Carrara 7

DAZ 3D

Revision G

11 December 2008

Table of Contents

Welcome to Carrara

Welcome to DAZ 3D™ Carrara™ 7, the latest 3D program for desktop artists and animators, Web designers, and multimedia producers. Carrara is ideal for creating 3D graphics for magazines, ads, Web sites, or almost any illustration that would benefit from the powerful 3D impact of realistic perspective and shading.

Carrara 7 software's powerful animation features let you create professional quality animations and models for video, multimedia, and the Internet. With Carrara software's key-event, dynamic, and timeline-based animation, you can produce anything from flying logos to character animations. You can create broadcast quality output for square or non-square pixels, and output that is NTSC color safe.

Carrara 7 offers a complete 3D graphic development environment in one package, allowing you to create a scene, model objects that populate it, and create environmental effects. You can then animate the results if desired and, finally, render the resulting scene, using one application.

Carrara can also be an invaluable tool for interactive multimedia. It supports low-polygon modeling and popular formats such as GIF, JPEG, OBJ, 3DS, AVI, Quicklime, and sequenced movies.

Carrara software's open architecture allows new functionality to be added through extensions. The easy-to-use interface is designed around the SmartFlow workflow, guiding you through the 3D creation process, yet allowing complete creative freedom.

What's New in This Version

Carrara 7 adds a number of improvements that cover its whole range of use. Whatever you use Carrara to do, Carrara 7 will help you do it better and faster. Here is quick description of some of those improvements:

Carrara 7

- **Vertex Modeling Updates**: Vertex objects can now be modified while in the Model room without switching back and forth between rooms.

- **Content LOD Support**: Adds the ability to load and use Level of Detail meshes and control which meshes are used based on distance from the camera.

- **UV UI Enhancements** (Standard, Pro): The UV Editor has been expanded allowing editing of UVs in the main viewport.

- **UV Unfolding** (Pro Only): Adds the ability to unfold UV maps when in the UV Editor.

- **UV Relax Seams** (Standard, Pro): Adds the ability to relax UV seams when in the UV Editor.

- **3D Painting** (Pro Only): Provides the ability to paint directly on objects using color, shaders, and imported textures. Also includes real bumps and specularity display in OpenGL

- **Multi-Pass Rendering** (Pro Only): Adds multi-pass rendering to Carrara so that different components of the final render can be rendered out individually to separate images or layers.

- **COLLADA Export**: Provides export in the COLLADA file format to support the DAZ Studio COLLADA Exporter and further integration between Carrara and DAZ Studio. Currently only available for PCs.

- **COLLADA Import** (Standard, Pro): Provides the ability to import files exported out in the COLLADA file format. Currently only available for PCs.

- **Non-Linear Animation Merge Tracks**: Provides an improved interface to create groups of tracks then collapse or expand the tracks in those groups. Helps optimize the screen space so you can focus on your animation.

- **Specularity and Bump Display in OpenGL** (Pro Only): When in OpenGL mode, specularity and bump can be seen in real time on the object when supported by the video card. Changes made to the texture for bump and specularity are updated in real time.

- **Larger OpenGL Texture Display**: Improves Carrara's rendering of textured objects in OpenGL by using higher texture resolutions when supported by the hardware.

- **Rendering Optimizations for Transparency**: Time consuming and complex transparency rendering operations have been optimized to provide better and faster results.

- **DAZ Studio Content Support Improvements** - Includes enhanced the handling of .DAZ scene files being loaded into Carrara 7.0. Requires DAZ Studio Build 2.3.3.138 or higher.

Carrara 6

General

- **New Unicode Support**: Allows you to use just about any character set you want (Chinese, Japanese, French, and so on). This feature not only allows you to use these characters to name objects, but you can also use them in the Text modeler to create text in any language. Carrara 6 also gives you the ability to add any character font to the user interface so that it supports your language.
- **Multiple Selections**: Support has been extended so that you can simultaneously edit some properties of two or more selected objects in the properties panel. For example, you can hide several objects at the same time, or change the intensity of several lights together, increasing your productivity.
- **OpenGL Improvements**: An improved engine that supports texture maps with alpha channels, and improved textured mode display. OpenGL performance has also been improved in the Vertex modeler to allow editing of larger meshes.

Animation

- **New Euler Angles for Rotations**: Until now, Carrara only supported Quaternion interpolation for rotation, which did not allow you to edit the three rotation angles independently. Carrara's new "Euler angle" interpolation mode adds even more control for character animation.
- **New Hair/Fur Modeling**: This major new feature adds a completely new hair primitive and a powerful Hair modeler and simulator. You can apply hair to an object, brush it, and style it. The hair can also be shaded and have various effects such as clumping or freezing.
- **New Hair Simulation**: In addition to creating hair, another major new feature is the addition of hair simulation, which allows you to simulate the motion of the hair. Hairs interact with other objects and can have forces applied to them.
- **Rigid Body Simulation**: Carrara's physics engine has been improved. Simulations now perform up to 10 times faster and are more stable. The workflow has been improved and includes a button that allows simulation on demand and avoids constant recalculation of the physics.
- **Non-Linear Animation**: Another major new feature in Carrara 6, this new engine completes the Carrara animation toolset. Create clips of animation that you can reuse and combine on multiple tracks. Blend clips together, or use a single clip (such as a walk cycle clip) to build a longer animation (such as a character walking down a road). Create a walking character in one layer, and then add another separate layer that makes him wave. The two individual animation layers are combined into one fluid animated result.

Content Support

- **Figure Support**: DAZ content is now fully supported in Carrara. This includes better support for skinning and morph targets. Manipulation is up to ten times faster. You can select a bone simply by clicking the geometry of the figure. Properties associated with a body part are displayed with its associated bone, and all parameters are supported.
- **Content Browser**: You can now use a content browser to browse through and select DAZ content to add to your scene. Drag and drop figures, clothing, props, and poses into your scene and apply them to your figures.
- **Conforming**: Conforming figures are now supported, including morph crosstalk.

Nature Modeling

- **New Scene Scaling**: You can now select the size of the scene when you create a new scene. Camera speed, the size of the working box, and the default size of the primitives are adjusted accordingly. This makes it easier to create large landscapes in real-world scale, and also increases realism with atmospheric effects such as clouds and atmospheric blues.

    NOTE! The default measurement unit in Carrara is feet. If you want to work in inches, type "**in**" after you enter a numerical value of measurement.

- **New Landscape Wizard**: Use the new landscape wizard to create many different landscapes. Combine terrains, shading, sky, and rendering presets. This wizard makes it easy to create a beautiful landscape scene that you can use as a starting point for your projects.
- **New Ocean Primitive**: Allows you to realistically simulate the surface of the ocean. You can create still and animated oceans with high realism, with complete control over the size of the waves or the speed and direction of the wind.
- **New Terrain Features**: To add even more detail on terrains, you can use the render time displacement features to apply a displacement shader to the surface. The displacement is calculated at render time, thereby allowing you to create incredibly detailed terrains without adding extra geometry that uses more memory.

Vertex Modeling

- **New Displacement Modeling**: This new modeler allows you to paint displacement on your objects, adding complex detail to your object without adding extra geometry.
- **New Symmetrical Modeling**: This new feature allows you to edit both sides of a symmetrical object at the same time, and is ideal for character editing. Most of the modeling tools support this feature, thereby cutting your modeling time in half.
- **New Edge Extrusion**: The Edge Extrusion tool lets you extrude corners of objects interactively.
- **New Figure Editing**: You can now edit the mesh of an imported figure, as well as create new morph targets and full body morphs within the Vertex modeler.
- **Vertex modeler Optimizations**: A number of optimizations improve performance on larger meshes. You'll see improvements when you move, scale, or rotate points. Other noticeable optimizations are in OpenGL redraw speed and subdivision surface updates.

Shading and Lighting

- **New Transparency with Absorption**: Transparencies now support realistic absorption functions. This feature is ideal for simulating shallow water.
- **New Single Scattering**: This new feature is used to simulate light from small particles that are inside a medium. For example, you can use it to simulate steam in the atmosphere. It is an indirect lighting effect that can also be used to simulate water more accurately.

Rendering

- **Blurry Reflections Optimized**: Speed and quality of blurry reflections have been optimized when multiple blurry reflections are used, or when they are combined with soft shadows or raytraced depth of field.
- **Depth of Field Improvements**: Speed and quality of raytraced depth of field is optimized when used in conjunction with multiple blurry reflections or soft shadows.
- **New Shadow Bias**: A new parameter is included in raytraced shadows. It corrects self-shadowing effects due to the low resolution of an object. Shadows are now much cleaner.

Import/Export

- **New DirectX Import/Export**: Import and export geometry, materials, skinning, and animation in DirectX format.
- **New DAZ Studio Import**: Import complex scenes from DAZ Studio into Carrara. You can then manipulate the figures and props directly within Carrara.
- **Discontinued Formats**: Carrara 6 no longer imports Amapi, DWG, OpenInventor/ VRML, or STL formats.

Installation

NOTE! See "Setting Imaging and Scratch Disk Preferences" for information on how Carrara uses the hard disk.

Macintosh Installation

System Requirements:

Carrara works on any Power Macintosh® with at least the following configuration:

- Mac®. Intel or Power Macintosh® 700 MHz (1 GHz or faster recommended).

- Mac®OS X 10.3.9 or above.

- 512 MB RAM minimum (1 GB recommended).

- OpenGL compatible graphics card.

- 300 MB free hard drive space for installation (500 recommended)

- 32-bit graphics color depth.

- CD-ROM drive.

NOTE! Will not function installed on a UNIX partition under Mac OS X.

To install your Carrara Software (Download version):

1. Turn off any virus protection and compression programs and close any other applications currently open.

2. Double-click on the installer icon you downloaded from your DAZ 3D account and follow the instructions on the screen.

To install your Carrara Software (CD-ROM version):

1. Turn off any virus protection and compression programs and close any other applications currently open.

2. Insert the Carrara 6 application CD-ROM into your CD-ROM drive.

3. Double-click the Installer icon from the CD-ROM and follow the instructions on the screen.

Windows Installation

System Requirements:

- Carrara works on any PC-compatible computer with at least the following configuration:

- Pentium III processor, 700 MHz (1 GHz or faster recommended).

- Windows 2000® (Service Pack 2), Windows XP®, Windows Vista®.

- 512 MB RAM minimum (1 GB recommended).

- OpenGL compatible graphics card.

- 300 MB hard drive space for installation (500 recommended).

- 32-bit graphics color depth.

- CD-ROM drive.

NOTE! For information on how your Carrara software uses the hard disk see "Setting Imaging and Scratch Disk Preferences."

To install your Carrara Software (Download version):

1. Turn off any virus protection and compression programs and close any other applications currently open.

2. Double-click on the installer icon you downloaded from your DAZ 3D account.

3. Follow the instructions on the screen. Click Next to confirm each option and move to the next screen.

4. After your software has finished installing, click Finish.

To install your Carrara software (CD-ROM version):

1. Insert the Carrara 6 application CD-ROM into your CD-ROM drive and click the Install button. Windows locates the installation program on the CD-ROM.

2. Follow the instructions on the screen. Click Next to confirm each option and move to the next screen.

3. After your software has finished installing, click Finish.

About Your Documentation

This manual provides all the information you need to get the most out of Carrara 6. It will help you learn the application and serve as a reference as you attempt more sophisticated procedures and effects.

This guide assumes you are already familiar with basic Macintosh and Windows concepts—menus, dialogs, and mouse operations such as clicking and dragging. If you need more information on these subjects, or on the Macintosh Finder or the Windows Desktop, refer to the Apple Macintosh User's Manual or the Microsoft Windows User's Guide, respectively.

Computer graphics and 3D modeling use special terms usually explained where introduced. In addition, the Glossary gives concise definitions of many terms.

Conventions

This Guide is for both Macintosh and Windows. By convention, Macintosh commands precede Windows commands in the text. For example:

- **Command/Ctrl+I** is equivalent to the Macintosh **Command-I** and the Windows **Ctrl+I.**

- For simplicity, the term **folder** refers to directories as well as folders. The Carrara interface for both platforms is identical unless otherwise specified.

There are also several conventions used to identify paths to certain tools and controls:

- The convention for a **menu** follows the rule of the **menu name> menu item.**

- The convention for a **palette** or **tray** is **palette name: button name: palette panel: palette item.**

Version Content

This manual contains features for all versions of Carrara: 3D Express, Standard, and Pro. Many features are included in 3D Express but where a feature is available in both the Standard and Pro versions, or only the Pro version, that feature will have a note added to it.

For example, the following list has Features 1 and 2 in all versions, Feature 3 in the Standard and Pro versions, and Feature 4 only in the Pro version.:

- Feature 1

- Feature 2

- Feature 3 (Standard, Pro)

- Feature 4 (Pro only)

Technical Support

You will find the answers to most of your questions within the documentation. If you need further assistance, you can contact DAZ 3D's Technical Support for Carrara in the following ways:

Web Support

Many of the answers to your questions are available 24 hours a day on our Web site:

http://www.daz3d.com/support

In addition to frequently asked questions (FAQs), the Web site provides troubleshooting techniques, late breaking product news, and other resources to help you get the most out of your Carrara product.

Phone Support

Phone support is available to registered users of Carrara for questions about installation, configuration, or functionality. For more information about phone support, check our Web site at:

http://www.daz3d.com/support

Creative Support

You can also refer to forums on the DAZ 3D Web site for questions about creative techniques, or visit:

http://www.daz3d.com/i.x/community/0/-/?

Using Tool Tips

Carrara displays Tool Tips by default whenever you rest your cursor over an interface element.

To turn off Tool Tips:

1. Choose **File > Preferences**.
2. Choose General from the pop-up menu.
3. Uncheck the **Show ToolTips** option.

Carrara Overview

This chapter introduces some of the major features and concepts of DAZ 3D Carrara 6. With Carrara, you can create dazzling 3D illustrations, animations, and Web output in six easy steps:

- Create a scene using object primitives.

- Build true three-dimensional objects with easy-to-use, intuitive modeling tools. (Standard, Pro)

- Apply colors and textures on your objects, giving them realistic properties such as transparency and reflection.

- Light your scene to enhance realism and depth.

- Create animations of your scene.

- Render your scene, animation, or Web output with Carrara's powerful rendering engines to produce extraordinary, Photorealistic illustrations or animations.

Carrara combines power with an easy-to-use, intuitive user interface.

Creating Scenes

A Carrara document is called a **scene**. A scene is the collection of objects, light sources, and cameras, saved together in a file. Each new scene has several windows associated with the different rooms. As you create your scene in Carrara, you'll work in the various rooms to create and place the various elements that you want to use.

Before you can create a new 3D illustration, you must create a new document. A new document starts with a blank scene.

Using the Scene Wizard

The **Scene Wizard** is a collection of useful basic scenes that can be used as a starting point to create a scene. The Scene Wizard also contains some demonstration scenes that show particular features of Carrara.

To open the Scene Wizard:

1. Choose **File > New.**

2. In the New Document window, click the **Presets** button. The **Scene Wizard** opens.

3. Highlight the scene that you want to use and click **OK**.

Using the Landscape Wizard

The **Landscape Wizard** allows you to quickly and easily create a landscape, complete with shaders and lighting.

To use the Landscape Wizard:

1. Choose **File > New**.

2. Click the **Landscape Wizard** button. The **Landscape Creation Wizard** opens.
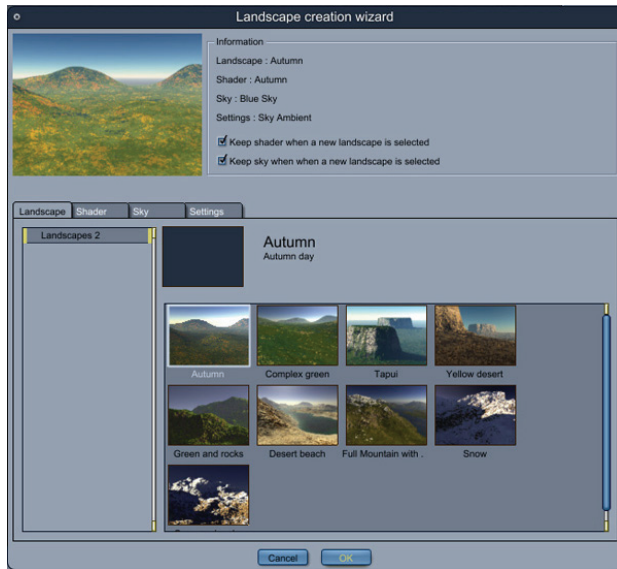
3. Check or uncheck the options to keep shader and sky settings while you make your changes.

4. Use the **Landscape**, **Shader**, and **Sky** tabs to choose the terrain, materials, and sky properties that you want to use in your new scene.

5. Use the **Settings** tab to choose illumination and shadow settings for your scene. The settings provide options ranging from fast renders with less detail to highly detailed renders that take longer.

6. Click **OK** to create your landscape.

Creating an Empty Scene

When you create an empty scene in Carrara, the document window opens in the Assemble room, using your default workspace setup. Your scene contains only one default light and a default camera.

