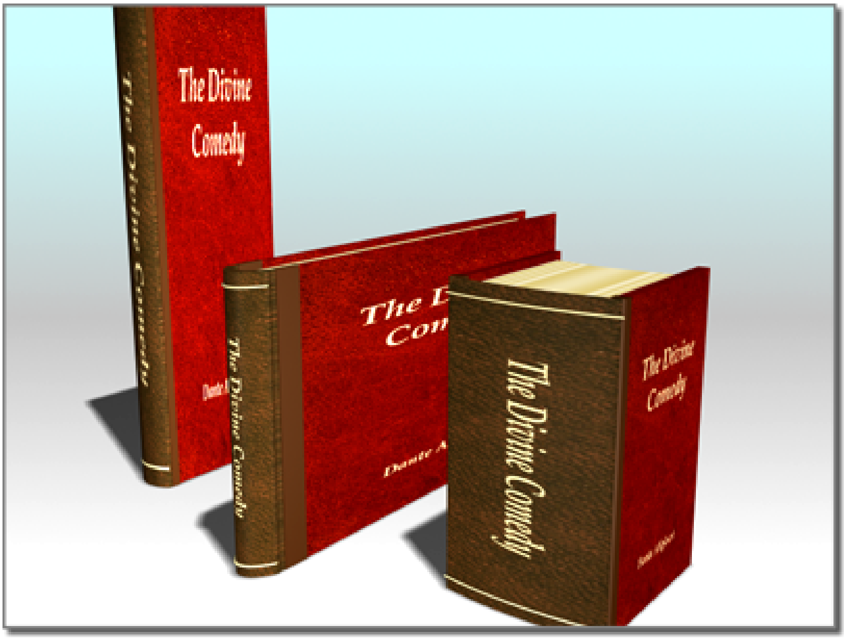
- Scale Gizmo on page 940

Select and Non-Uniform Scale 

Main toolbar > Select and Non-Uniform Scale (on Select And Scale flyout)

Right-click an object. > Scale (selects current toolbar Scale mode)

The Select And Non-Uniform Scale button, available from the Select And Scale flyout on page 962, lets you scale objects in a non-uniform manner according to the active axis constraint.



Non-uniform scale can change proportions with different values for different axes.

You can restrict the objects' scaling about the X, Y, or Z axis, or to any two axes, by first clicking the appropriate button on the Axis Constraints toolbar on page 7503, or with the Transform gizmo on page 934.

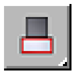
To scale a single object, you don't need to select it first. When this tool is active, clicking an object selects it and dragging the mouse scales it.

IMPORTANT Avoid applying non-uniform scale at the object level. Non-uniform scaling is applied as a transform and changes the axes of the object, so it affects other object properties. It also alters the properties passed hierarchically from parent to child. When you perform other operations on the object, such as rotation, inverse kinematic calculations, and other positioning operations, you might not get the results you expect. To recover from these problems, use the Hierarchy panel's Reset Scale button or the Utilities panel's Reset XForm utility. Either of these options will reset the axes to use the non-uniform scale as the fundamental scale for the object.

As an alternative to non-uniform scaling, consider using the XForm modifier on page 2004.

See also:

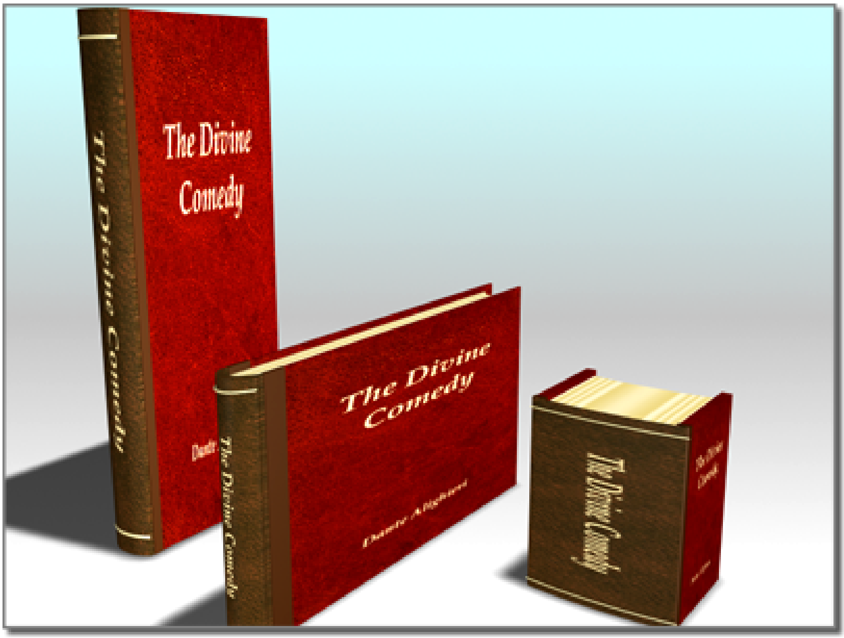
- Scale Gizmo on page 940

Select and Squash 

Main toolbar > Select And Squash (on Select And Scale flyout)

Right-click an object. > Scale (selects current toolbar Scale mode)

The Select And Squash tool is useful for creating different phases of the “squash and stretch”-style animation often found in cartoons. The Select And Squash tool, available from the Select And Scale flyout on page 962, lets you scale objects according to the active axis constraint. Squashing an object always involves scaling down on one axis while simultaneously scaling up uniformly on the other two (or vice-versa).



Squash scales two axes in opposite directions, maintaining the object's original volume.

You can restrict object scaling to the X, Y, or Z axis, or to any two axes, by first clicking the appropriate button on the Axis Constraints toolbar on page 7503.

When the Select And Squash tool is active, clicking an object selects it and dragging the mouse scales it.

IMPORTANT Avoid using Select And Squash at the object level. The non-uniform scaling that it effects is applied as a transform and changes the axes of the object, so it affects other object properties. It also alters the properties passed hierarchically from parent to child. When you perform other operations on the object, such as rotation, inverse kinematics calculations, and other positioning operations, you may not get the results you expect. To recover from these problems, use the Hierarchy panel's Reset Scale button or the Utilities panel's Reset XForm utility. Either of these options will reset the axes to use the non-uniform scale as the fundamental scale for the object.

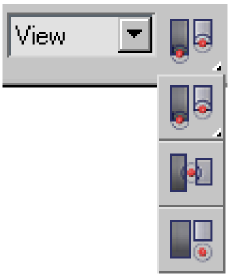
As an alternative to non-uniform scaling with Select And Squash, consider using the XForm modifier on page 2004.

See also:

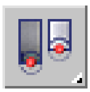
- Scale Gizmo on page 940

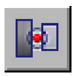
Transform Coordinates and Coordinate Center

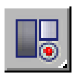
Controls for setting the coordinate system and the active center for transforms to use are on the default main toolbar on page 7499.



Reference Coordinate System on page 967

 Use Pivot Point Center on page 976

 Use Selection Center on page 978

 Use Transform Coordinate Center on page 979

See also:

- Moving, Rotating, and Scaling Objects on page 929

Reference Coordinate System

Main toolbar > Reference Coordinate System list

The Reference Coordinate System list lets you specify the coordinate system used for a transformation (Move, Rotate, and Scale). Options include View, Screen, World on page 8174, Parent, Local on page 8026, Gimbal, Grid, Working on page 3479, and Pick.