To create an empty scene:

1. Choose **File > New**. The **New Document dialog** appears.

2. Select the size of the scene you want to create: **Small (30 in), Medium (30 ft)** or **Large (3000 ft).**

3. Select **Empty Scene**.

NOTE! The default measurement unit in Carrara is **feet**. If you want to work in inches, type "**in**" after you enter a numerical value of measurement.

Opening an Existing File

You can open any file you created in Carrara, and can also open files saved in many other common 3D formats.

To open an existing file:

1. Choose **File > Open**. The **Open dialog** appears.

2. Locate the file using the dialog controls and click **Open**.

Additional supported file types can be opened into the current scene file by choosing **File > Open**. To open other file types as a new document, refer to "Importing 3D Objects."

Carrara's Rooms

Carrara provides five different rooms for working on your scene. Each room allows you to focus on designing a particular part of your scene. You don't need to use multiple software applications to create professional quality images or animations. Carrara allows you to seamlessly switch rooms as you work so that you have easy access to the tools you need while you create your scene.

You can easily switch rooms from Assemble to Storyboard, Texture, Render, or one of the modelers, by clicking the Room buttons in the upper-right portion of your screen. As you change rooms, you are still working on the same document. The room button for the current room is highlighted, and the name of the room displays next to the buttons.

Use the Room buttons to switch between Carrara's rooms.

Assemble Room

Carrara software's SmartFlow workspace is designed to support your workflow. You can start by placing objects in your scene in a number of different ways:
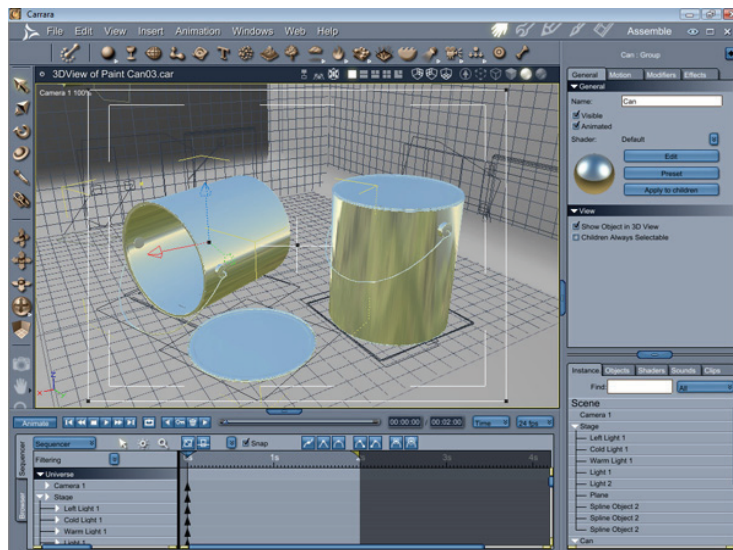
Working on your scene in the Assemble Room.

*   Drag content from the Browser Tray into your scene.

*   Use the primitive modeling tools to add content to your scene

*   Create new, original models using Carrara's different modelers (Standard, Pro)

At any time, you can position multiple light sources and cameras, and shift between cameras to gain different perspectives on your work.

Carrara supports multiple scene files, allowing you to easily cut and paste information between files.

For further information about the Assemble Room and its various controls, see "Assemble Room Overview."

Model Room

Depending on the version you have, Carrara software's modelers use many of the tools standard to 2D drawing programs and feature a variety of time saving utilities and tools, so you can create complex spline-based, polymesh, and Metaball objects quickly and easily.

There are several different modelers in Carrara depending on the version you have. The appropriate modeler opens in the **Model room** when you add one of the many different types of primitive objects to your scene.

When you double-click a primitive object in the Assemble Room, Carrara switches to the appropriate modeler where you can model them into finished objects.

Carrara Standard and Pro offer the following main types of modelers to create or edit objects:

- Primitive modeler
- Vertex modeler
- Spline modeler
- Metaball modeler
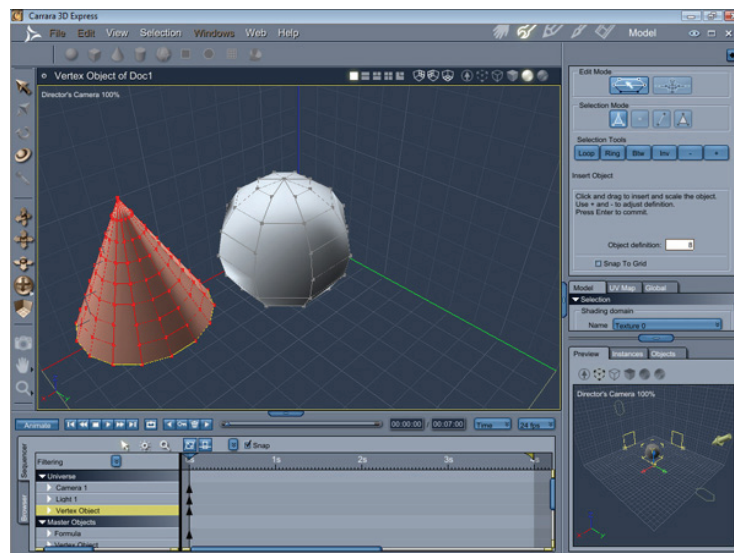
There are also other primitive objects that open the Model room when you edit or add them to your scene:

- Text
- Particle Emitters
- Terrains (shown below)
- Plants
- Volumetric Clouds

- Old Volumetric Clouds

- Surface Replicators

- Replicators

- Hair

- Ocean

- Displacement modeler (Pro only)

NOTE! Carrara 3D Express only provides terrains and plants.
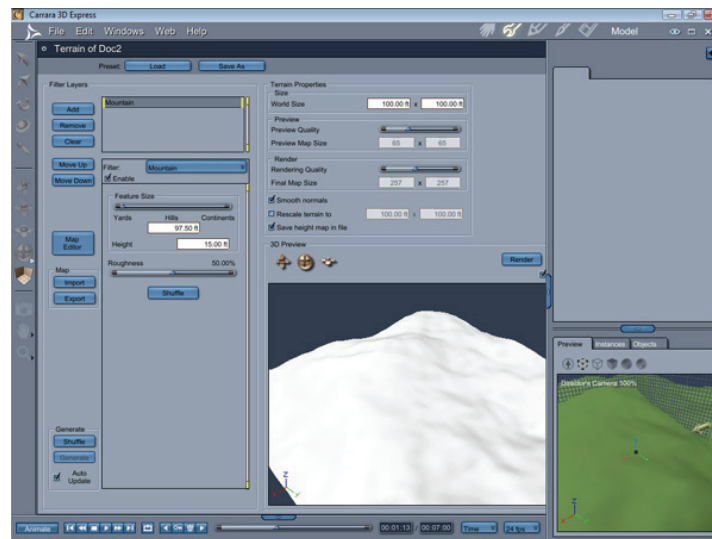
Finally, you can edit properties of the following types of objects in the Assembly Room or the Model room:

- Fire

- Clouds

- Fountain

- Formulas
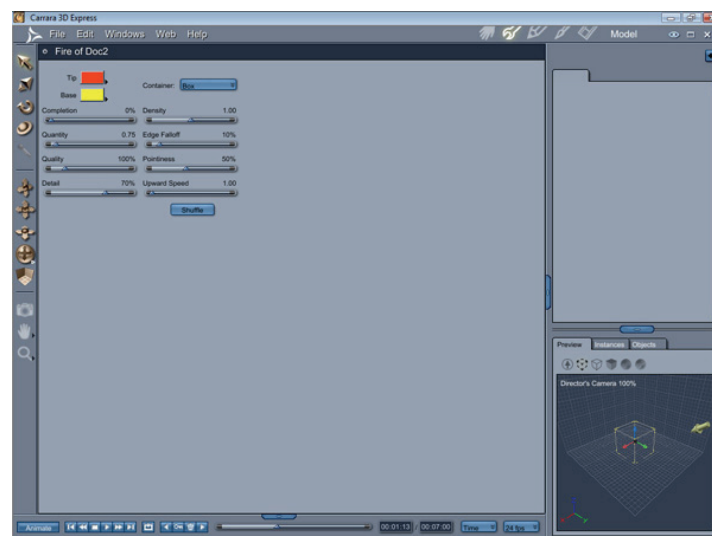
You can edit other primitives in the **Assemble room** or the **Model room**.

To create the objects that go into the scene, you must use a modeler. The Model room provides different modelers for various types of objects, including Vertex and Text. Each modeler has its own window, with individual tools and menu commands.
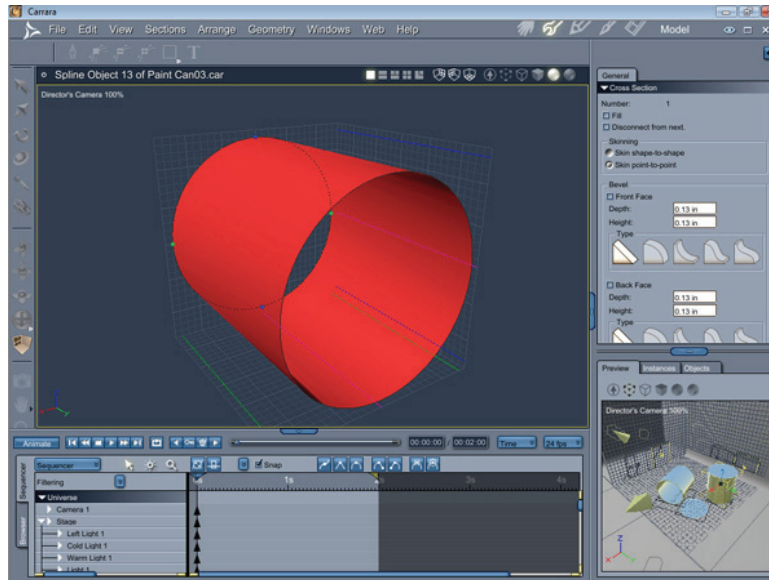
The **Model room** allows you to model and edit the objects in your scene using one of several types of modelers.

When you are using a modeler, you can view the entire scene in the **Scene Preview** window. This helps you keep the object in context. Refer to "The Interactive Renderer" to learn more about using the Scene Preview window.

Although the Modeling and Assemble rooms look similar, there are a few ways you can tell which room you're in:

- The title of the room displays in the upper right corner of the screen.

- The Room Switcher icon for the currently selected room is highlighted.

- In the Assemble room, all the objects in your scene are visible. In a modeler, only the object you're modeling is visible.

- Many of the tools in the Assemble room, such as the camera, lights, and environmental object tools, are not visible in the modeling windows.

To learn more about the modelers, refer to "Spline Modeling Overview," "Vertex Modeling Overview," and "Metaball Modeling." (Standard, Pro)

Storyboard Room

When you've added or created all the objects in your scene, you can use the easy-to-use features of the Sequencer tray to create animations.

If your goal is to create an animation, you can animate your scene using the **Storyboard room**. When using the Storyboard room, you see several views of your scene at once. You can view multiple frames of your scene and specify which frames should be displayed. For example, you can show just the key frames of a selected object.

You can create the animation at the same time you create the scene. You can make changes in your scene over time by creating and moving objects in the various frames, creating entire animations in a single window.

If you are creating an animation, you can also use the Sequencer to animate objects and apply modifiers or physical properties to objects that affect their behavior when animated.

If needed, you can switch to the Assemble room to get a larger view of your scene. Refer to "Animation Techniques" to learn more.
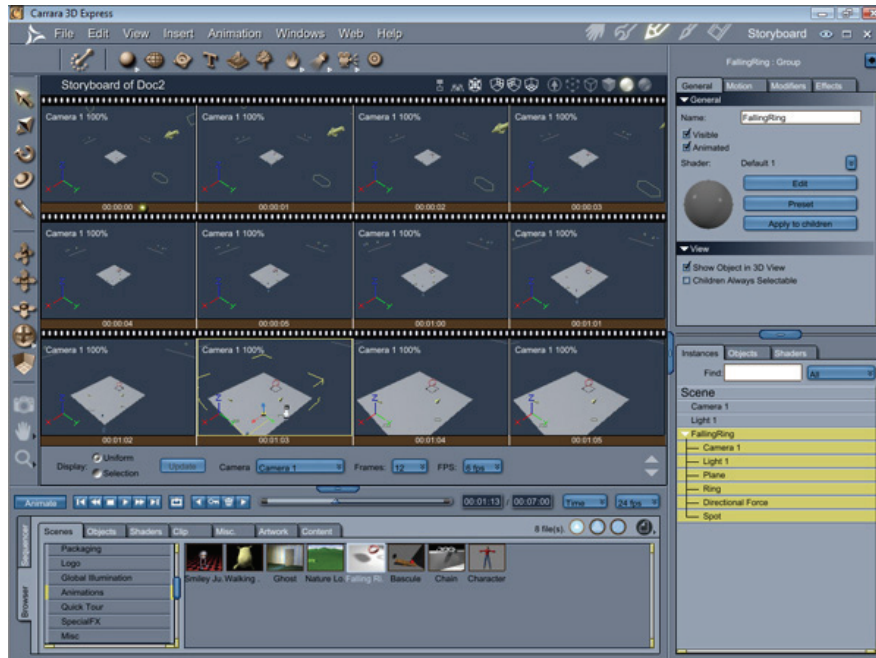
Texture Room

After you add content, you can create or modify shaders to enhance the realism. Carrara features state-of the-art shading technology, which lets you apply colors and surface textures to your objects. You can apply shaders that emulate wood grain or marble, or customize effects like bump, transparency, and reflection.

The **Texture room** is where you create, edit, and apply shaders to all or part of an object. Use shaders to create almost any kind of surface you can imagine. Shaders define aspects of the surface of an object, such as the color, texture, reflectiveness, and transparency. You can apply a Multi-Channel shader to an object or create a Layers List shader that combines several sub- shaders into one composite shader to create more complex effects. Each shader becomes a part of the scene and is referred to as a Master shader.

You can also see a preview of your scene while you are shading objects or creating a shader.

To learn more about shaders, refer to "What are Shaders?."
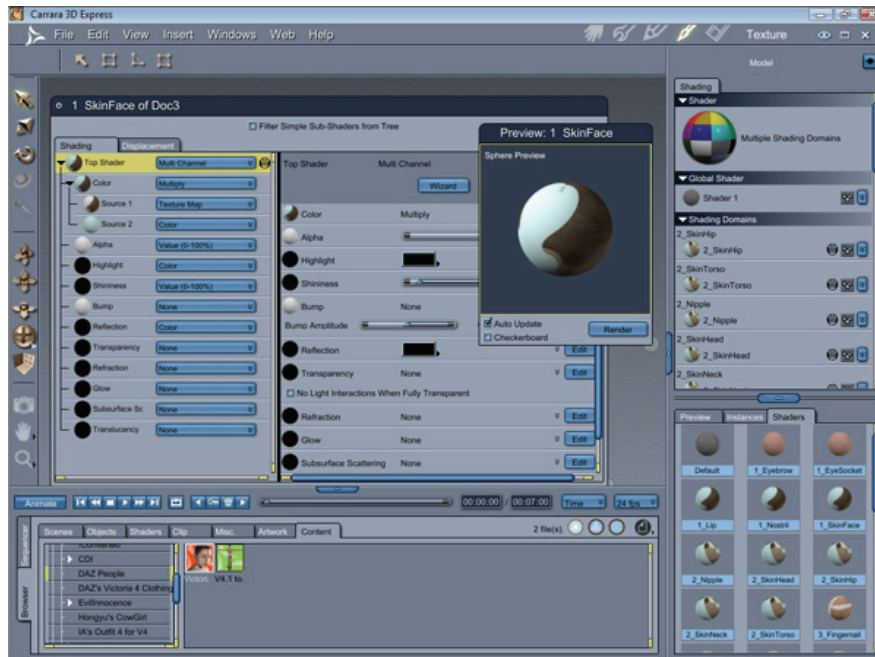
Render Room

The Render room provides three different renderers: Draft, Photorealistic, and Non-Photorealistic. It provides all the features necessary to create full-color 3D illustrations and animations. Carrara Standard and Pro also provide Web output using Viewpoint Experience technology

When your scene is completed, you can render it. Rendering is the process of capturing a two-dimensional image, such as a photograph, from your three-dimensional scene. You can also render an animation.

The rendered images from Carrara are compatible with popular Macintosh and Windows 2D graphics and page-layout programs, and can be saved for Web output in formats including JPEG, animated GIF, AVI, and QuickTime. You'll have no trouble compositing renderings with other images, or integrating text with your artwork.

The Render room is where you specify render settings to render an image or an animation.

After you have created primitives, added textures to your objects, and possibly animated your scene, you can render either a single image or a complete animation. Set up the Render options you want to use, then render the resulting image or animation.