In the Screen coordinate system, all views (including perspective views) use the viewport screen coordinates.

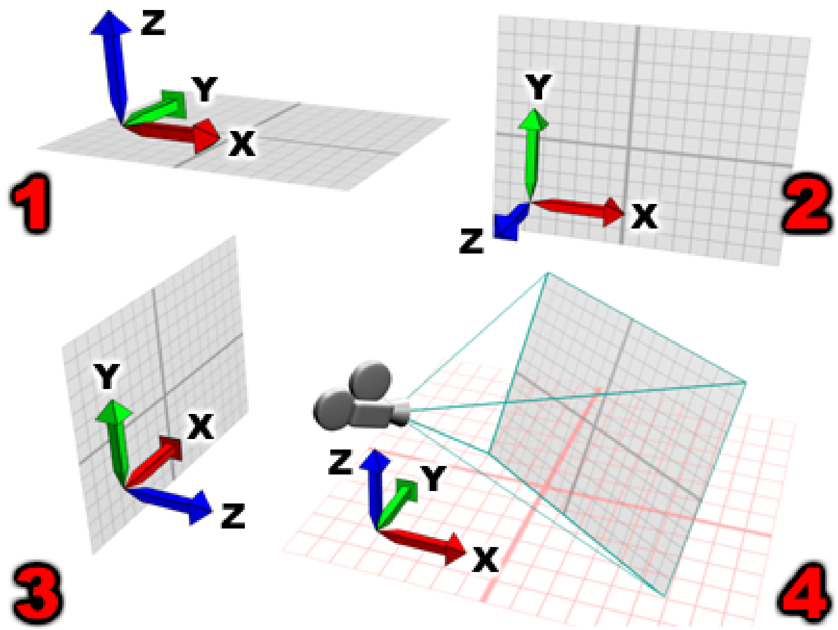
View is a hybrid of World and Screen coordinate systems. Using View, all orthographic views use the Screen coordinate system, while perspective views use the World coordinate system.

TIP The coordinate system is set on a transform-by-transform basis, so choose the transform before you specify the coordinate system. If you do not want the coordinate system to change, turn on Customize menu > Preferences > General panel > Ref. Coord. System group > Constant.

Interface

View In the default View coordinate system, X, Y, and Z axes are the same in all orthogonal viewports. When you move an object using this coordinate system, you are moving it relative to the space of the viewport.

- X always points right.
- Y always points up.
- Z always points straight out of the screen toward you.



Different orientations of the View coordinate system:

1. Top viewport.

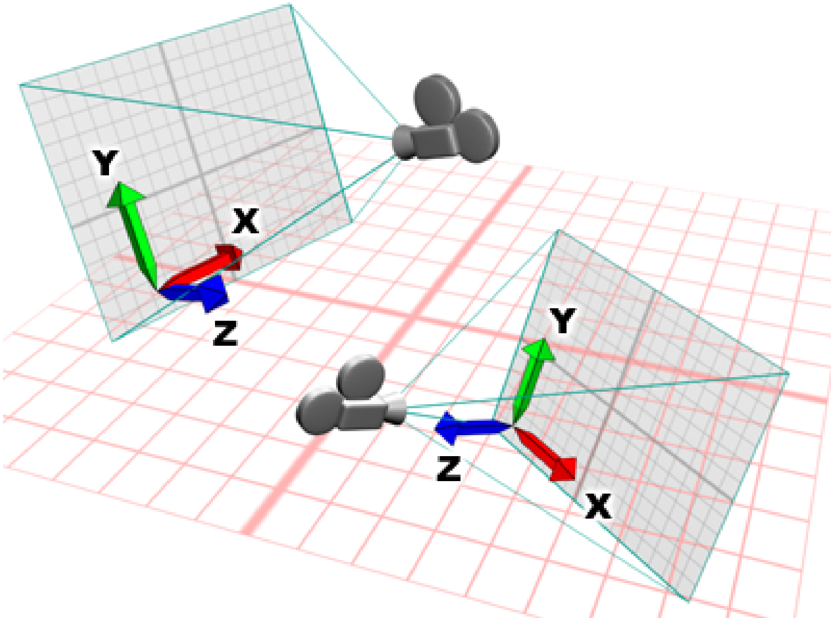
2. Front viewport.

3. Left viewport.

4. Perspective viewport.

Screen Uses the active viewport screen as the coordinate system.

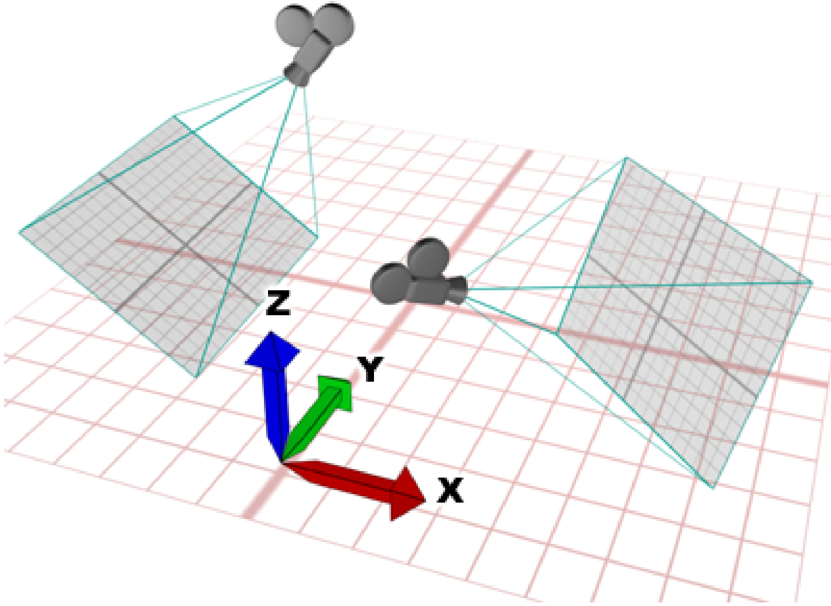
- X is horizontal, running in a positive direction toward the right.
- Y is vertical, running in a positive direction upward.
- Z is depth, running in a positive direction toward you.
  Because the Screen mode depends on the active viewport for its orientation, the X, Y, and Z labels on an axis tripod on page 950 in an inactive viewport show the orientation of the currently active viewport. The labels on that tripod change when you activate the viewport it is in.



The coordinate system in Screen mode is always relative to the point of view.

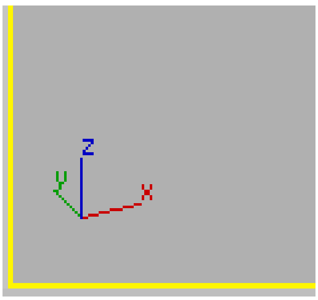
World Uses the world coordinate system. Seen from the front:

- X runs in a positive direction to the right.
- Z runs in a positive direction upward.
- Y runs in a positive direction away from you.



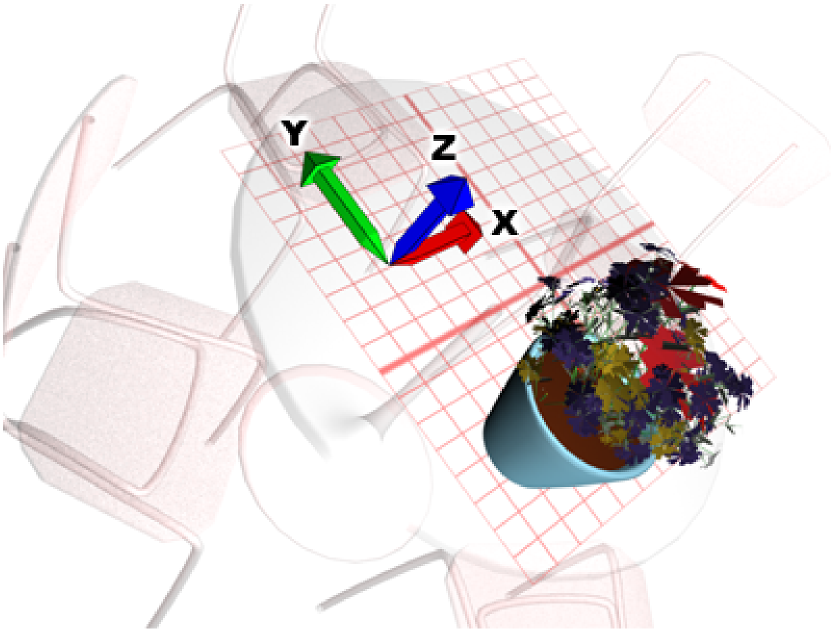
The World coordinate system is always fixed.

NOTE The world axis shows the current orientation of the viewport with respect to world coordinate system. You can find it in the lower-left corner of each viewport. The world axis colors are red for X, green for Y, and blue for Z. You can toggle the display of the world axis in all viewports by turning off Display World Axis on page 7756 on the Viewports panel of the Preference Settings dialog.



The world axis shows the current viewport orientation.

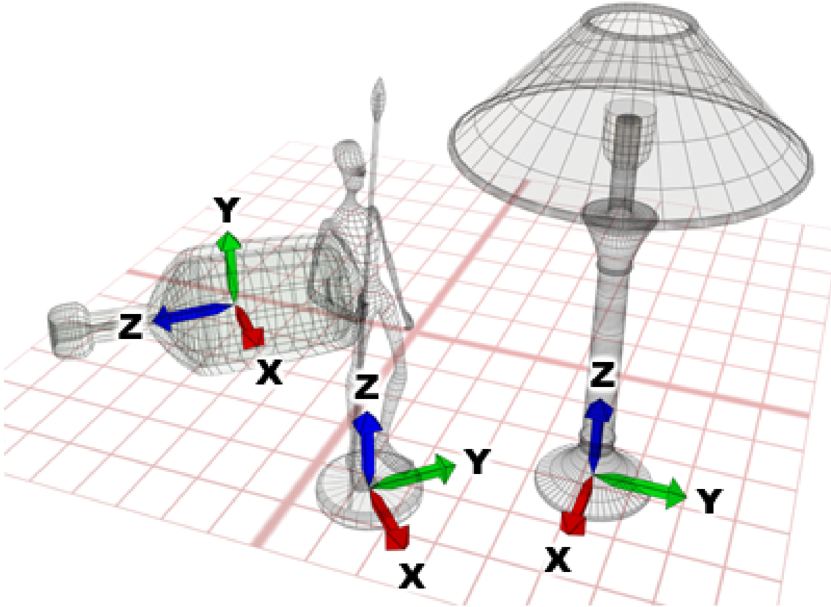
Parent Uses the coordinate system of the parent of the selected object. If the object is not linked to a specific object, it's a child of the world, and the parent coordinate system is the same as the world coordinate system.



Example of a Parent object coordinate system

Local Uses the coordinate system of the selected object. An object's local coordinate system is carried by its pivot point on page 8091. You can adjust the position and orientation of the local coordinate system, relative to its object, using the options on the Hierarchy command panel.

When Local is active, the Use Transform Center button is inactive and all transforms use the local axis as the center of transformation. In a selection set of several objects, each uses its own center for the transform.



Local uses an individual coordinate system specific to each object.

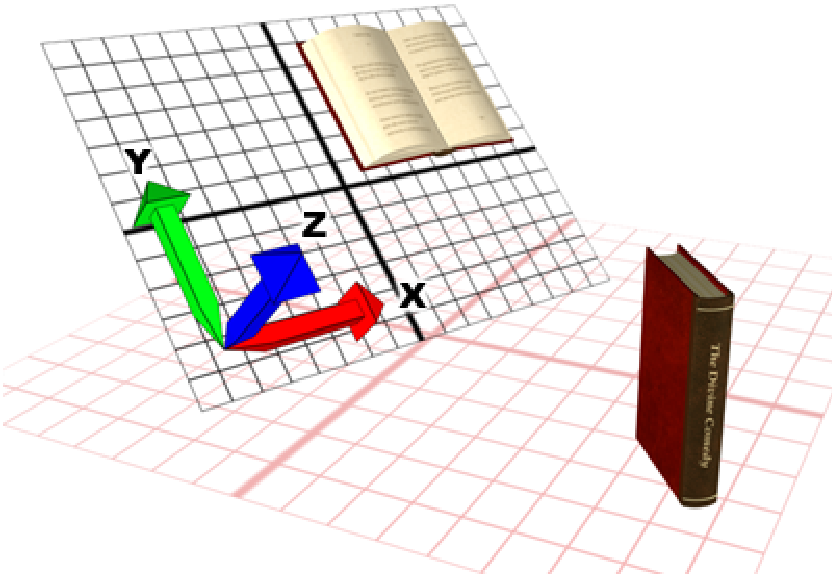
Gimbal The Gimbal coordinate system is meant to be used with the Euler XYZ Rotation controller on page 3151. It is similar to Local, but its three rotation axes are not necessarily orthogonal to each other.

When you rotate about a single axis with the Local and Parent coordinate systems, this can change two or three of the Euler XYZ tracks. The Gimbal coordinate system avoids this problem: Euler XYZ rotation about one axis changes only that axis's track. This makes function curve editing easier. Also, absolute transform type-in with Gimbal coordinates uses the same Euler angle values as the animation tracks (as opposed to Euler angles relative to the World or Parent coordinate system, as those coordinate systems require).

For move and scale transforms, Gimbal coordinates are the same as Parent coordinates. When the object does not have an Euler XYZ Rotation controller assigned, Gimbal rotation is the same as Parent rotation.

The Euler XYZ controller can be the active controller in a List controller, too.

Grid Uses the coordinate system of the active grid.



Using an active grid coordinate system.

Working Uses the coordinate system of the working pivot on page 3479. You can use this coordinate system at any time, whether or not the working pivot is active. When Use Working Pivot on page 3482 is on, this is the default coordinate system.

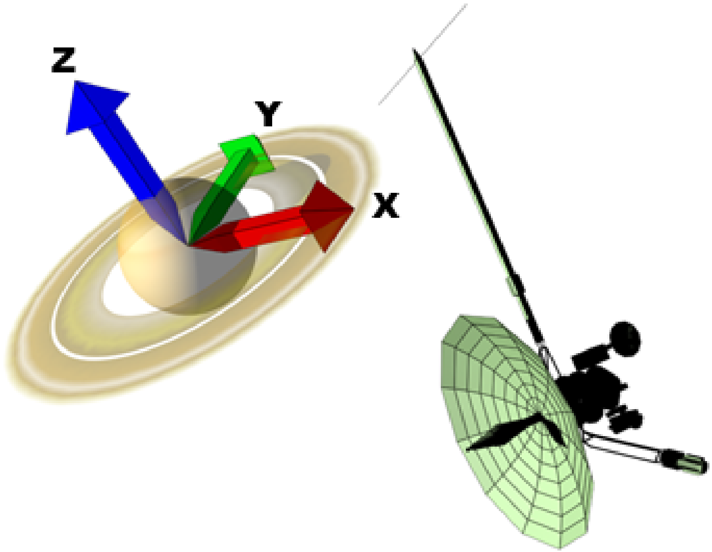
Pick Uses the coordinate system of another object in the scene.