You can also use the Render room to perform a batch render of a list of files. To learn more, refer to "Rendering from a Batch List."
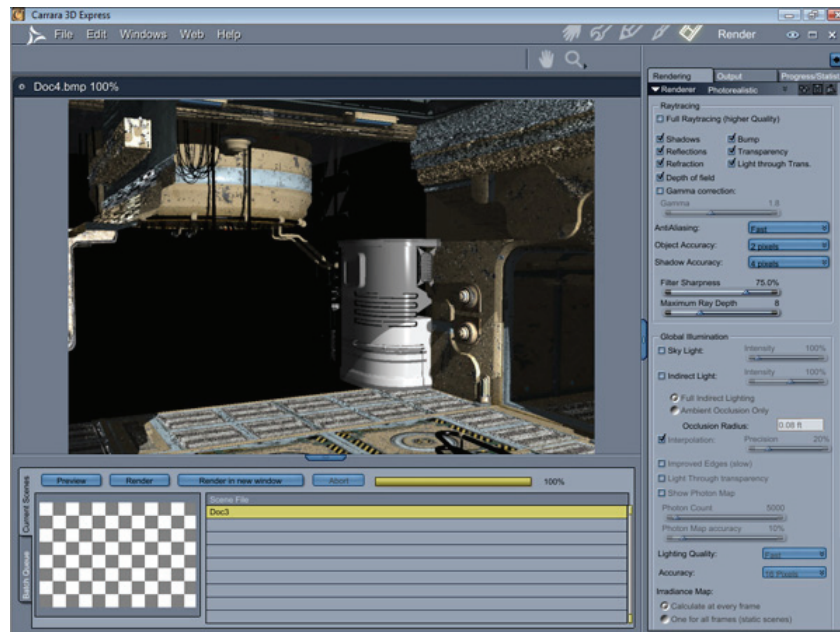
Carrara Interface Elements

When you create a new file or open an existing file, you first see the Assemble Room and its main interface elements. These are the primary work areas. Although they are described here, instructions for working in these elements appear in subsequent sections:
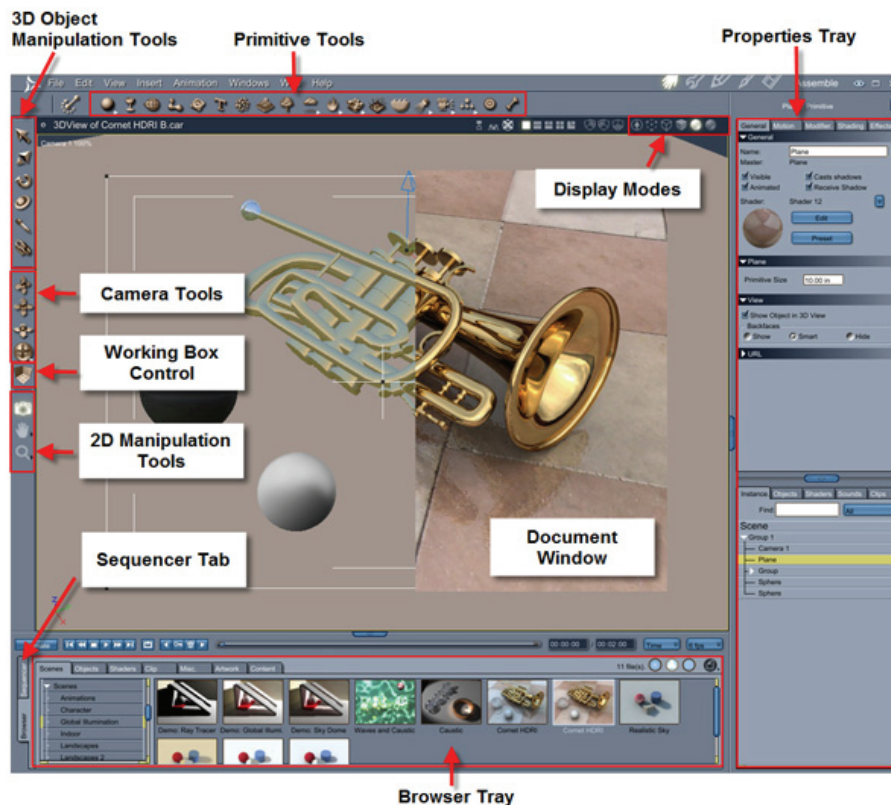
Carrara Menus

The commands you see in the menu bar vary, depending on the room and what version of Carrara that you are working in. Briefly, you'll see the following menus in Carrara:

- When you initially start Carrara, the Assemble Room includes commands in the **File**, **Edit**, **Windows**, **Web**, and **Help** menus. Most of these commands are also available in the Storyboard Room. Note that some commands are disabled until you have a scene open in the document window.

- When you create or open a scene, the Assemble and Storyboard rooms also display **View**, **Insert**, and **Animation** menu commands.

- When you enter the Model Room, there are additional menu commands that are particular to the modeler you are using. For example:

  - The Vertex modeler displays additional commands in the **View**, **Selection**, **Construct**, and **Model** menus. (Express, Standard, Pro)

  - The Spline modeler displays additional commands in the **View**, **Sections**, **Arrange**, and **Geometry** menus. (Standard, Pro)

  - The **Metaball** modeler displays additional commands in the **Geometry** menu. (Standard, Pro)

- The Texture and Render Rooms include additional commands in the **View** and **Insert** menus.
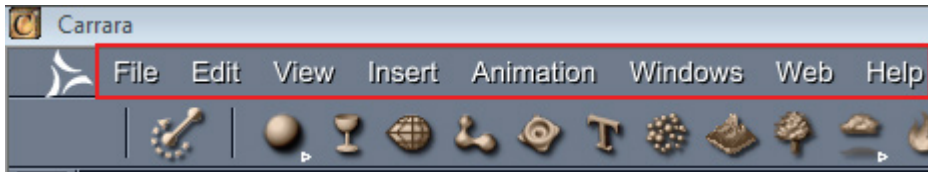
Carrara Tools

Carrara contains toolbars along the top and left sides of the document window by default. You can move these toolbars to any location you choose:

Common Tools: The tools that appear at the left side of the document window are common to the Assemble, Model, Storyboard, and Texture Rooms. These tools are described in "Working with the Common Tools."

Room Tools: Many of the menu commands have tools associated with them. They are located directly beneath the menu bar, and the tools available in the individual rooms vary depending on the room or modeler you are in. The tools are discussed where appropriate this manual, in conjunction with their associated menu commands.
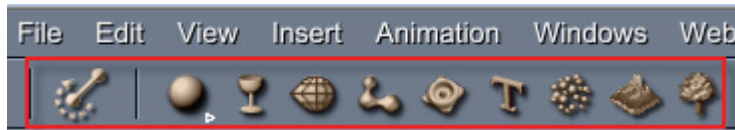
Document Window

The document window shows a view of the 3D workspace. The 3D workspace itself is also called the Universe.

In the Assemble room, the document window shows where objects, lights, and cameras are arranged to create a scene.
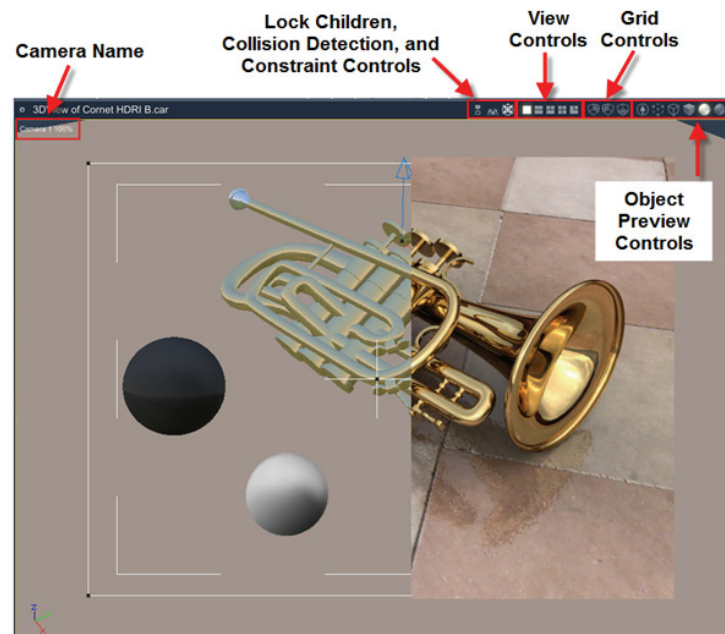
Browser Tray

The **Browser tray**, located at the bottom of the screen in the Assemble, Model, Storyboard, and Texture rooms, allows you to load and save content for use in your scenes. You'll find information about the Browser tray in "Using Content Files."
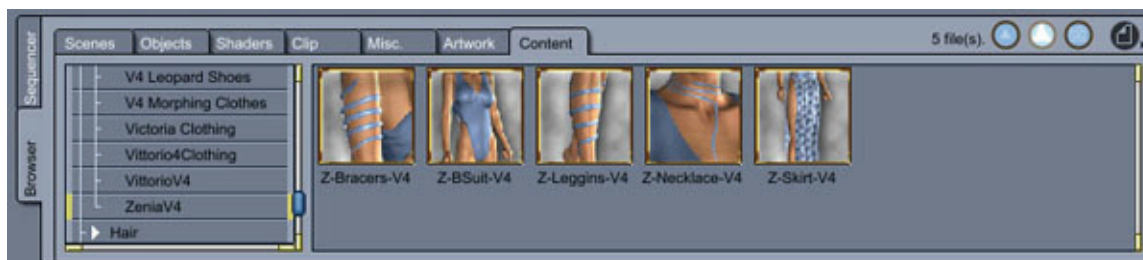
Properties Tray

The upper part of the **Properties tray** displays the properties of any element selected in the document window, Sequencer tray, or in the lower part of the Properties tray.

The lower part of the Properties tray contains several tabs that help you manage your scene.

For further information about the Properties tray, see "Using the Properties Tray."

Sequencer Tray

The Assemble, Storyboard, Model, and Texture rooms show a logical hierarchical representation of the scene, and also displays a timeline that aids in the creation of animation. All objects, cameras, shaders, and lights you bring into the universe are listed in the **Sequencer hierarchy**. You can expand or collapse the lists to view more or less detail.
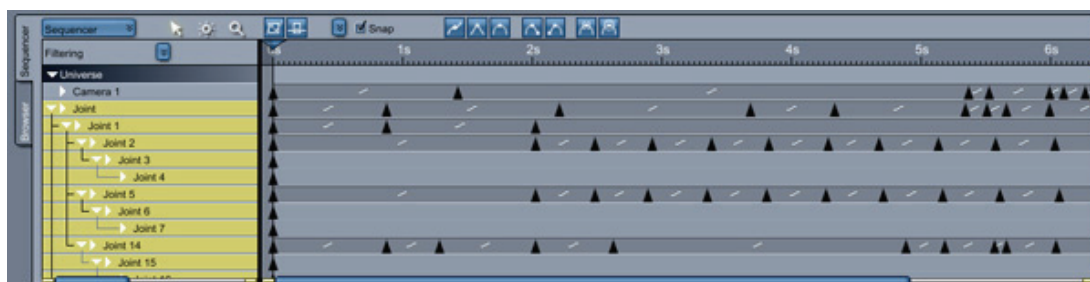
The Interactive Renderer

The **Scene Preview tab** (also known as the **Interactive Renderer**) is located in the lower part of the Properties tray. This renderer allows you to see a small view of the scene while you are in the modelers or the Texture room.
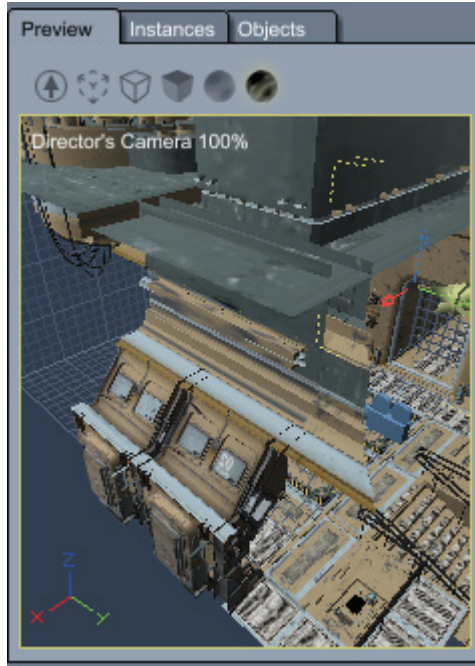
Displaying the Carrara Application Windows

Several commands in the Windows menu allow you to show and hide the various windows in Carrara:

- Choose **Windows > Maximized Window Mode** to maximize the display of the current scene in the available document window space. Uncheck this option if you want a resizeable window.
- Choose **Windows > Full Screen Mode** to display the Carrara application in full screen. Repeat the command to revert back to the previous state. The Toggle Full Screen Mode icon in the upper-right corner of the application window serves the same purpose.
- Choose **Windows > Hide Application** to minimize the application window. The Minimize icon in the upper-right corner of the application window serves the same purpose.

Setting Up Carrara

You can specify several settings that affect the way Carrara works. For example, you can set up application preferences, change the 3D preview settings, or modify your workspace. Carrara software's Preferences let you customize many of the application's default settings.

This section describes only the preferences that affect the entire application.

Setting Application Preferences

To set application launch preferences:

1. Choose **File > Preferences**.

2. Choose **Application** from the pop-up menu.
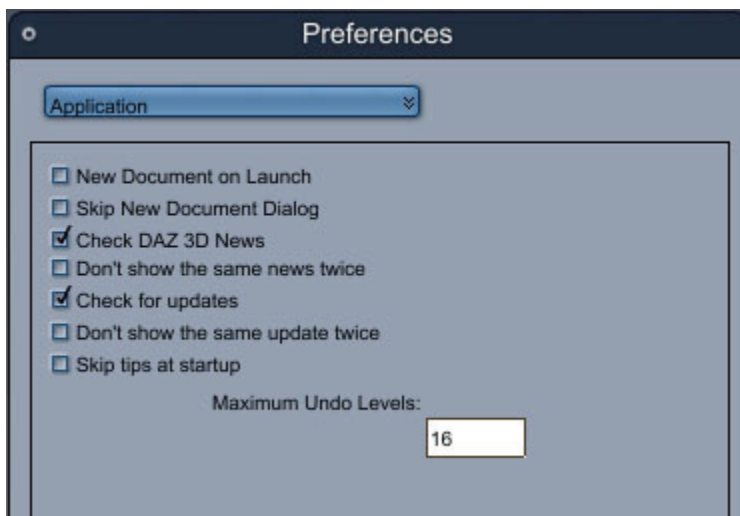
3. Check or uncheck the options that you want to apply when Carrara starts:

   o **New Document on Launch**: Check this option to automatically create a new document when you start Carrara.

   o **Skip New Document Dialog**: Check this option to prevent the New Document dialog from appearing when Carrara starts.

   o **Check DAZ 3D News**: When checked, displays news from DAZ 3D when you launch Carrara.

   o **Don't Show the Same News Twice**: When checked, does not display news that was previously displayed when launching Carrara.

   o **Check for Updates:** Check this option to search for Carrara updates and receive notification when updates are available.

   o **Don't Show the Same Update Twice:** Check this option to prevent repeated notification of the same update.

   o **Skip Tips at Startup:** Check this option to prevent the Tips window from opening when you start up Carrara.

   o **Maximum Undo Levels:** Enter the number of undo levels that you want to store.

Setting General Preferences

To set general application preferences:

1. Choose **File > Preferences**.

2. Choose **General** from the pop-up menu.
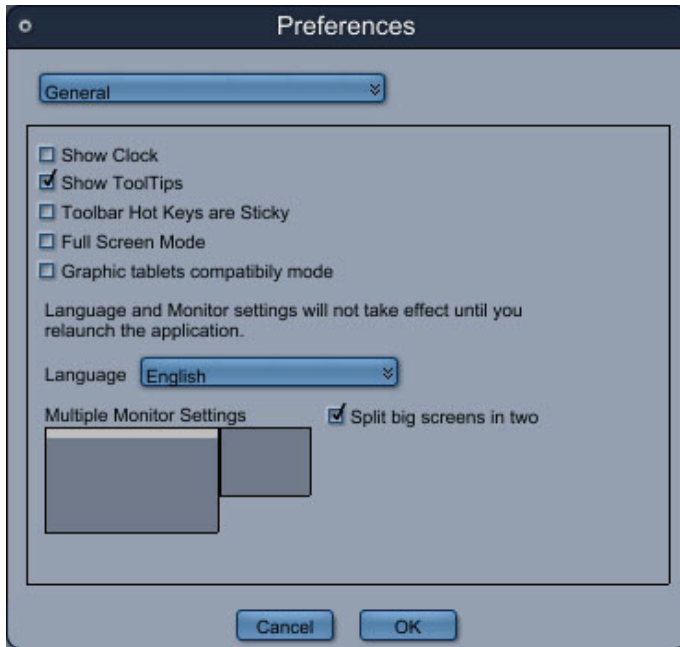
3. Check or uncheck the options that you want to apply:

- **Show Clock**: When checked, displays the current time in the main toolbar area.

- **Show ToolTips**: When checked, displays a brief description of the current tool when you hover the mouse over it.

- **Toolbar Hot Keys are Sticky**: When checked, a tool remains selected after you release the shortcut keys that activate the tool.

- **Full Screen Mode**: Displays Carrara in full screen mode.

- **Graphics Tablets Compatibility Mode**: Check this option if you are using Carrara with a graphics tablet.

- **Language**: Select the language (English or French) that will be used in the Carrara application.