After you choose Pick, click to select the single object whose coordinate system the transforms will use. The object's name appears in the Transform Coordinate System list.

Because the software saves an object's name in the list, you can pick an object's coordinate system, change the active coordinate system, and then use the object's coordinate system again at a later time. The list saves the four most recently picked object names.

When using Pick to specify an object as a reference coordinate system, you can press H to open the Pick Object dialog, which works like Select From Scene on page 228, and pick the object from there.

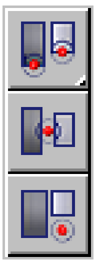
NOTE You can pick objects within an XRef scene as coordinate reference system.



Using another object as the coordinate system

Use Center Flyout

Main toolbar > Use Center flyout



The Use Center flyout provides access to three methods you can use to determine the geometric center for scale and rotate operations. They are, from top to bottom:

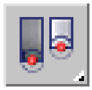
Use Pivot Point Center on page 976

Use Selection Center on page 978

Use Transform Coordinate Center on page 979

See also:

- Choosing a Transform Center on page 952

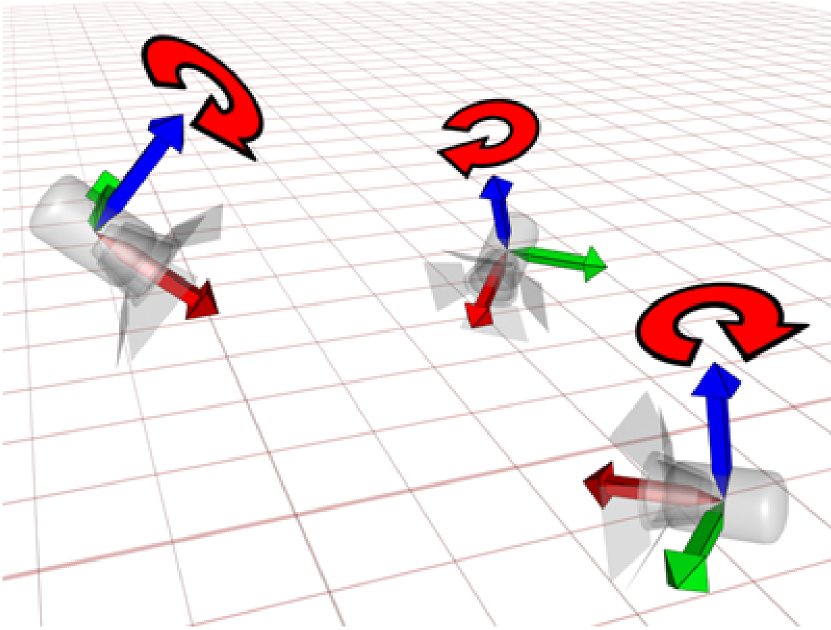
Use Pivot Point Center 

Main toolbar > Use Pivot Point Center (on Use Center flyout)

The Use Pivot Point Center option, available from the Use Center flyout on page 975, lets you enable rotation or scaling of one or more objects around their respective pivot points on page 8091. When Auto Key on page 7549 is active, Use Pivot Point Center is automatically chosen and no other option is available.

The axis tripods on page 950 show the centers that are currently being used.

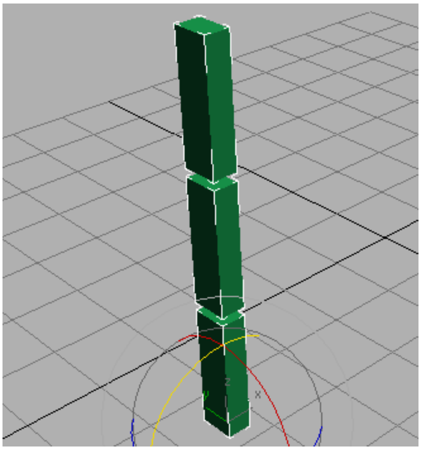
NOTE The transformation center mode is set on a transform-by-transform basis, so select the transform before you select the center mode. If you do not want the center setting to change, turn on Customize menu > Preferences > General tab > Reference Coordinate System group > Constant.



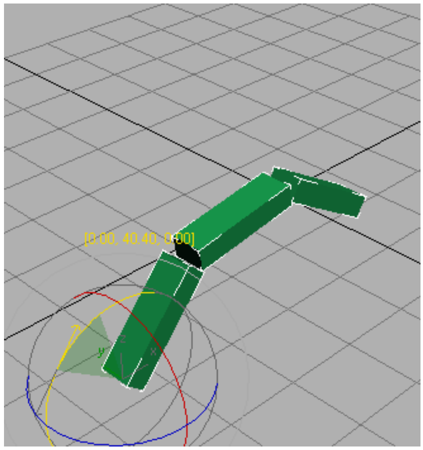
Applying a rotation with the Pivot Point rotates each object around its own local axis.

Rotating Multiple Linked Objects

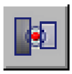
When rotating a chain of linked on page 3342 objects (that is, a hierarchy) with Use Pivot Point Center active, the rotation is applied equally to each object in the chain. This results in a accumulated rotations, which makes it easy to animate such effects as fingers curling.



Hierarchy before rotation (parent at bottom)



Hierarchy rotated

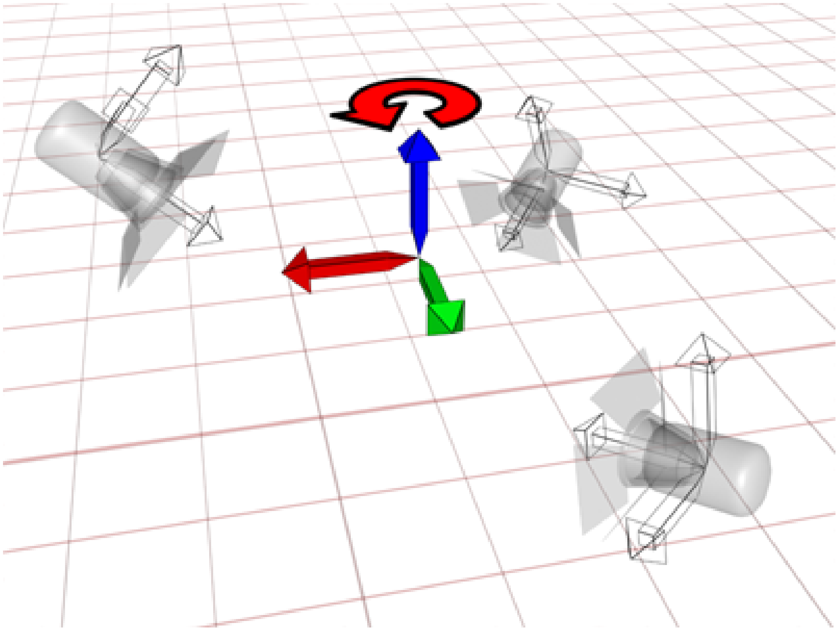
Use Selection Center 

Main toolbar > Use Selection Center (on Use Center flyout)

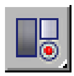
The Use Selection Center button, available from the Use Center flyout on page 975, lets you enable rotation or scaling of one or more objects around their collective geometric center. If you transform multiple objects, the software calculates the average geometric center of all the objects and uses that for the transform center.