- **Multiple Monitor Settings**: If you are using multiple monitors, click the monitor icons to select or deselect the monitors that you want to use with Carrara. Monitors that are used are colored gray.

- **Split Big Screens in Two**: Some combinations of Windows and video cards do not allow you to use dual monitor setups that behave like a single big monitor. If you check this option, you can still use Carrara in dual monitor mode.

Setting UI Effects Preferences

To set user interface effects:

1. Choose **File > Preferences**.

2. Choose **UI Effects** from the pop-up menu.
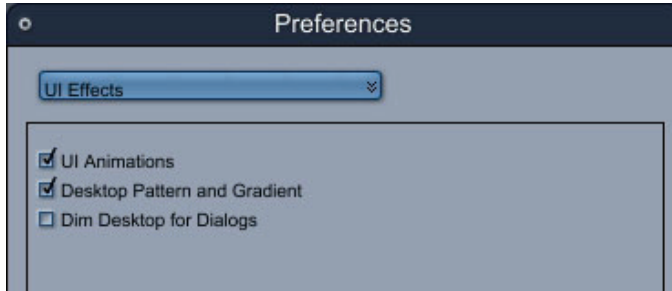
3. Check or uncheck the options that you want to apply:

   - **UI Animations**: When checked, displays animated UI windows where applicable.

   - **Desktop Pattern and Gradient**: Toggles the display of patterns or gradients that are applied to the desktop.

   - **Dim Desktop for Dialogs**: Check this option to dim the user interface in the background while dialogs are opened.

Changing Keyboard Shortcuts

In Carrara, you can change the keyboard shortcuts of most commands and tools.

To edit a shortcut:

1. Choose **File > Preferences**.

2. Choose **Keyboard Shortcuts** from the pop-up menu.
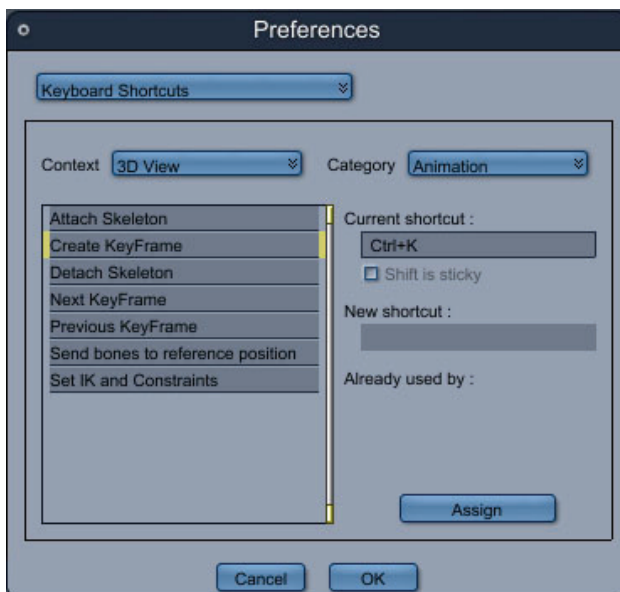
3. From the Context pop-up menu select the context of the command. Choices are **3D View**, **Application**, **Shading Room**, **Spline modeler**, and **Vertex modeler**.

4. From the Category pop-up menu select the menu or tool bar where the command appears.

5. In the shortcut list, click on the name of the command or tool that you want to modify. The current shortcut, if any, appears in the Current Shortcut field.

6. To add or change a shortcut, click in the "New Shortcut" input box to activate it.

7. Press the combination of keys you'd like to use. For example press Command/Control and the S' key. If the shortcut is already used, the application tells you by which command. Generally, tools use a simple letter, while menu commands use a letter with Command/Control.

8. Click on the **Assign** button and then on the **OK** button to record the shortcut.

Setting Imaging and Scratch Disk Preferences

Carrara uses free space on your hard drive to store work in progress or unsaved renderings of animations. This data is stored in the DAZ 3D\DazTmp folder. The partition containing the DAZ 3DTmp folder is known as the "scratch disk."
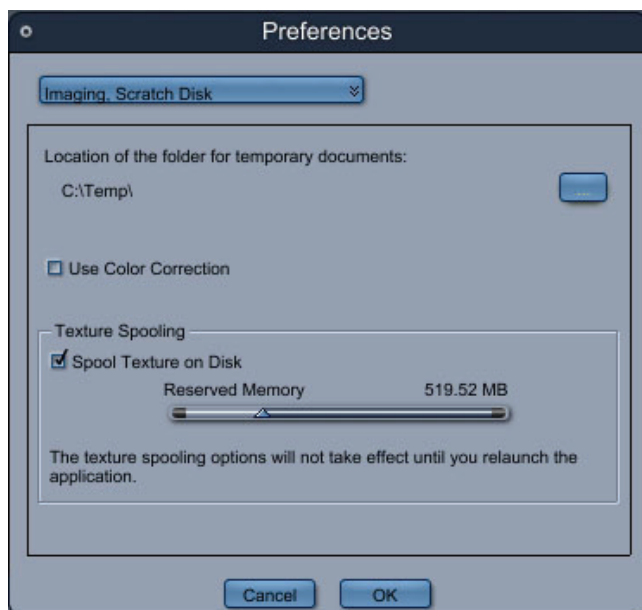
To select the scratch disk:

1. Choose **File > Preferences**.

2. Choose **Imaging, Scratch Disk** from the pop-up menu.

3. Click on the button to select the location of the DAZ 3D\DazTmp folder.

4. Click **OK**.

To set Imaging Preferences:

1. Choose **File > Preferences**.

2. Choose **Imaging, Scratch Disk** from the pop-up menu.

3. Check or uncheck the following options:

    o **Texture Spooling**: Check **Spool Texture on Disk** if you want to use your hard drive to store texture information when the amount of memory specified by the **Reserved Memory** setting is exceeded.

General Scene Settings Preferences

To set General Scene Settings:

1. Choose **File > Preferences**.

2. Choose **General Scene Settings** from the pop-up menu.
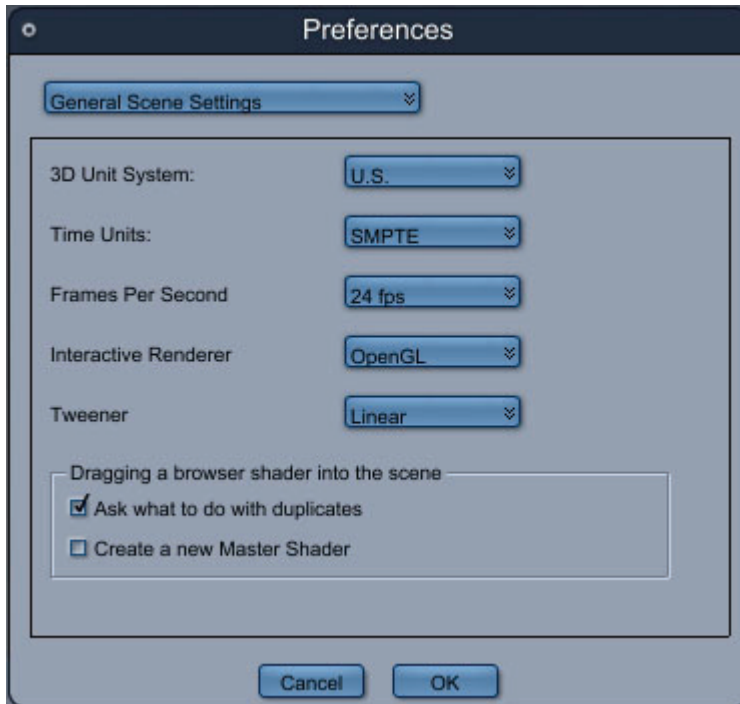
3. Choose the following options:

    - Choose a measurement system (U.S. or Metric) from the **3D Unit System** list. Your selection becomes the default unit of measure for all 3D objects.

    - Choose a measure system for time in the **Time Units** list. Choices are SMPTE or frames.

    - Choose the default number of **Frames Per Second** that you want to use in animation. The default setting is 24 FPS, but available options range from 1 to 60 FPS.

    - In the **Interactive Render** list, select the type of display you want to use for the Interactive Renderer. Choices are **OpenGL** or **Software**.

    - In the Tweener list, check the default option for calculating movement that is in between keyframes. Options are **Bézier**, **Discrete**, **Formula**, **Linear**, **Noise Tweener**, **Oscillate**, **Spline**, and **Velocity**. These options are discussed in more detail in "Tweeners."

    - In the **Dragging a Browser Shader into the Scene** section, check or uncheck the options to ask what to do with duplicates (shaders that are already used in your scene), or to create a new master shader automatically when you add a browser shader into the scene.

Setting Units of Measurement in the Properties Tray

When you configure General Scene Settings preferences, Carrara allows you to choose between U.S. and Metric measurement systems. The Interface tab in the Properties tray allows you to define the displayed units that are used in your scene.

To configure the displayed units for your scene:

1. Open the Scene tabs in the Properties tray using any of the following methods:

    o   In the document window, click the Working Box, or click outside any objects in your scene.

    o   From the Sequencer tray, choose **Scene Effects**.

    o   From the Instances tab, click **Scene**.

2. Open the **Properties tray: Interface tab: Displayed Units panel**.
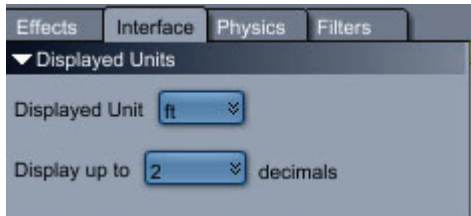
3. Use the **Displayed Unit** menu to select the units of measurement. Choices are **pixel**, **pt**, **pica**, **in**, **ft**, **mi**, **mm**, **cm**, **m**, or **km**.

4. Choose the desired number of decimal points (0 through 5) from the **Display up to** menu.

OpenGL Preferences

Two options appear in the General Scene Settings preferences. These settings allow you to choose between OpenGL or Software rendering. The difference between these options is as follows:

- **Software**: This is the default renderer. It does not take advantage of hardware acceleration. However it is often the fastest and most reliable renderer. We recommend that you stick with it unless you get much better performance in OpenGL.

- **OpenGL**: Carrara offers an improved OpenGL engine that improves support for texture maps with alpha channels, as well as an improved textured mode. These improvements also enhance performance in the Vertex modeler, allowing for editing of larger meshes. This renderer uses the OpenGL API to take advantage of your video card to display 3D objects. If your graphic card fully supports OpenGL, you should get better performance. However, it should be noted that many cards do not fully support OpenGL (especially older cards or the cheaper ones), in that case you will get a better redraw speed when using the Software renderer.

The OpenGL renderer uses your graphic card to accelerate the display of 3D objects. Although it should run on most system configurations, there are some specific requirements for OpenGL.

The OpenGL renderer requires a recent version of the graphic driver for your video card. (The graphic driver is the interface between your operating system and the graphic hardware).

To update your graphic driver, please contact the manufacturer of your graphic card or computer. It is usually possible to download the latest version of the driver you need on their web site. In addition to choosing between OpenGL and software rendering, you can find the following option in the Preferences dialog:
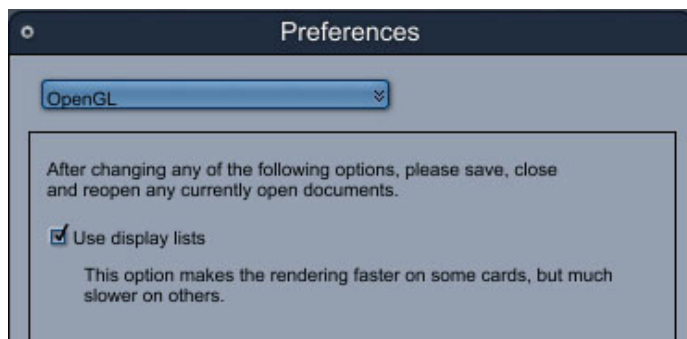
To use display lists:

1. Choose **File > Preferences**.

2. Choose **OpenGL** from the pop-up menu.

3. Check **Use Display Lists** to cache the geometry of existing objects on the graphic card. This speeds up the display of your objects but may cause problems with some video drivers. When you change this setting, you will need to save, close, and reopen any documents that were open at the time you changed the setting.

File Format Preferences

To choose default file format preferences:

1. Choose **File > Preferences**.

2. Choose **File Format** from the pop-up menu.
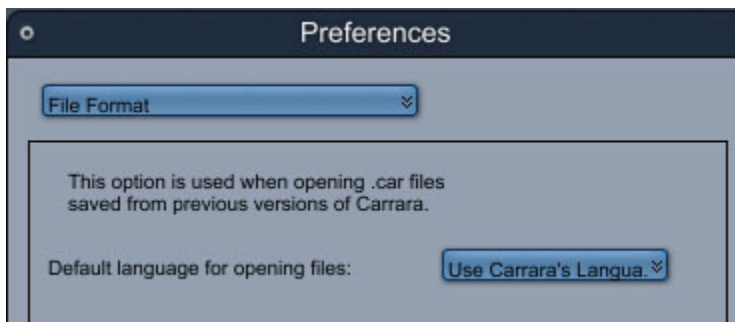
3. Select the default language to use when opening files. Choices are **Use Carrara's Language**, **Western**, or **Japanese**.

Interface Settings Preferences

To choose the font used in the interface:

1. Choose **File > Preferences**.

2. Choose **Interface Settings** from the pop-up menu.
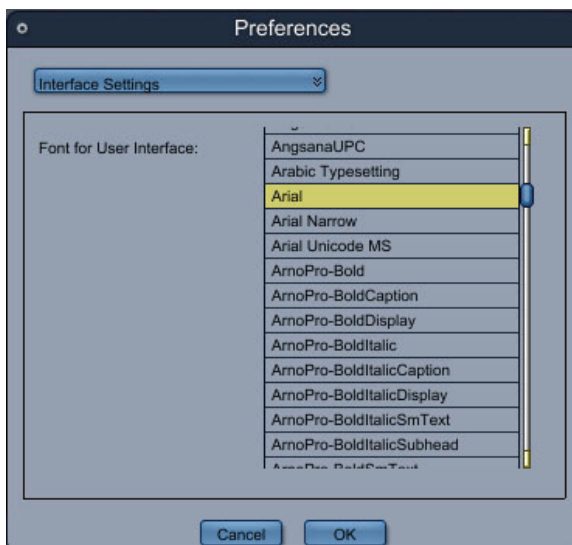
3. A list of all fonts in your system appears. Select the font that you want to apply to your interface settings.

User Interface Color Preferences

To set interface element colors:

1. Choose **File > Preferences**.

2. Choose **UI Colors** from the pop-up menu.

3. Click the color chip near the element whose color you want to change, or select the triangle to quickly select a color. The color chips control the Background, Major Accent, Control, Highlight, Alternate Highlight, and Drag and Drop colors.

4. Select a color from the color picker that appears.

5. Click **OK**.

3D View Preferences

To set 3D View preferences:

1. Choose **File > Preferences**.

2. Choose **3D** View from the pop-up menu. The 3DView Preferences dialog appears.
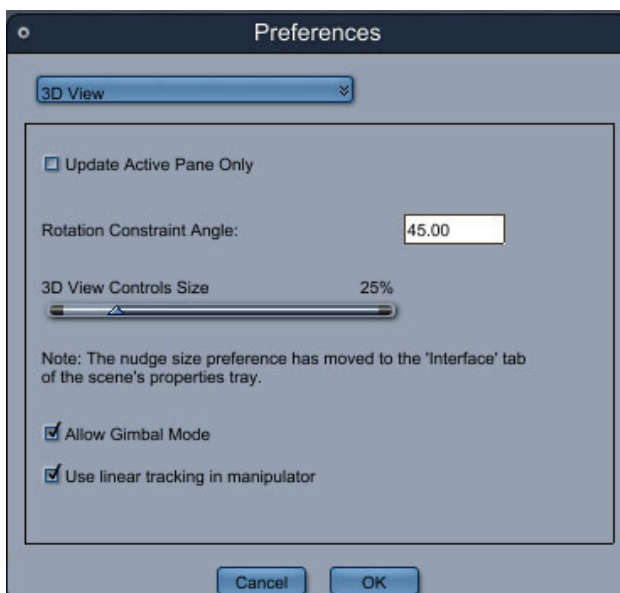
3. Set options as desired:

- **Update Active Pane Only**: Updates the active pain automatically when you move an object.

- **Rotation Constraint Angle**: Snaps the rotation to the specified angle when the rotation constraint is activated (using Shift).

- **3D View Controls Size**: Specifies the size of the manipulators in the 3D view.

- **Allow Gimbal Mode**: The Gimbal mode is a special mode where each axis of rotation is displayed so that it corresponds to the actual effect of modifying a particular angle. When this option is off, the circles that appear in relation to axes are perpendicular to the axis to which they correspond.

- **Use Linear Tracking in Manipulator**: When using the rotation manipulator with this option on, linear tracking looks at the motion of the mouse in tangent with the circle of rotation. When off, tracking tries to keep the position of the mouse cursor on the sphere of rotation.

3D View Color Preferences

To set 3D View colors:

1. Choose **File > Preferences**.

2. Choose **3D View Colors** from the pop-up menu.

3. Click the color chip near the element whose color you want to change or select the triangle to quickly select a color. The color chips control the colors of the Working Box, Wireframes, Ground Plane, Lights, Cameras, Background, Active Pane Border, Camera Names, Selected Objects, and Silhouette on Walls.

4. Select a color from the color picker that appears.

5. Click **OK**.

Modeling Room Preferences

To set Modeling Room preferences:

1. Choose **File > Preferences**.

2. Choose **Modeling Room** from the pop-up menu.
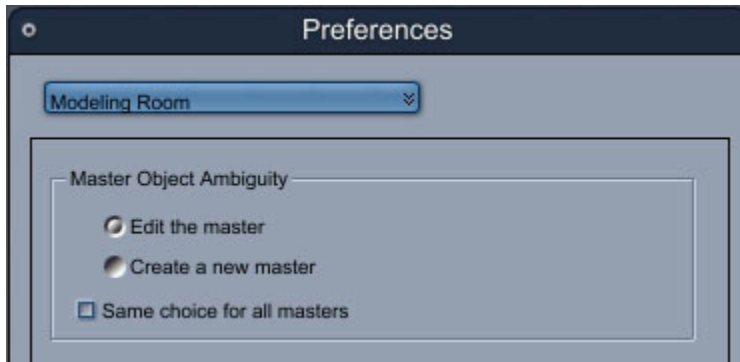
3. Set options as desired:

- **Master Object Ambiguity**:

    - **Edit the Master**: Edits all instances of a particular master object.

    - **Create a New Master**: Creates a new master object for the current instance, and leaves other instances unchanged.

- **Same Choice for All Masters**: This option does not display the dialog when you edit an instanced object, and uses previous settings.

Shader Editor Preferences

To set Shader Editor preferences:

1. Choose **File > Preferences**.

2. Choose **Shader Editor** from the pop-up menu.
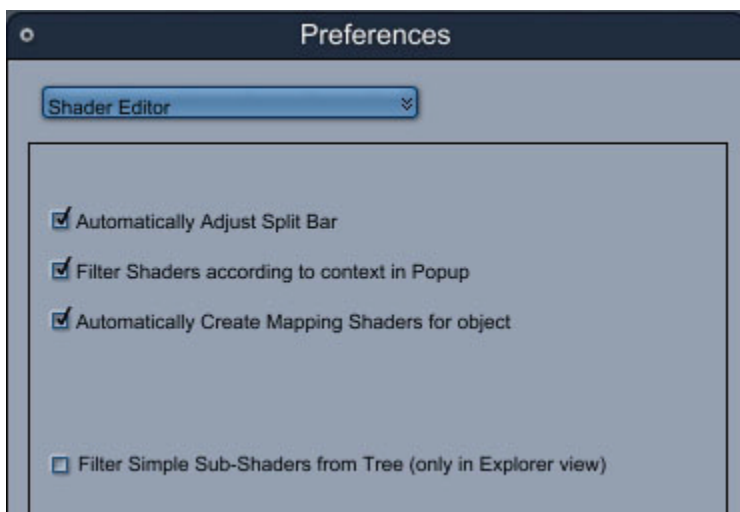
3. The options that you can set for the Shader Editor preferences are as follows:

- **Automatically Adjust Split Bar**: Automatically adjusts the position of the split bar in the shader editor window.

- **Filter Shaders According to Context in Popup**:

- **Automatically Create Mapping Shaders for Object**:

- **Filter Simple Sub-Shaders from Tree (only in Explorer View)**: When displaying complex shaders, filters out the display of sub-shaders to simplify the shader tree.

Shading Room Preferences

To set Shading Room preferences:

1. Choose **File > Preferences**.

2. Choose **Shading Room** from the pop-up menu.
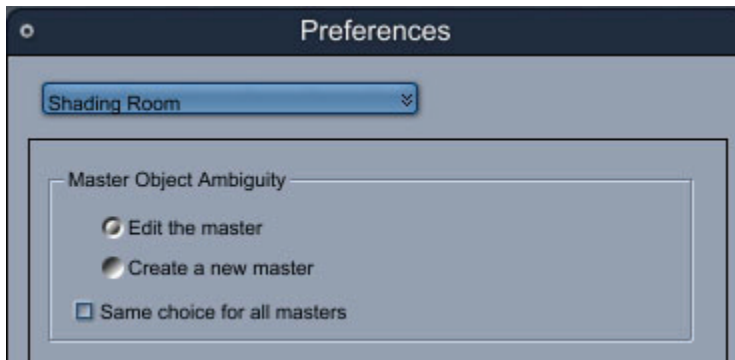
3. The options that you can set for the Shading Room preferences are as follows:

- **Master Object Ambiguity**:

  - **Edit the Master**: Edits all instances of a particular master shader.

  - **Create a New Master**: Creates a new shader for the current instance, and leaves other instances unchanged.

- **Same Choice for All Masters**: This option does not display the dialog when you edit an instanced shader, and uses previous settings.

Resetting Application Preferences

You can reset the preferences of the application to their default values by using the **File > Reset Preferences** command. This will exit the application and the default preferences will be used the next time it is launched.

Adding Objects

Assemble Room Overview

The Assemble room provides a large document window for working on your scene. When you create a scene, you assemble the objects, place lighting and cameras, and specify effects used in that scene. You can even apply pre-defined textures to objects in your scene while in the Assemble room.
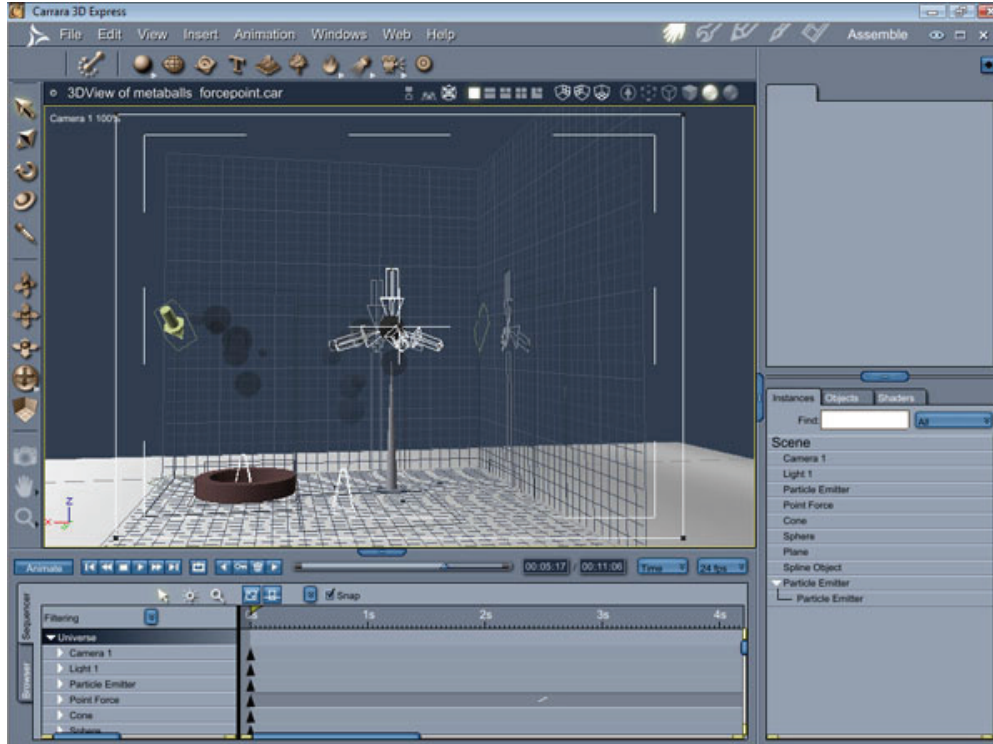
The Assemble room is where you arrange objects, cameras, and lights to form a scene. Like the Modeling windows, the Assemble room provides up to four separate panes, each containing a different view of your scene. You can change views using the view layout controls.

The 3D workspace of the Assemble room is called the **universe**. The universe is where all objects are displayed, assembled, and manipulated. Carrara uses a coordinate system called the Cartesian coordinate system to reference positions in the universe. A trio of unique coordinates (X, Y, and Z) is associated with each position in the universe.

The tools available in this room let you place and manipulate objects and position cameras so you can view, and later render, your scene from different viewpoints. By default, Carrara opens new scenes with one distant light and one conical camera.

You can also begin the modeling process in the Assemble room by dropping some object primitives into a scene. After you add shaders in the Texture room, switch back to the Assemble room to add modifiers and lights as needed. You can position your cameras, adding additional cameras as needed, or you can view the results from the Director's camera.

You can change the preview options for viewing objects as you assemble your scene, and you can do a quick test render of a small area of the scene to check your work. Refer to "Choosing a Preview Mode" to learn more.

Assemble Room Menu Commands and Tools

The Assemble room menu commands are arranged in eight categories.

- **File menu commands** allow you to open, close, import, and export various types of content. Many of these commands are described in "Import/Export." You'll also find information on the Preferences commands in "Setting Up Carrara."

- **Edit menu commands** are primarily used when making changes to various types of objects in your scene. Commands in this menu are described where appropriate throughout the manual.

- **View menu commands** primarily deal with zooming and panning in the working box, and showing or hiding production frames, grids, and wireframes.

- **Insert menu commands** are associated with adding and creating objects. The majority of these commands also have tools associated with them. You'll find instructions for using these commands and tools in the following chapters:

  - See "Using Primitive Tools" for information on the Sphere, Cube, Cone, Cylinder, Icosahedron, Plane, Infinite Plane, and Splat tools. You'll also find information about the Text and Particle Emitter tools in this chapter.

  - See "Spline Modeling Tools and Techniques" for information on working with Spline Objects. (Standard, Pro)

  - See "Vertex Modeling Basics" for information about the Vertex Object tools. (3D Express)

  - See "Vertex Modeling Basics" and "Vertex Modeling Techniques" for information about the Vertex Object tools. (Standard, Pro)

  - See "Metaball Modeling" for information on how to work with Metaball Objects. (Standard, Pro)

  - See "Using Formulas in Carrara" for further information about Formulas.

  - See "Using Environmental Primitives" for instructions on Terrains, Plants, Clouds (Volumetric, Old Volumetric, and Clouds tools), Fire, Fountains, Fog, and Oceans.

  - See "Setting Lights" for instructions on the Spot, Bulb, Distant, Moon Light, Sun Light, Tube Light, Shape Light, and Anything Glows tools.

  - See "Setting Cameras" for information on creating and using cameras.

  - See "Physics" for further information about the force tools (Directional, Point, Damping, Torque, and Flow Force tools). (Standard, Pro)

  - Refer to "Character Animation" for further information about Target Helpers and Bones. Standard, Pro)

  - Refer to "Adding Sound to Your Scene" for information about sound.

  - **Animation menu commands** are covered in "Animation Techniques." (3D Express)

  - **Animation menu commands** are covered in "Animation Techniques" and "Character Animation." (Standard, Pro)

- **Windows menu commands** open and close the various windows that are available in Carrara.

- **Web menu commands** open your web browser and display several different support pages that are located on the DAZ 3D web site.

- **Help menu commands** open the reference manual in PDF format and display tool tips when you hover over the Carrara tools.

Working with the Common Tools

You'll use many of the same tools while working in the different rooms and modelers. This section introduces the common tool sets you'll be using.

Carrara provides tool tips to help you identify what each tool is. Simply place the cursor over the tool to display the name of that tool.

Customizing Tool Locations

You can customize the location of Carrara's tool sets.

To move tool sets:

1. Click the tool set that you want to move.

2. Drag the tool set to a new location on the screen. The tool sets' dock areas on the left of the screen and under the menu bar can be used to automatically align the tool sets.

When a tool set is floating outside a tool dock, press Control, then drag the tool set to rotate it.

3D Object Manipulation Tools

The 3D Object Manipulation Tools appear in the left portion of the interface window. You can position these tools in the left or top sections of the user interface, or float them.

Unlike the 2D view tools, these tools manipulate the objects in your scene. You can do all these manipulations numerically or with the mouse.

From top to bottom, the tools are:

*   **Move tool**: Repositions the objects in your scene. Refer to "Arranging Objects" to learn the different ways to move objects.

*   **Scale tool**: Increases or decreases the size of the object. Refer to "Scaling Objects" to learn how to scale objects.

*   **Rotate tool**: Shifts an object around its hot point. Refer to "Rotating Objects" to learn how to rotate objects.

*   **Universal Manipulator tool**: Moves, scales, and/or rotates the object depending on how you use the tool.

*   **Eye Dropper tool**: Selects an object's shader so that you can apply it to another object. To learn more, refer to "Shading an Entire Object."

Camera Tools

Use the Camera Rotation tool and the different Track tools to position the camera of the current view in the Assemble, Storyboard, or Model rooms. When you use the camera tools, you are moving the camera in your scene.

To learn more about working with the cameras, refer to "Setting Cameras."

To position a camera, do one of the following:

- Click one of the Camera tools, then click and drag in the document window to directly manipulate the camera.

- Click a tool and drag in a direction to indirectly manipulate the camera.
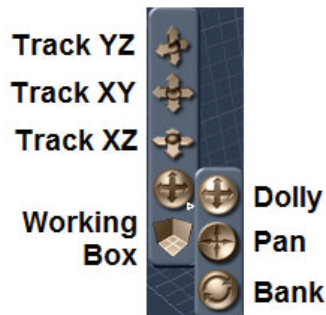
The **Track tools** affect the position of the camera without affecting its orientation. The objects in your field of view will appear to change position in relation to each other as you view your scene from many different angles.

- **Track YZ**: Drag the mouse up and down to track the camera up and down. Drag the mouse left and right to track the camera toward or away from you.

- **Track XY**: Drag the mouse up and down to track the camera up and down. Drag the mouse left and right to track the camera left and right.

- **Track XZ**: Drag the mouse up and down to track the camera toward or away from you. Drag the mouse left and right to track the camera left or right.

Using Shortcut Keys:

- Use the shortcut **Option/Alt + Left Mouse Button** to rotate the camera (Dolly). Refer to "Dollying".

- Use the shortcut **Option/Alt + Right Mouse Button** to track the camera horizontally and vertically.

- Use the shortcut **Option/Alt + Middle Mouse Button** to move the camera back and forth.

- If your mouse is equipped with a Mouse Wheel you can use it to move your camera back and forth

The **Camera Rotation tool** has three options, which you access by clicking the arrow that appears at the bottom-right corner of the tool. The popup menu displays the additional options:. Select the one you want to use:

- **Dolly** changes the camera's position and orientation while keeping it pointed at the same spot in the scene. .

- **Pan** works like turning your head: the camera's position in 3D space does not change when you orient it in any direction. Refer to "Panning."

- **Bank** allows you to roll the orientation of the camera without changing its location. Imagine tipping your head to the side. Refer to "Banking."

NOTE! In isometric views it is not the camera that is moved when using those shortcuts but the zoom and pan that are modified.

2D View Tools

Carrara also provides tools that allow you to pan and zoom the scene in the document window, and to perform test renders from within the document window. These tools appear along the left side of the document window by default. They are the "Test Render Tool," the "2D Pan Tool," and the "2D Zoom Tool."

Test Render Tool

Use the Test Render tool to render an area of your scene in the Assemble or Storyboard rooms, or in the Scene Preview window. You can easily check out an effect without rendering the entire scene.

To use the Test Render tool:

1. Choose the **Test Render** tool.

2. In the document window, drag a marquee across the area to be previewed.

3. Carrara renders the area you have selected.
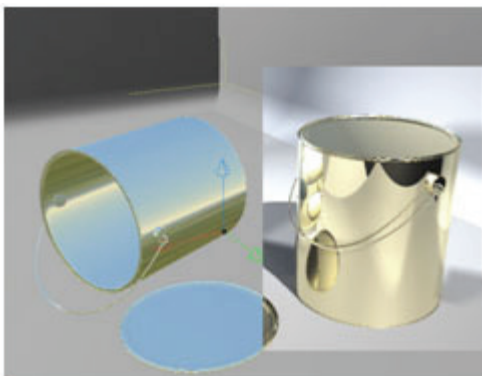
To work efficiently, keep the size of your render previews small. Larger areas take longer to ray trace and require more memory. If you want to ray trace more than a small portion of your scene, you might as well do a low resolution rendering. Refer to "Rendering."

2D Pan Tool

If your scene is too large to fit within the document window, you can use the 2D Pan tool to reposition the view of the camera like a flat image. Drag the tool up, down, right, or left to reposition the scene in the document window.

To move an area of the scene in the document window:

1. Click the 2D Pan tool.

2. Drag the cursor in the direction you want to move the view of the scene.

NOTE! To reset the 2D pan, use the **View >Reset 2D Pan** command, or click the arrow at the right of the 2D Pan tool and choose **Reset 2D Pan**..

2D Zoom Tool

As you create your 3D artwork, your scene can quickly become too large to see the entire scene in the Assemble room. The 2D Zoom tool allows you to zoom out to view your entire scene, if necessary. You can also use it to enlarge areas to precisely edit objects.

To zoom in to an area of your scene:

1. Click the 2D Zoom tool.

2. Click on a point in your scene to enlarge. The scene is magnified. You can also drag a marquee around an area to magnify it.