The axis tripod on page 950 shows the center that is currently being used.

NOTE The transformation center mode is set on a transform-by-transform basis, so select the transform before you select the center mode. If you do not want the center setting to change, turn on Customize menu > Preferences > General tab > Reference Coordinate System group > Constant.



With the Selection Center option, an averaged coordinate system is used to rotate the objects.

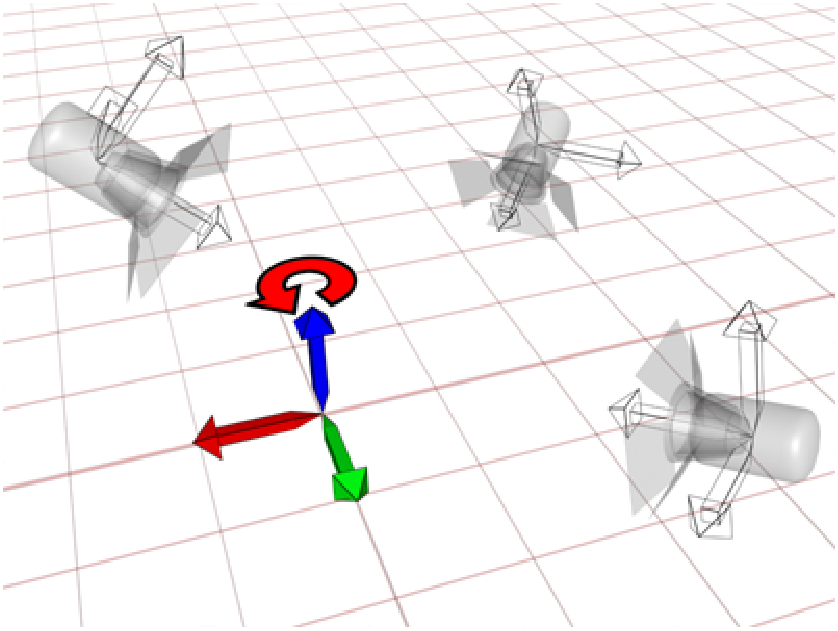
Use Transform Coordinate Center 

Main toolbar > Use Transform Coordinate Center (on Use Center flyout)

The Use Transform Coordinate Center button, available from the Use Center flyout on page 975, lets you enable rotation or scaling of an object or objects around the center of the current coordinate system. When you designate another object as the coordinate system with the Pick function (see Specifying a Reference Coordinate System on page 951), the coordinate center is the location of that object's pivot.

The axis tripod on page 950 shows the center that is currently being used.

NOTE The transformation center mode is set on a transform-by-transform basis, so select the transform before you select the center mode. If you do not want the center setting to change, turn on Customize menu > Preferences > General tab > Reference Coordinate System group > Constant.

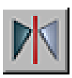


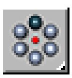
An example of the World coordinate center

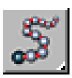
Transform Tools

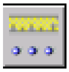
The transform tools can transform objects according to certain conditions. Some of them, such as Array, can also create copies of objects.

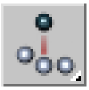
These tools (except for Array, Snapshot, Spacing Tool, and Clone And Align) are available on the default main toolbar on page 7499; the remainder are on the Extras toolbar on page 7506. Also, they all appear on the default Tools menu on page 7475.

 Mirror Selected Objects on page 982

 Array on page 986

 Snapshot on page 992

 Spacing Tool on page 996

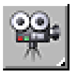
 Clone and Align Tool on page 1004

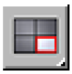
 Align on page 1009

 Quick Align on page 1015

 Normal Align on page 1015

 Place Highlight on page 1018

 Align Camera on page 1020

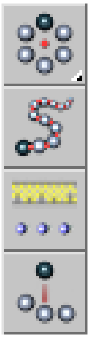
 Align to View on page 1021

See also:

- Moving, Rotating, and Scaling Objects on page 929
- Using +Clone on page 1037
- Creating Copies and Arrays on page 1023

Array Flyout

Extras toolbar > Array flyout



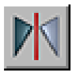
The Array flyout, available from the Extras toolbar on page 7506, provides access to various tools for creating arrays of objects. These are, from top to bottom:

Array on page 986

Snapshot on page 992

Spacing Tool on page 996

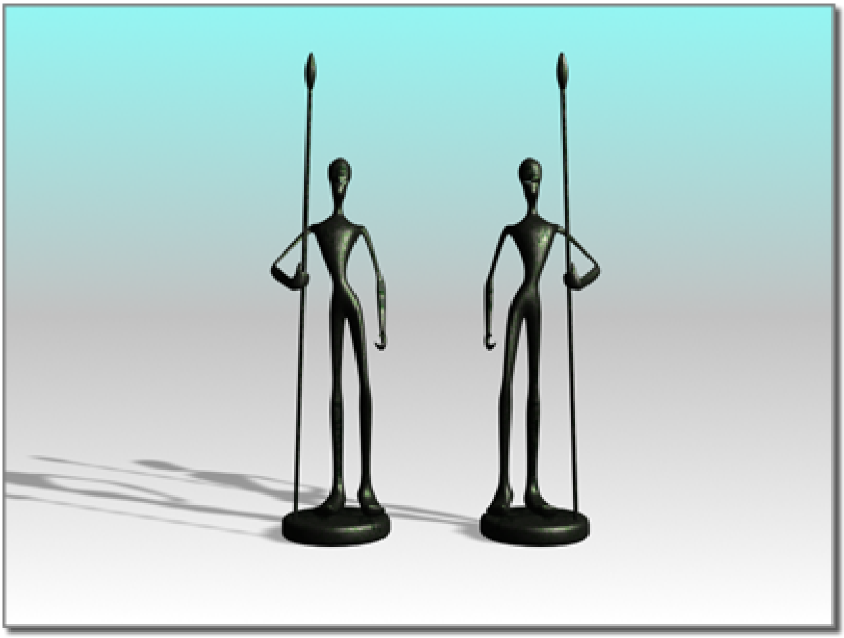
Clone and Align Tool on page 1004

Mirror Selected Objects 

Main toolbar > Mirror Selected Objects

Tools menu > Mirror

Clicking Mirror displays the Mirror dialog, which enables you to move one or more objects while mirroring their orientation. The Mirror dialog also allows you to mirror the current selection about the center of the current coordinate system. You can create a clone with the mirror dialog at the same time. If you mirror a hierarchical linkage, you have the option to mirror the IK limits.



Mirroring an object

The Mirror dialog uses the current reference coordinate system on page 967, as reflected in its name. For example, if Reference Coordinate System is set to Local, the dialog is named Mirror: Local Coordinates. There is one exception: If Reference Coordinate System is set to View, Mirror uses Screen coordinates.

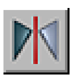
As you adjust the various settings in the Mirror dialog, you see the results in the viewports.

For more information on using Mirror, see Mirroring Objects on page 1065.

Procedures

To mirror an object:

1 Make any object selection.

2  Click Mirror on the Main toolbar, or choose Tools menu > Mirror.

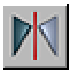
The Mirror dialog opens.

3 Set the mirror parameters in the dialog and click OK.

The active viewport changes to show the effect of each parameter as you set it. When you click OK, the software creates the choice of mirror that you see previewed.

To make a clone using mirror:

1 Make any object selection

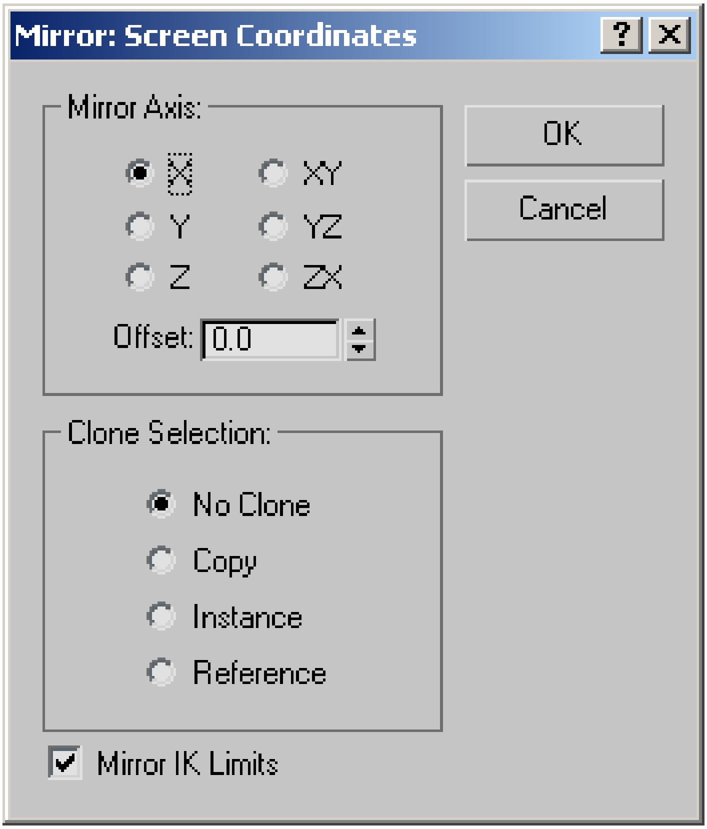
2  Click Mirror on the Main toolbar, or choose Tools menu > Mirror.

The Mirror dialog opens.

3 In the Clone Selection group, choose Copy, Instance, or Reference.

4 Make any additional settings as desired and then click OK.

Interface



Mirror Axis group

The mirror axis choices are X, Y, Z, XY, XZ, and YZ. Choose one to specify the direction of mirroring. These are equivalent to the option buttons on the Axis Constraints toolbar on page 7503.

Offset Specifies the distance of the mirrored object's pivot point on page 8091 from the original object's pivot point.

Clone Selection group

Determines the type of copy made by the Mirror function. Default is No Clone.

No Clone Mirrors the selected object without making a copy.

Copy Mirrors a copy of the selected object to the specified position.

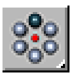
Instance Mirrors an instance on page 8014 of the selected object to the specified position.

Reference Mirrors a reference on page 8106 of the selected object to the specified position.

If you animate on page 3078 the mirror operation, mirroring generates a Scale key. If you set Offset to a value other than 0.0, mirroring also generates Position keys.

Mirror IK Limits Causes the IK constraints to be mirrored (along with the geometry) when you mirror the geometry about a single axis. Turn this off if you don't want the IK constraints to be affected by the mirror command.

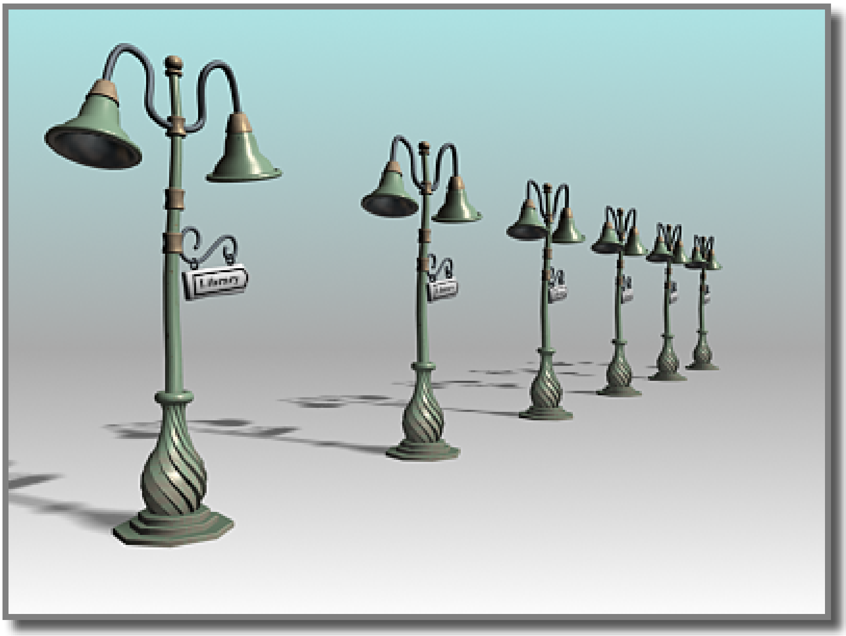
The end effectors used by the IK are not affected by the Mirror command. To successfully mirror an IK hierarchy, first delete the end effectors: Go to the Motion panel > IK Controller Parameters rollout > End Effectors group and, under Position, click the Delete button. After the mirror operation, create the new end effector using the tools on the same panel.

Array 

Extras toolbar > Array

Tools menu > Array

The Array command displays the Array dialog, which enables you to create an array of objects based on the current selection.



A one-dimensional array

The items in the Array Dimensions group let you create one-, two-, and three-dimensional arrays. For example, a row of five objects is a single-dimension array, even though it takes up three-dimensional space in the scene. An array of objects that's five rows by three columns is a two-dimensional array, and an array of objects that's five rows by three columns by two levels is a three-dimensional array.

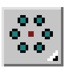
TIP You can preview the array by turning on the Preview button. With Preview on, changing the array settings updates the viewports in real time.

For more information on using Array, see Arraying Objects on page 1050.

Procedures

To create an array:

1 Select the objects to array.

2  Click the Array button, or choose Tools > Array.

3 Choose Tools > Array.

4 On the Array dialog, select the type of object to output: Copy, Instance, on page 8014 or Reference on page 8106).

5 In the Preview group, click the Preview button to turn it on.

   This lets you see the results of the array operation in the viewports, with changes appearing in real time.

6 In the Array Transformation group, click the arrows to set Incremental or Totals array parameters for Move, Rotate, and Scale.

7 Enter coordinates for the Array Transformation parameters.

8 Indicate whether you want a 1D, 2D, or 3D array.

9 Set Count to the number of copies on each axis.

10 Enter the appropriate values in the numeric fields for Incremental Row Offsets.

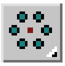
11 Click OK.

   The current selection is duplicated the specified number of times, with each object transformed as indicated.

To replace an array:

1 Undo the array to replace, using Edit > Undo Create Array, or press Ctrl+Z.

2 Change the coordinate system and transform center, if needed.

3  Click the Array button, or choose Tools > Array, and adjust any parameters on the Array dialog that is displayed.

4 Choose Tools > Array, and adjust any parameters on the Array dialog that is displayed.

5 Click OK to create a new array, which replaces the previous version. Repeat these steps to fine-tune the array.

Example: To create an array of objects that numbers 5 x 4 x 3:

1 Create a teapot with a radius of 10 units.

2 Choose Tools > Array to display the Array dialog.

3 In the Incremental set of parameters, set Move X (the upper-left field) to **50**. This causes each object in the array to be positioned 50 units apart on the X axis.