To zoom out of an area:

1. Click the Zoom tool.

2. Hold down **Option/Alt** and click an area.

You can also zoom in or out of your scene to a selection, a specified percentage, or preset percentage

To zoom in or out to a selection, object, or fixed percentage, use one of the following methods:

- Choose **View > Zoom**, and choose a zoom option from the popup menu.

- Click the arrow at the right of the Zoom tool and choose a zoom option from the popup menu.

  - **Zoom to Selection**: Choose this option to zoom into the objects that you have selected.

  - **Zoom to**: Choose this option to enter a new zoom percentage of your choosing.

  - **Fixed percentages**: Allows you to zoom in to 25%, 50%, 75%, 100%, 200%, 300%, 400%, 500%, or 600%.

Using the Document Window

The document window shows a view of the 3D workspace. The 3D workspace itself is also called the universe. A single document window is associated with a scene in the Assemble and Storyboard rooms.

In the Assemble room, the document window shows where objects, lights, and cameras are arranged to create a scene.

You can easily display one or up to four views in the document window. As you move an object in one view, it changes location in the other views as well.
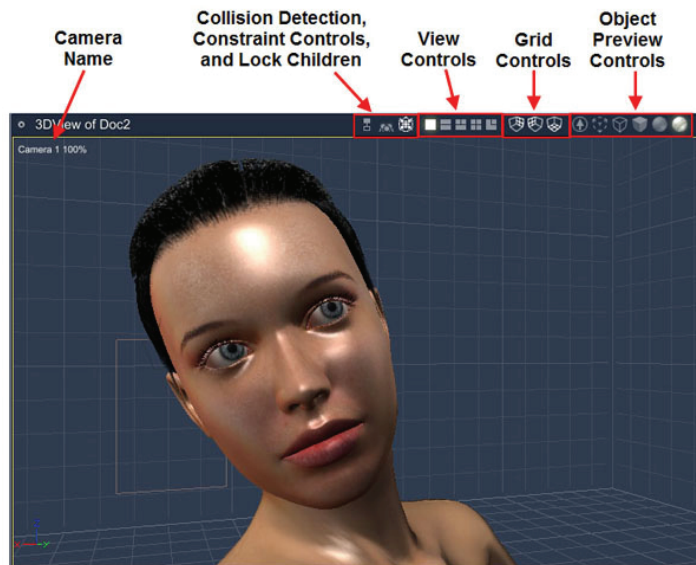
Displaying Multiple Camera Views

The top bar of the document window contains several controls that allow you to view the items in your scene in many ways. They are broken down into four types of controls:

The **View controls** allow you to display up to four different camera views in your document window. Options from left to right are:
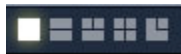
View controls.

- **One View**: Displays one camera view in the document window. By default, the Camera 1 view is used.
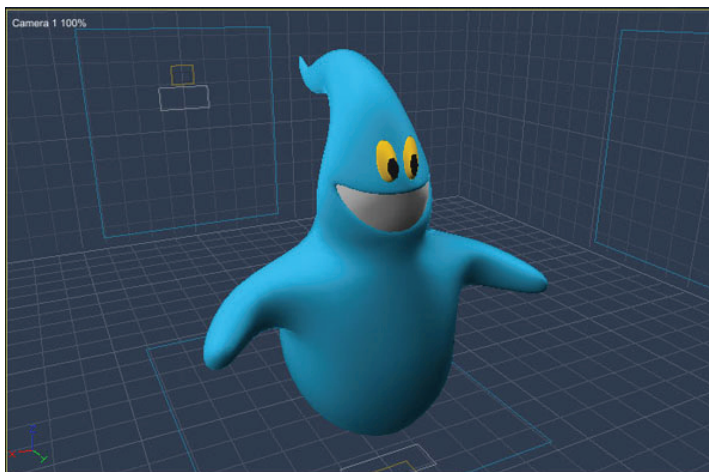
- **Two Views Horizontal**: Displays two different views stacked one on top of the other. By default, the views are Camera 1 and Top.
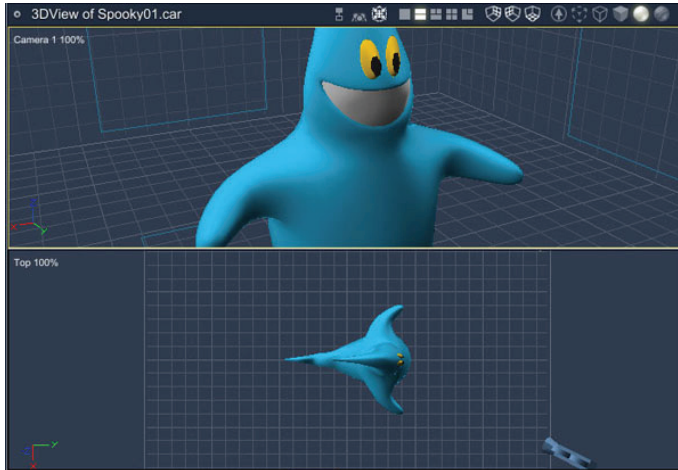
- **Three Views Top**: Displays two camera views on top and one view on the bottom. By default, the views are Top, Left, and Camera 1.
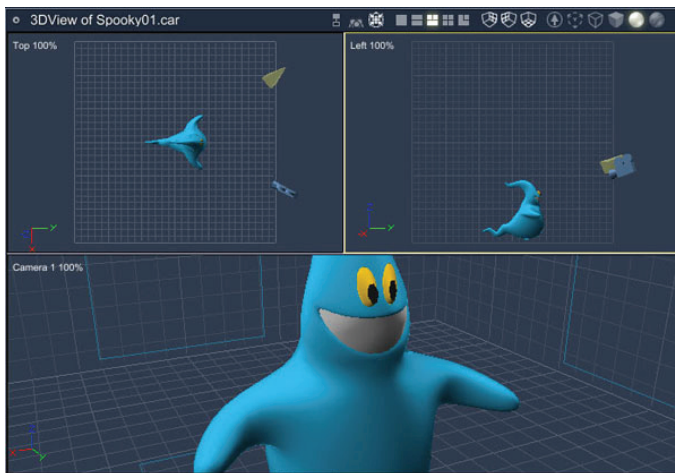
- **Four Views**: Displays four camera views. By default, Top and Camera 1 are on the top, and Left and Front are on the bottom.
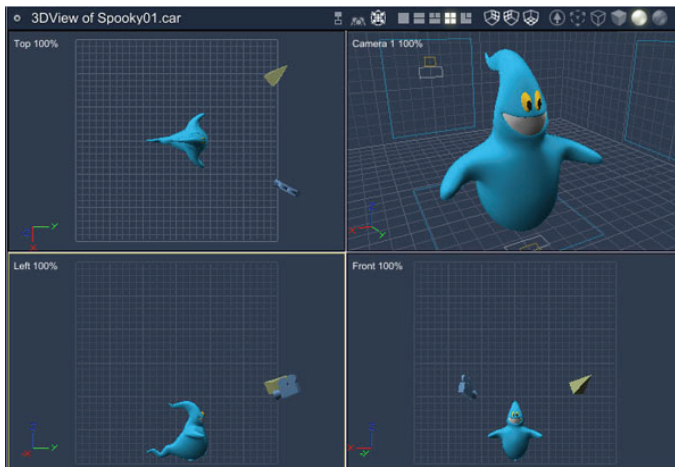
- **Inset View**: Displays two camera views. By default, Camera 1 appears in the main document window, and the Top camera appears as in inset.
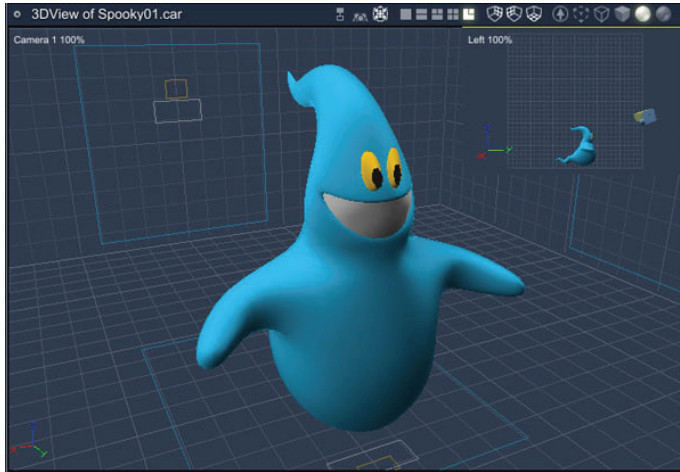
Using the Working Box

The **working box** is a representation of a coordinate system and is visually presented by three intersecting planes. It provides a framework that helps you work in a three-dimensional universe with two-dimensional devices: the mouse and monitor.

The working box is a three-dimensional reference for positioning and arranging objects. Each of the views shows an aspect of the working box:

- The Director's Camera and any cameras you create show a three-dimensional view of the working box.

- The other views show a two-dimensional view of part of the three-dimensional space.
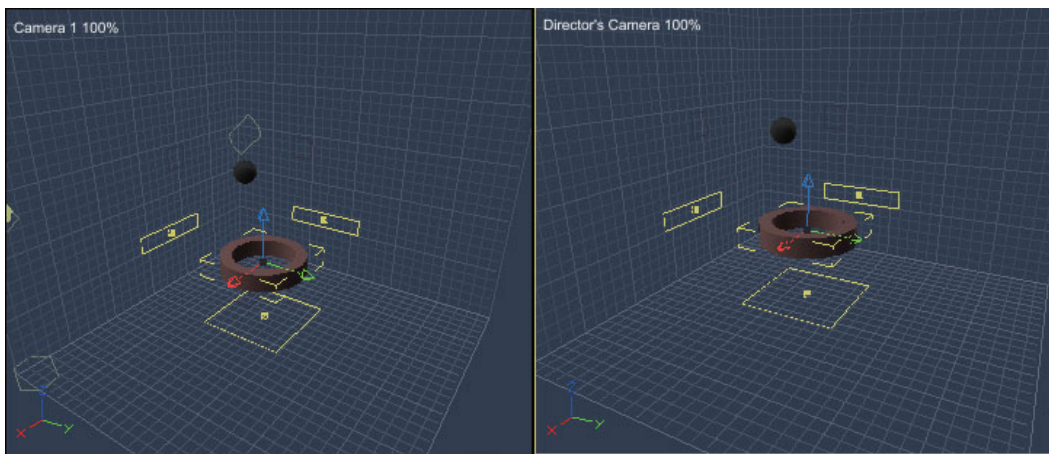
You can think of the working box as a 3D ruler that helps you bring objects together more easily and accurately. The grids of the working box are a visual reference for the dimensions and orientation of the universe. By default, the working box floor is centered on the origin of the universe (0, 0, 0) and aligned with the axes of the universe. The ground plane, lying on the XY plane, is positioned at Z = 0.

Object Bounding Boxes and Projections

The bounding boxes of all objects, including cameras and lights, cast 2D profiles called **projections** on the planes of the working box. These projections show the object's position and orientation in relation to each of the three planes. They help you see which objects are "farther away," "higher," or "tilted," and help you move the objects into the positions and orientation you want. You can drag the projection to scale, move, or rotate the object in the plane of the projection.
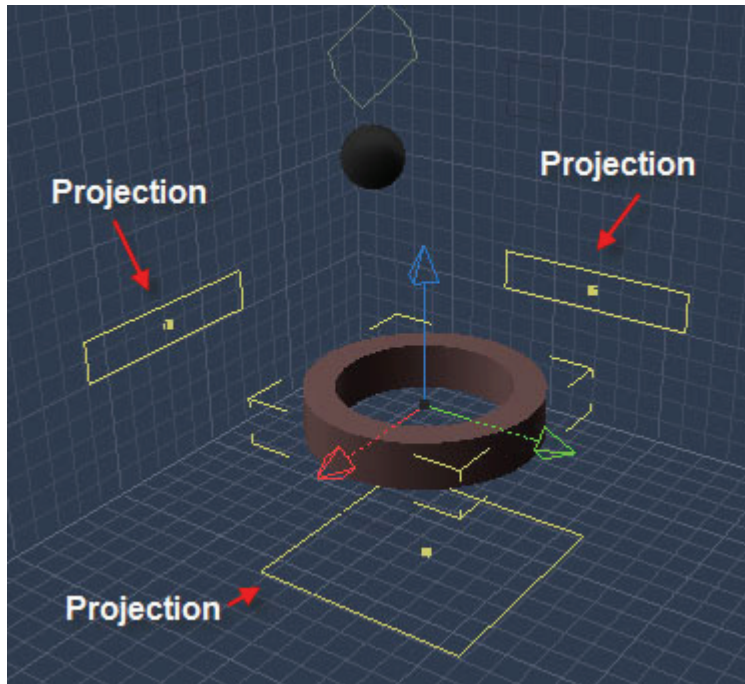
Choosing the Active Plane

You can select the active plane using the Working Box control, located in the left tool area. To set the active plane, simply click the working box control icon on the plane you want to activate. The active plane is highlighted.

Changing the Working Box

Changing the working box does not affect your existing objects. The orientation of the working box applies only to subsequent arrangement operations.You can change the working box in several ways to facilitate the assembly of your scenes:

- Change plane colors to suit your personal preferences. For more information, refer to "Color Pickers."

- Change the size of the working box to simulate the real-world size of your scene.

Aligning the Working Box to an Object

The working box is moveable and scalable, so you can set it at different angles, change its size, and move it away from the origin. In some cases, you might want to move the working box into other positions.

You can move the working box so it aligns with an object or group of objects. For example, if you are creating a spaceship, and one of the wings is at an angle to the rest of the ship, you might want to work in that angled coordinate system to easily maneuver objects relating to the wing.

This feature aligns and repositions the working box around a selection, which is useful for creating large scenes where you would otherwise run out of grid. It is also useful for creating an object or a motion path that is set at an angle to the overall orientation of the scene.

To send the working box to an object or group:

1. Select an object or group.

2. Choose **View > Send Working Box to Object**. Carrara sets the working box planes parallel to the bounding box planes of the selected object or group bounding box. The coordinate systems of the working box and the object (or group) are aligned.
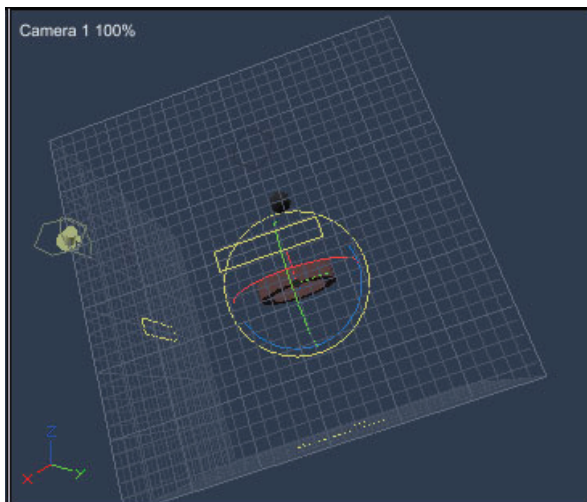
You'll find it particularly useful to move the working box when you want to model or view a specific part of a vertex object. You can also drag the planes of the working box separately. If you drag a plane far enough, you will drag the working box along with it, re-centering the working box in the universe.

Aligning the Working Box to Specific Coordinates

To move the working box to specific coordinates:

1. Choose **View > Send Working Box to**. The Send Working Box dialog appears.

2. Enter the coordinates where you want to position the center of the working box.

3. Click **OK**.

Returning the Working Box to its Origin

When you center the working box in the global universe, it becomes useful for positioning objects in your scene. By using real-world units of measure and centering your scene at the universe's 0,0,0 coordinates, you always have an idea of where your objects are in relation to the overall scene.

To return the working box back to the global coordinate system, choose **View > Send Working Box to Origin**.
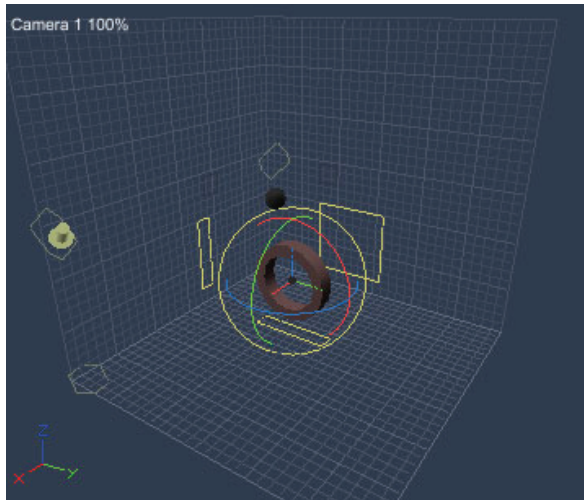
Changing the Grid

Each plane in the working box has a grid. Each grid represents an axis in three dimensional space: X, Y, and Z.