4 In the Array Dimensions group, choose the 3D button to enable all the spinners in that group.

5 Set the 1D Count spinner to **5**, the 2D Count spinner to **4**, and the 3D Count spinner to **3**.

This creates a row of 5 objects that are 50 units apart, and then 4 rows of those five objects, and then 3 rows of the 5 x 4 matrix of objects, resulting in a box array.

6 In the 2D row, set the Y spinner to **80**.

7 In the 3D row, set the Z spinner to **100**.

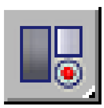
8 Click OK.

A box array of teapots appears. The first dimensional array is five teapots created along the X world axis, 50 units apart (as specified in the Array Transform group). The second dimensional array is four layers created along the Y world axis, 80 units apart (as specified in the Array Dimensions group). The third dimensional array is three layers created along the Z world axis, 100 units apart. The total number of objects in the array is 60.

Example: To create a 360-degree array:

1 Reset 3ds Max.

2 Near the top of the Front viewport (away from its center), create a long, thin box at the twelve-o'clock position (as if the viewport were a clock face).

3  From the User Center flyout on the main toolbar, choose Use Transform Coordinate Center.

4 Choose Tools > Array.

5 Click the arrow button to the right of the Rotate label to enable the three Rotate fields in the Totals section.

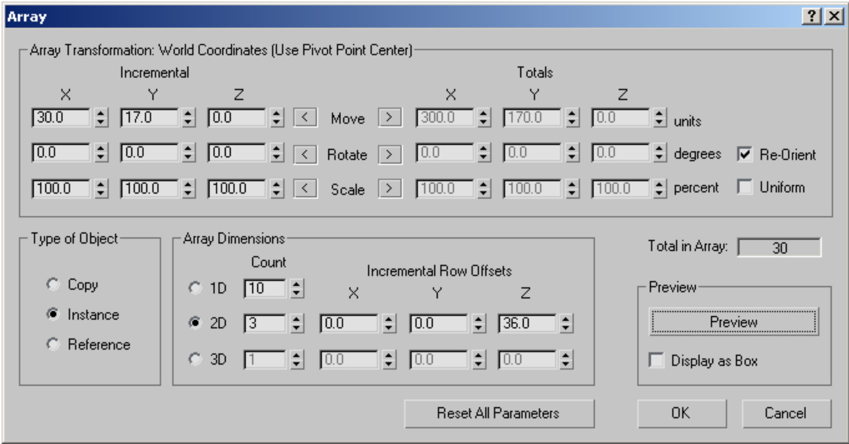
6 Set the Z parameter to **360.0**.

7 In the Array Dimensions group, choose 1D and set Count to **12**.

8 Click OK.

The software creates an array of 12 boxes in a full circle.

Interface



Array Transformation group

Specifies which combination of the three transforms to use to create the array. You also specify the extent, along the three axes, for each transform. You can specify the extent of the transform in increments between each object, or in totals for all objects. In either case, the distances are measured between the pivot points of the objects. The arrays occur using the current transform settings, so the group title changes depending on the transform settings.

Click the left or right arrow button for Move, Rotate, or Scale to indicate whether you want to set Incremental or Total array parameters.

Incremental

Move Specifies the distance between each arrayed object along the X, Y, and Z axes, in units.

Rotate Specifies the degree of rotation about any of the three axes for each object in the array, in degrees.

Scale Specifies the percentage of scale along any of the three axes for each object in the array, in percentages.

Totals

Move Specifies the overall distance, along each of the three axes, between the pivot points of the two outer objects in the resulting array. For example, if you're arraying 6 objects and set Move X total to 100, the six objects will be arrayed in a row that's 100 units between the pivot points of the two outer objects in the row.

Rotate Specifies the total degrees of rotation applied to the objects along each of the three axes. You can use this, for example, to create an array that totals 360 degrees.

Re-Orient Rotates the generated objects about their local axes while rotating them about the world coordinates. When clear, the objects maintain their original orientation.

Scale Specifies the total scale of the objects along each of the three axes.

Uniform Disables the Y and Z spinners and applies the X value to all axes, resulting in a uniform scale.

Type of Object group

Determine the type of copies made by the Array function. The default is Copy.

Copy Arrays copies of the selected object to the specified position.

Instance Arrays instances of the selected object to the specified position.

Reference Arrays references of the selected object to the specified position.

Array Dimensions group

Lets you add to the Array Transformation dimension. The additional dimensions are positional only. Rotation and scale are not used.

1D Creates a one-dimensional array, based on the settings in the Array Transformation group.

Count Specifies the total number of objects along this dimension of the array. For 1D arrays, this is the total number of objects in the array.

2D Creates a two-dimensional array.

Count Specifies the total number of objects along this second dimension of the array.

X/Y/Z Specifies the incremental offset distance along each axis of the second dimension of the array.

3D Creates a three-dimensional array.

Count Specifies the total number of objects along this third dimension of the array.

X/Y/Z Specifies the incremental offset distance along each axis of the third dimension of the array.

Total in Array Displays the total number of entities that the array operation will create, including the current selection. If you're arraying a selection set, the total number of objects will be the result of multiplying this value times the number of objects in the selection set.

Preview Toggles a viewport preview of the current array settings. Changing a setting updates the viewports immediately. If the update slows down feedback with large arrays of complex objects, turn on Display As Box.

Display as Box Displays the array-preview objects as bounding boxes instead of geometry.

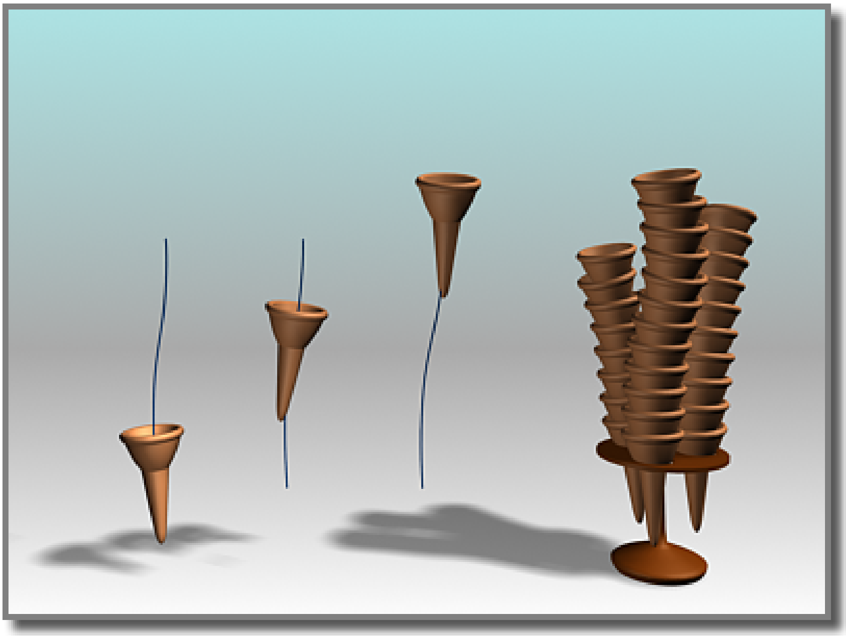
Reset All Parameters Resets all the parameters to their default settings.