Showing and Hiding Grid Planes

Three grid plane controls appear in the upper right corner of the document window. The three visible grid planes of the working box shown in the different views represent the XY, ZX, and ZY planes. You can choose which planes to display.
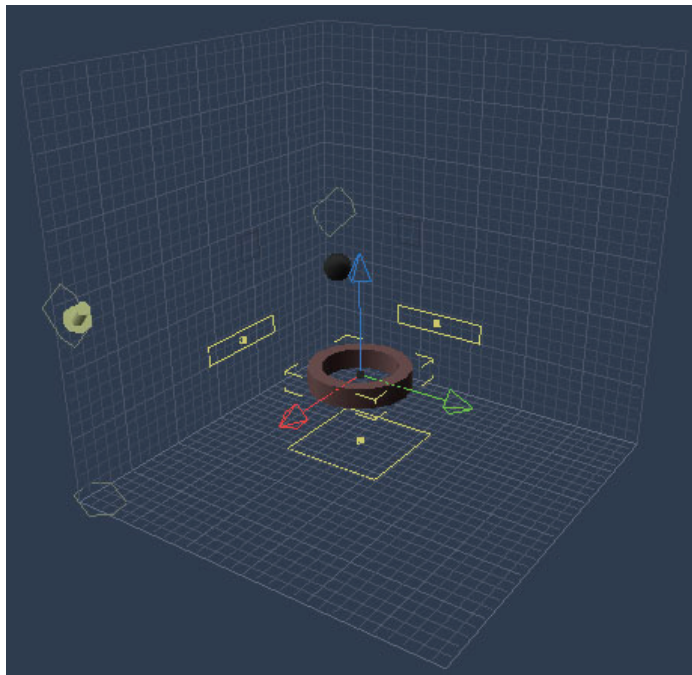
The working box is made up of three grids that represent the X, Y, and Z axes.

You can display or hide any of the grid planes in the working box as follows:

- Click on the **Grid Controls** on the top of the document window.

- **Option/Alt+click** on a plane in the Working Box control. The specified plane is hidden or displayed.

Setting up the Grid

The grid in Carrara can be an invaluable tool for placing objects in 3D space.

To set up the grid:

1. Choose **View > Grid .** The Grid Settings dialog appears. Options are as follows:

   o **Center:** Enter the X, Y, and Z axis coordinates for the center of the grid.

   o **Size:** Enter the X, Y, and Z dimensions to size the grid.

   o **Draw a line every:** Controls how often grid lines are drawn. For example, a value of 1 draws a grid line for every grid point, as set in Spacing below. A value of 2 draws a grid line at every other grid point.

   o **Spacing:** Adjusts the spacing between grid lines in the box and specifies units of measure. For example, a value of 4 creates four feet of space between each grid line if you are using feet for units.

   o **Snap to Grid:** Causes objects to "snap" to the nearest grid line as you drag them. Another and faster way to turn on or off this option is available in **View> Snap To Grid**.

2. Choose **OK** to apply your settings.

In the example above, a value of 1 would draw a line every 4 inches.

Choosing a Preview Mode

Click an object preview tool in the Assemble or Storyboard rooms to view your work in that preview mode. The more detailed previews take longer to render.

Click one of the **Preview Quality** icons to change the preview quality of the objects in the Assemble room. Click on the arrow to open the interactive renderer preferences dialog and to have more options.

You can preview objects in the Assemble room at five different levels of quality: Bounding Box, Wire, Flat, Gouraud, and Textured (or Phong). The various preview modes are described in the sections that follow.

To increase application efficiency, you might want to work in Wireframe or Flat mode at the outset of a project, then switch to Textured mode as specific details become important.

Higher preview modes take longer to redraw, so it's a good idea to use a Wireframe or Flat mode when you're arranging objects. Switch to the higher modes only as specific shading detail becomes more important.

Use the Test Render tool whenever possible, to preview your work.

To change the preview quality, choose the **Preview Quality tool** you want:

Bounding Box

The **bounding box** is the smallest box that encloses an object or group of objects. Bounding boxes appear around objects that are selected.
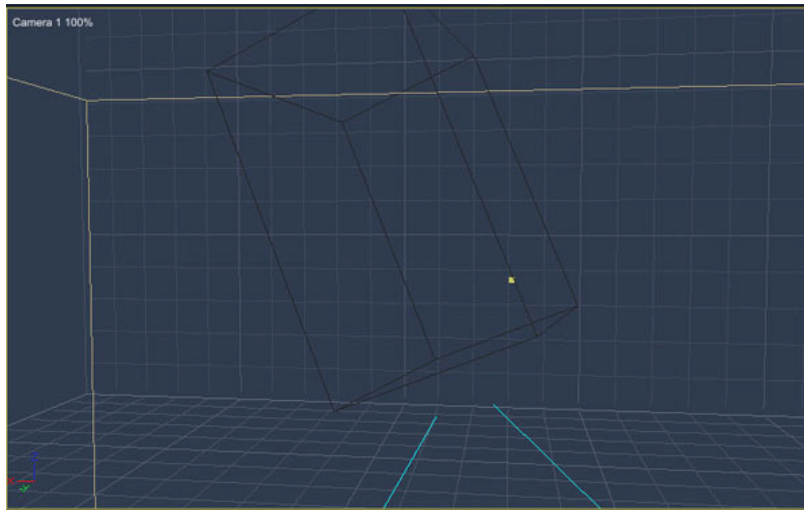
Wireframe

Wireframe mode displays objects as a mesh of wires.
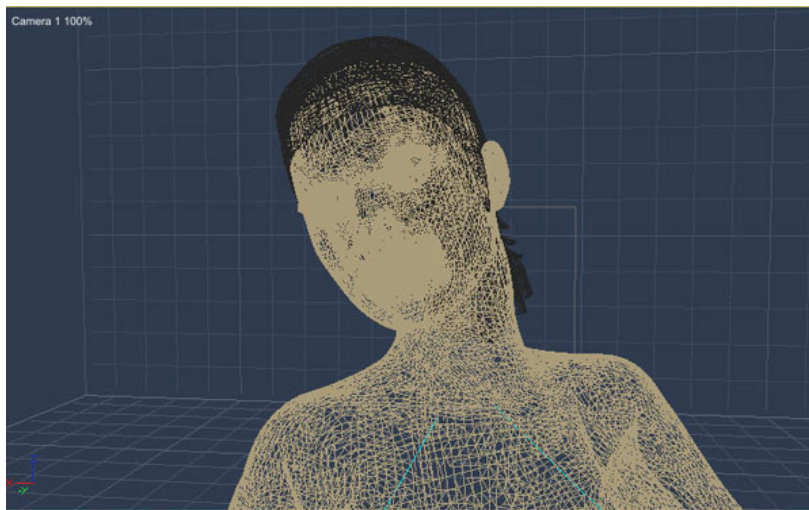
Flat

Flat mode displays objects with low faceting and colored surfaces.

Gouraud

Gouraud mode shows objects with shading at a low level of detail.

Textured

Textured (Phong) mode shows objects with shading, texture maps, and paint regions in detail., but takes longer to calculate and draw.

Using the Properties Tray

The Properties tray is split into two parts. The upper portion of the Properties tray contains several tabs in the Assembly room: General, Motion, Modifiers, and Effects. As you select different objects, you'll notice that the content of the upper part of the Properties tray changes to reflect the options available for that object.

The controls available in the Properties tray change as you select different items. For example:

- When you select an object in the document window, the tray displays controls to change the object's name, position, modifiers, shading, links, and rendering settings.

- When you select a light, the tray displays the light's properties and controls for changing the light type, color, and other properties.

- When you select a camera, the tray displays the camera's name, position, and camera type properties.

- When you select the scene from the list in the Properties tray, you can set scene level properties.

- When you select a point in the Spline modeler, the tray displays the point's position and angle.

- When you select an object in the Vertex modeler, the tray displays controls for adjusting its numerical position.

- When you select DAZ Studio or Poser content that contains morphs, morph dials are located in the lower portion of the Properties tray.

You can expand or collapse the options displayed in the Properties tray. In some instances, you can expand or collapse several levels of options.
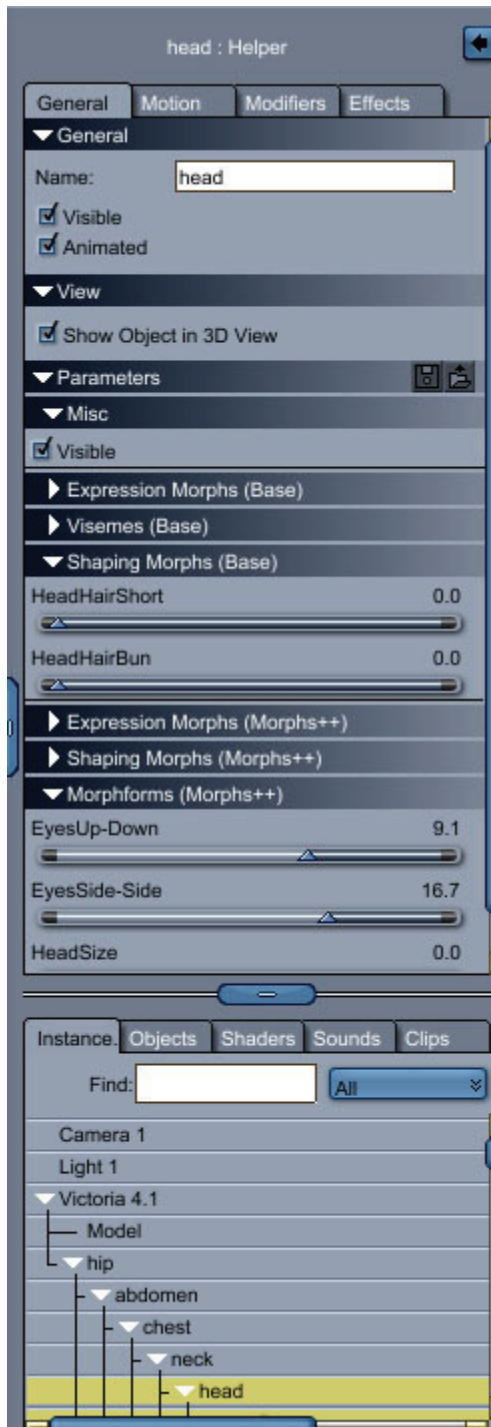
The lower part of the Properties tray contains several tabs that help you manage your scene.

The Instances tab shows the same hierarchy as in the sequencer. When you are working with complex scenes, you might find it easier to select small or hidden objects in the Hierarchy list, rather than in the document window.

To use the Properties tray:

1. Use one of the following methods to show the Properties tray:

   o Choose **Windows > Properties**.

   o Click the handle for the Properties tray and drag it open.

   o Do a short click on the Properties tray handle.

2. To switch categories in the Properties tray, click on the tabs to switch between the different property options: General, Motion, Modifiers, or Effects.

   o Click the arrow to the left of a section name to expand or collapse the contents of that section. The arrow points down to show a section has been expanded, or sideways to show that the section has been collapsed.

   o If needed, drag the scroll bar up or down to scroll the contents of the tray.

   o In some sections, you can click the downward-pointing double arrow next to a section name to display a list of choices.

   o The currently selected choice displays to the left of the double arrow.

   o If you have selected Modifiers, click the + or - sign to add or remove modifiers.

   o Once you have added modifiers, you can expand or collapse each one.

3. The tabs in the lower section of the Properties tray allow you to select or browse through the objects in your scene:

   o Click the Instances tab in the Properties tray to display the hierarchy of all the instances in your scene. The first line, labeled Scene, displays properties for the entire scene.

o The Objects tab displays previews of the Master Objects. You can edit the name of the object if desired. For more information about the Master Objects, refer to "Working with Master Objects."

o The Shaders tab displays previews of the Master Shaders in the scene file.

o The Sounds tab displays the list of the Master Sounds in the scene file.

o The Clips tab displays a list of the non-linear animation clips in your scene.

4. To close the tray, use one of the following methods:

    o   Choose **Windows > Properties**.

    o   Drag the Properties tray shut.

    o   Do a short click on the handle.

Choosing Interactive Renderer (3D Preview) Settings

The **Properties tray: Preview tab** appears when you are in the Model or Texture rooms. It shows a 3D preview of your scene.

The 3D Preview window (3D View) is also known as the **Interactive Renderer**. This preview window has a number of options that can be used to customize the display of your scene. You can select a preview option to optimize for speed or for image quality. For example, use Bounding Box to optimize for speed while initially creating your scene and positioning objects.

You will want to set these options based on your preference for working and the capabilities of your system. You will probably want to experiment with different settings to find what is right for you. The available preview preferences are based on the renderer you are using.

To open the Interactive Renderer Settings dialog, do one of the following:

• Click the first button that appears in the Preview controls section of the document window tools (the arrow in circle symbol).
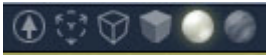
• Click the first button that appears in the 3D Preview window in the Properties tray: Preview tab.
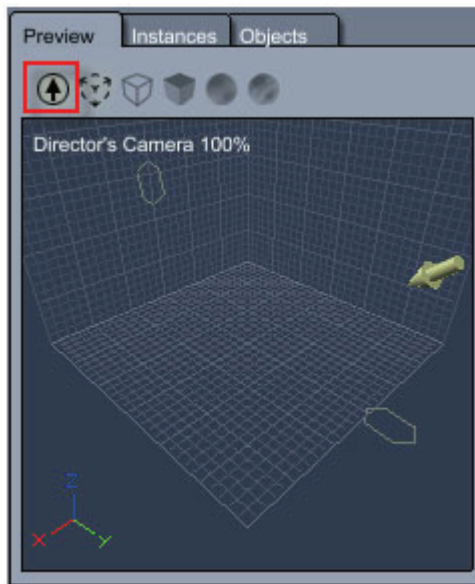
The Interactive Renderer Settings dialog will come up.
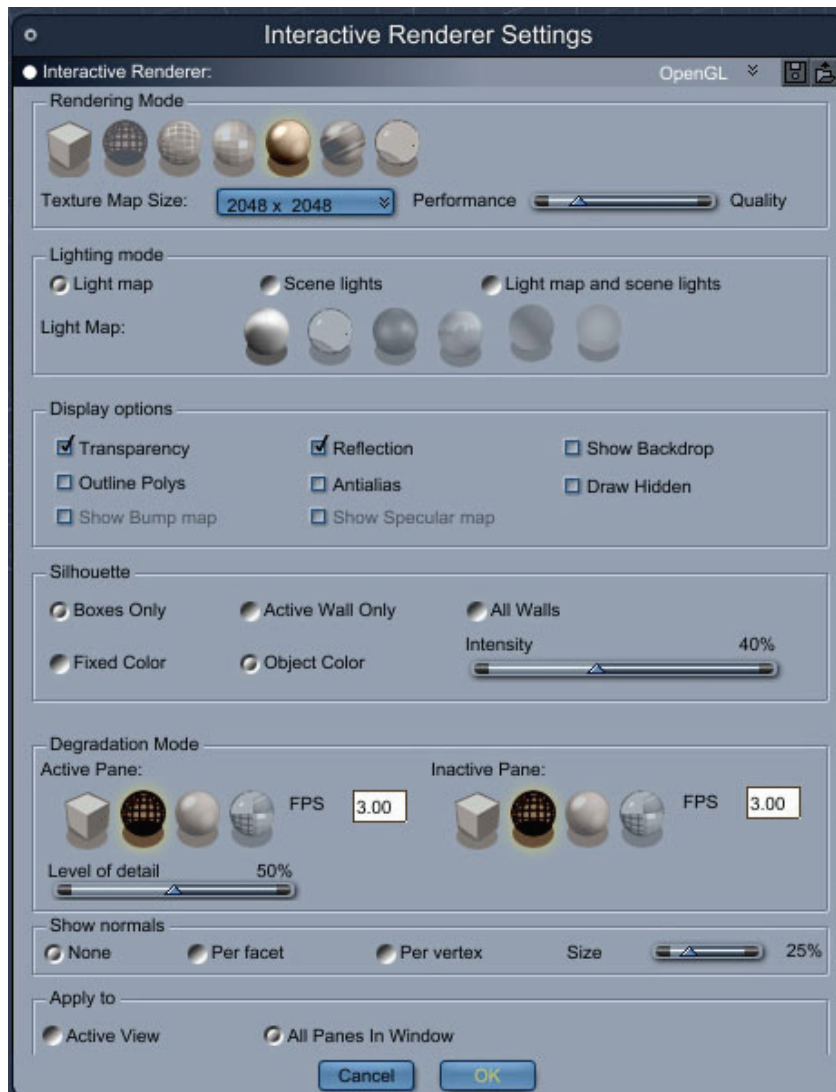
When you open the Interactive Renderer Settings dialog, you are offered several different options.

- **Rendering Mode**: You can choose between several rendering modes (bounding box, wireframe, Gouraud, textured...) Refer to "Choosing a Preview Mode" for details about each of the rendering modes. Select textured if viewing bump and specularity in the 3D view.

    - **Texture Map Size**: Choose the size of the texture map used in the rendering mode. Software mode can use up to a 512x512 map size while OpenGL mode can use up to a 2048x2048 map size.

    - **Performance/Quality**: Use the slider to select better performance or better quality while in OpenGL mode.

- **Lighting Mode**: Determines the lighting mode used in the 3D view.

    - **Light Map**: Uses a light map to light the scene in the 3D view.

    - **Scene Lights**: Uses the lights in the scene to light the 3D view.

    - **Light Map and Scene Lights**: Uses both a light map and the scene lights to light the 3D view. Only available in OpenGL mode.