Snapshot 

Extras toolbar > Snapshot (on Array flyout)

Tools menu > Snapshot

Choosing Tools > Snapshot opens the Snapshot dialog. This enables you to clone an animated object over time.



Using an ice-cream cone animated along a path, Snapshot creates a stack of cones.

Snapshot spaces the clones equally in time. Adjustments in Track View let you space the clones equally along the path instead (see the second procedure, below).

Like other clone techniques, Snapshot creates copies, instances, or references. You can also choose a mesh option for use with particle systems.

Particle Snapshots

You can clone particle systems as static mesh objects. You can also produce clones of the particles themselves as meshes, when using the Snapshot dialog > Clone Method > Mesh option. This works with all configurations of particle systems, including those using MetaParticles. Usage is the same as with other types of objects.

Procedures

To clone an object over time:

1 Select an object with an animation path.

Snapshot also shows the effect of any other transform animations, such as rotate or scale as well as parametric modifier animation.

2  Click the Snapshot button on the Extras toolbar > Array flyout, or choose Tools menu > Snapshot.

3 Set parameters in the dialog, and click OK.

To space clones evenly by distance:

1 Select an object with an animated position.

2 Open Track View and find the Position track for the original object.

3 Click Assign Controller and check that the track is using a Bezier Position controller. Do one of the following:

- If the track is already using a Bezier Position controller, proceed to step 4.

- If the track is not using a Bezier Position controller, change the controller on page 3593, then proceed to step 4.

4 Select all the transform keys and right-click one of the selected keys to display the Key Info dialog on page 3127.

5 Click Advanced to expand the dialog.

6 Click Normalize Time.

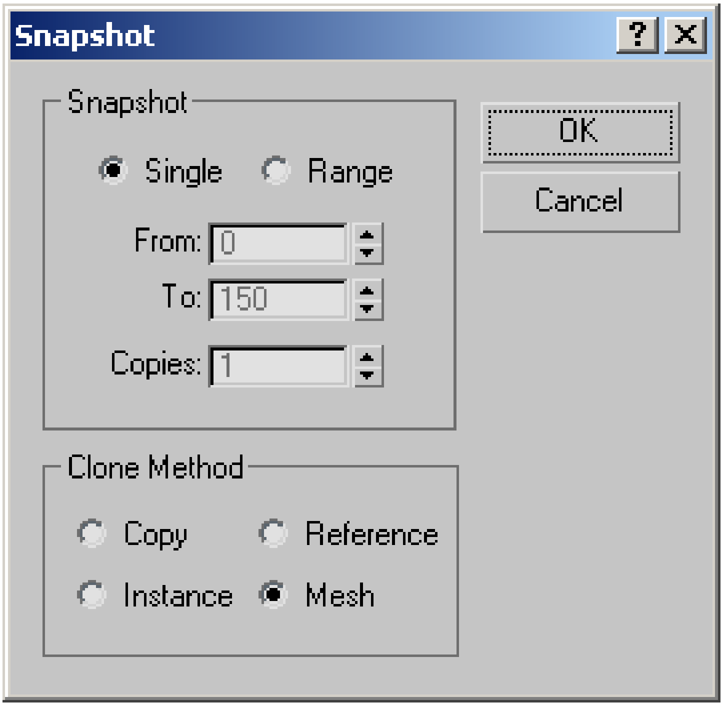
7 Set Constant Velocity on.

8 Choose Tools menu > Snapshot.

The Snapshot dialog appears.

9 Set parameters in the dialog, and click OK.

Interface



Snapshot group

Single Makes a clone of the geometry of the object at the current frame.

Range Makes clones of the geometry of the object along the trajectory over a range of frames. Specify the range with the From/To settings and the number of clones with the Copies setting.

From/To Specifies the range of frames to place the cloned object along the trajectory.

Copies Specifies the number of clones to place along the trajectory. They are evenly distributed over the time period, but not necessarily over the spatial distance along the path.

Clone Method group

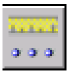
With the Copy, Instance, and Reference methods, the clone retains any animation within the object, so all the clones will be animated in the same way.

Copy Clones copies of the selected object.

Instance Clones instances on page 8014 of the selected object. Not available with particle systems.

Reference Clones references on page 8106 of the selected object. Not available with particle systems.

Mesh Use this to create mesh geometry out of particle system. Works with all kinds of particles.

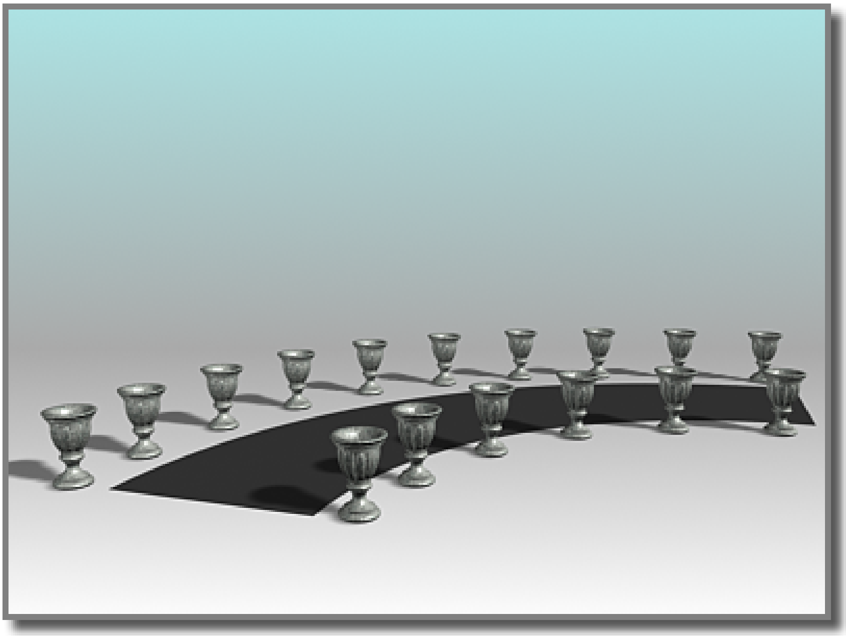
Spacing Tool 

Extras toolbar > Spacing Tool (on Array flyout)

Tools menu > Spacing Tool

The Spacing tool lets you distribute objects based on the current selection along a path defined by a spline or a pair of points.

The distributed objects can be copies, instances on page 8014, or references on page 8106 of the current selected object. You define a path by picking a spline or two points and by setting a number of parameters. You can also specify how the spacing between objects is determined and whether the pivot points of the objects align to the tangent of the spline.



The Spacing tool distributes the vases along the sides of the curved street. The vases are all the same distance from each other; fewer appear on the shorter side.

TIP You can use compound shapes containing multiple splines as the spline path for distributing objects. Before creating shapes, turn off Start New Shape on the Create panel. Then create your shapes. The software adds each spline to the current shape until you turn Start New Shape back on. When you select the compound shape so that the Spacing tool can use it as a path, objects are distributed along all of the splines of the compound shape. For example, you might find this technique useful in spacing light standards along a path defined by separated splines.

You can pick splines within an XRef scene as path reference.

For more information, see Using the Spacing Tool on page 1067.

Procedures

To distribute objects along a path:

1 Select the objects to distribute.