- **Display Options**:

    - **Transparency**: Activates the display of transparency in the 3D view.

    - **Reflection**: Activates the display of reflections in the 3D view.

    - **Show Backdrop**: Select this option to display the backdrop image in the back of the 3D view.

    - **Outline Polys**: Shows the outline of polygons in 3D view.

    - **Antialias**: Turns on antialiasing of wireframes in the 3D view.

    - **Draw Hidden**: Draws hidden wireframes in the 3D view.

    - **Show Bump Map**: Shows the bump map when in the 3D view and using OpenGL mode. Requires a bump map layer to have been applied. (Pro Only)

    - **Show Specular Map**: Shows the specular map when in the 3D view and using OpenGL mode. Requires a specular map layer to have been applied. (Pro Only)

        NOTE! When in Software mode or if your video card does not support viewing bump and specular maps, then the Show Bump Map and Show Specular Map options will not be visible.

- **Silhouette**: Determines the style of objects as projected onto the working box. Choices are **Boxes Only**, **Active Wall Only**, **All Walls**, **Fixed Color**, and **Object Colo**r.

- **Degradation Mode**: Select to determine which mode the interactive renderer will switch to when the redraw speed drops under a certain threshold (expressed in frames per second (FPS)).

    - **Active Pane**: Display mode used in the active pane when the degradation mode is triggered.

    - **Inactive Pane**: Display mode used in the other panes when the degradation mode is triggered.

    - **Level of Detail**: Level of details used for the geometry when the degradation mode is triggered.

- **Show Normals**: Displays the normals of the objects. Options are **None**, **Per Facet**, and **Per Vertex**. The **Size** slider is used to adjust the size of the normals.

- **Apply To**: Determines which settings are modified by the dialog. You can apply the settings to the Active View or to All Panes in Window.

To set preview options:

1. Click the **Interactive Renderer Settings** control. The Interactive Renderer Settings dialog is displayed.

2. Select the renderer (Software or OpenGL) from the menu at the top of the dialog.

3. Select the rendering mode you want to use. See "Choosing a Preview Mode" for more information on the different rendering modes.

4. Enable the features that you want to use and disable those that you don't want. These additional options are described in the sections that follow.

5. Click **OK**.

Using the Sequencer Tray

The Sequencer tray supplies animation tools and different types of information specific to the room you are currently using. As you work, you'll find the Sequencer tray an invaluable tool for arranging and animating objects in your scene.

You can view only the animation tools by partially opening the tray.

Drag or double-click the tray handle to display or hide the contents of the tray.
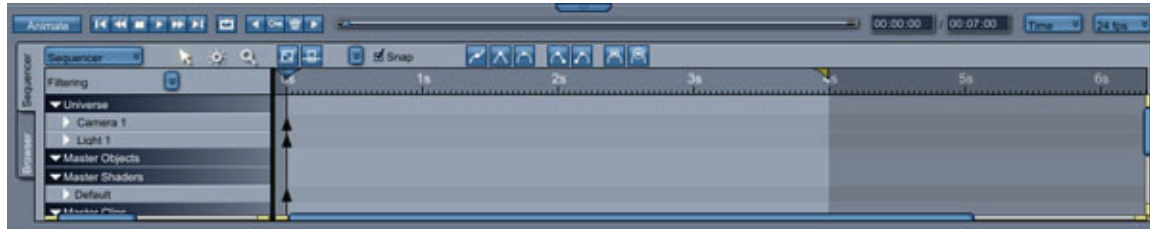
Other than in the Render room, the Sequencer tray merges two types of information: time line and hierarchy. In the Time Line section of the Sequencer tray, you have control over animating the events in your scene. You can display only the basic animation tools, or open the tray further and show the Time Line and Hierarchy information as well.

In the Hierarchy section of the Sequencer tray, you'll see an expanding/collapsing visual hierarchy that provides a display of how elements are grouped and linked.

The active window and the Sequencer tray are synchronized. As you add or remove objects from one, the display in the other updates automatically. Note that any object selected in the Sequencer is also selected in the document window.

The list in the Sequencer tray can be filtered to display information specific to the type of task you're working on.
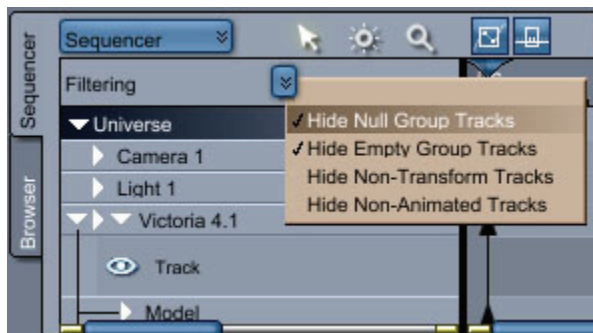
The Sequencer supplies certain information on the scene's construction not immediately apparent in the Assemble or Storyboard rooms: for example, how several elements can be grouped.

When you're modeling or arranging objects, you'll need to see information on grouping and linking. For this task you can drag the tray all the way open to display the Hierarchy section of the Sequencer. The Hierarchy section displays all objects, groups and scene elements in an expanding/collapsing indented list, along with animation timing information.

When you're creating an animation you must see key frames and tweeners. For this task you can use the Time Line information in the tray.

To display the Hierarchy and Time Line information in the Sequencer, drag it all the way open until the Hierarchy section of the Sequencer is displayed.

When the Sequencer tray is fully open, you can see the key frames and tweeners in your animation.

1.   Click the drawer handle of the Sequencer tray to make it active.

2.   Drag it all the way open until the Hierarchy section of the Sequencer is displayed.

Refer to "Building a Hierarchical Structure" for complete instructions on using the Hierarchy.

Editing Your Scene's Contents

Carrara has several of the basic operations you're familiar with from other applications. You can cut, copy, and paste any item in the document window using the Edit commands.

To copy an item in the document window:

1.   Select the object, light, or camera.

2.   Choose **Edit > Copy**, or press **Command/Ctrl+C**. A copy of the item is placed on the Clipboard.

To cut an item in the document window:

1.   Select the object, light, or camera.

2.   Choose **Edit > Cut**, or press **Command/Ctrl+X**. The item is removed from the scene and placed on the Clipboard.

To paste an item in the document window:

•   Choose **Edit > Paste**, or choose **Command/Ctrl+V**. The object appears in the Assemble room.

To delete items in the document window:

1.   Select the item.

2.   Choose **Edit > Delete**. The object is deleted from the scene.

There are many other ways you can edit the contents of your scene. You can duplicate, move, rotate, and re size objects. Refer to "Arranging Objects" for more information.

Undoing Operations

You can reverse the effects of your last action by using the Undo command. Carrara has multiple undo levels so that you can undo a series of operations by choosing undo several times. The number of undo levels is determined by the value you set in the Carrara Preferences dialog. The maximum number of undo/redo levels is 64.

To set Undo levels:

1.   Choose **File > Preferences**.

2.   Choose **Application** from the pop-up menu.

3. Enter a value in the **Maximum Undo Levels** entry box.

To undo operations:

1. Choose **Edit > Undo**, or press **Cmd/Ctrl+Z**. Carrara undoes the last operation.

2. To undo the next operation, choose Undo again.

NOTE! There are some operations you can't undo. In this case, the **Undo** command is replaced by **Can't Undo**.

Redoing Operations

You can repeat your last action using the Redo command. The command is available only after you undo an operation. You can redo multiple operations.

To repeat an operation:

- Choose **Edit > Redo**, or press **Cmd/Ctrl+Y**. Carrara repeats the last action.

Saving Your File

To save your scene:

1. Choose **File > Save**, or **Cmd/Ctrl+S**.

2. Choose a location for your file.

3. Enter a name for the file in the **File name** entry box.

4. Choose a file type from the **Format/Save as type** pop-up menu and click **Save**.

To save your file under a different name:

1. Choose **File > Save As**, or press **Cmd/Ctrl+Shift+S**.

2. Select a location for the file.

3. Enter a new name for the file in the **File name** entry box and click **Save**.

Closing Your File

When you're done editing a scene file you can close it.

To close a file:

1. Choose **File > Close**.

2. If you have any changes, click **Yes** to save them or **No** to discard them.

Using Content Files

Carrara allows you to work with many different types of content. Along with the many scenes and objects that are provided with Carrara, you can also create your own content. The Browser tray stores this content so that you can easily add it to any scene that you create.

This chapter provides an overview of the Browser tray and how you can use it to manage and use your content as you build your scenes and animations.

The Browser Tray

The Browser is a visual catalog of many of the elements you can use to create a scene. The Browser has sections for each different category of element. Each section can display multiple directories.

The Browser tray is composed of the following categories: Scenes, Objects, Shaders, Clip, Misc. (Modifiers, Constraints, Effects), Artwork, and Content. Each category lets you organize items you save into directories.
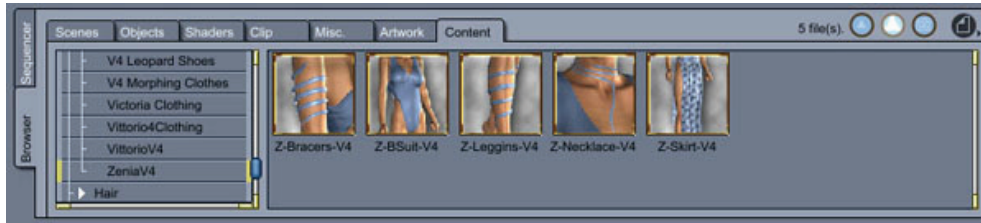
The Browser is a visual catalog of some of the items you can use to create 3D scenes.

After you create or customize an item in one of these categories, you can save it to the Browser, where it will be easy to retrieve and use again later. Most operations with the Browser are the same in all categories.

NOTE! Objects, light sets, and cameras are saved as normal Carrara documents (.CAR files). Items in other Browser categories are saved as special Browser documents (.CBR files).

Showing and Hiding the Browser

The Browser tray appears at the bottom of your screen. If you need more room to create or edit your scene you can hide the Browser tray.

To show or hide the browser, do one of the following:

- Choose **Windows > Browser**.

- Drag the Browser tray by its handle to open it.

- Click the Browser tray handle.

Choosing Your Display Mode

The Browser offers three view modes for its contents: Text, Small Icon, and Large Icon.

To choose a display mode for the Browser contents:

1. Locate the **View icons** in the upper-right corner of the Browser tray.
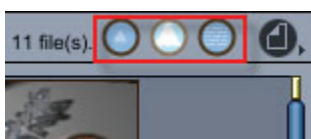
2. Click the view icon for the desired style:

- **Small Icons**: Displays half-size icons with a text label beneath.
- **Large Icons**: Displays full-size icons with a text label beneath.
- **Name Only**: Displays no icons, only text labels.
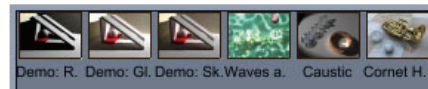
Small Icons
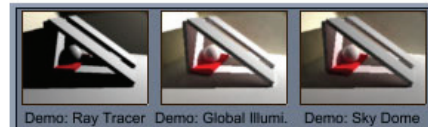
Large Icons
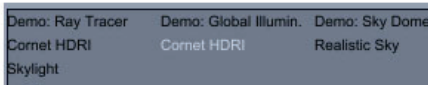
Name Only

If a folder contains more items than the Browser tray can display, use the scroll bar at the right side of the Browser tray to show additional items.

Using the Browser File Menu

The File menu is located in the upper-right corner of the Browser tray. It contains commands that help you manage your content and content folders.
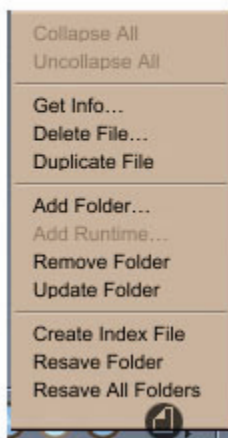
Use commands in the Browser's File menu to manage your content and content folders.

Displaying Browser File Information

Any item that appears in the Browser tray is saved as a separate document. When you display file information, you can view the name and type of file, and see the path to the folder that stores the file.

To get information on a Browser tray item:

1. Click the desired tab in the Browser.

2. Select the item for which you want to view information.

3. From the Browser menu, choose **Get Info.** A dialog provides Name, Type, Path, and Comment information on the selected item.

4. Use the Name entry box to change the name, and optionally add a comment as a reminder of how to use this particular item.

Deleting Browser Files

The Browser menu includes a command that allows you to delete files from the Browser. When you use this command, it will also delete the file from your hard drive.

To delete a Browser item:

1. Click the desired tab in the Browser.

2. Select the item that you want to delete.

3. From the Browser menu, choose **Delete File.**

4. Carrara asks you to confirm the deletion before the file is removed. Click **OK** to delete the file.

Duplicating Browser Files

There might be cases when you want to keep an existing Browser item, but use it as a starting point to create another similar item. For example, you might want to make some slight changes to a shader and save the modified version as a new Browser item. You can duplicate the existing item and make changes to the copy.
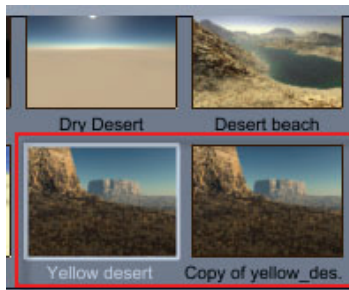
To duplicate a Browser tray item:

1. Click the desired tab in the Browser.

2. Select the item you want to duplicate.

3. From the Browser menu, choose **Duplicate File**.

4. Carrara creates a duplicate of the selected item, naming it **Copy of (original object name)** by default. You can now rename and open the duplicate for editing.

Editing Browser Files

Double-clicking a Shader, Modifier, Constraint, or Effects item in the Browser opens the appropriate editor for the item. For example, you can double-click a Shader in the Browser to open it in the Shader Tree Editor.

To edit a Browser item:

1. Click the desired tab in the Browser.

2. Double-click the item you want to edit. Carrara opens an editable item as follows:

    o For an object, camera or light, Carrara opens it in a scene.

    o Carrara opens all other items in a document window, which contains the tools appropriate to its type.

Adding Browser Folders

Some of the Browser tabs include folders in which you can store your own content. For example, you'll find empty My Objects, My Shaders, My Clips, and My Artwork folders at the top of the Objects, Shaders, Clips, and Artwork tab folder lists. These folders are all located within the Carrara installation folder hierarchy.

You can also add other folders that reside on any drive in your system. While doing so, Carrara asks which tab will include your new folder.

To create a new folder in the Browser tray:

1. Click the desired tab in the Browser.

2. From the Browser menu, choose **Add Folder**. The Browse for Folder dialog appears.

3. Locate the folder that you want to add to the browser. Highlight the folder and click OK.

4. If the selected folder contains different kinds of items, a dialog asks you to choose the types of files that you want to display in the selected Browser tab. Choices are **Objects**, **Terrains**, **Clouds**, **Particles**, **Plants**, or **Leaves**.

5. Select the appropriate option and choose OK. The new folder appears at the bottom of the folder list in the tab that you selected.

Removing Browser Folders

Use the Remove Folder command to remove a folder without deleting the objects from your hard drive. This action is not undoable.

To remove a folder from the Browser tray:

1. Click the desired tab in the Browser.

2. Click the folder title you want to remove. Selected folders are highlighted.

3. From the Browser menu, choose **Remove Folder**. Carrara removes the folder column from the Browser.

Updating Browser Folders

When you change the contents of a Browser folder, you might need to force Carrara to build a new list of the contents.

To update listings in a folder:

1. Click the desired tab in the Browser.

2. Click the folder title you want to update. It will be highlighted.

3. From the Browser menu, choose **Update Folder**.

Resaving Browser Folders

The Resave Folder and Resave All Folders commands update all files contained in all or the currently selected folders.

To resave all files in one or all folders:

1. Click the desired tab in the Browser.

2. Choose one of the following:

   o To resave all items in a single folder, select the folder you want to update and choose **Resave Folder**.

   o To resave all items in all folders, choose **Resave All Folders.**

3. Carrara asks if you want to overwrite the files. Choose **OK** to proceed.

Browser Tabs

There are several different tabs in the Browser tray, each of which holds a different category of files. This section gives an overview of the various types of content that you find in the Browser.

Browser Tray: Scenes Tab

The Scenes tab stores complete scenes in several different categories. You'll find scenes that provide the following types of examples:

- Animations

- Global Illumination

- Indoor and Outdoor scenes, including space, landscapes, and skies

- Lighting examples, such as global illumination, lighting, and lighting effects

- Logos

- Packaging
- Special Effects

To access the Scenes Browser, click the **Browser tray: Scenes tab**.

To load a scene from the Browser tray:

1. Click the Scenes tab in the Browser tray.
2. Navigate through the category list until you find the scene you want to open.
3. Double-click the desired Small, Large, or Name Only preview in the browser tray. The scene opens in the document window.

To save a scene to the Scene category in the Browser:

1. Make sure the scene you want to save is open.
2. Choose **File > Save As**. The Save As dialog appears.
3. Select one of the Scenes subfolders in your Carrara installation folder. For example, if you want to save an animation, choose the **Scenes: Animation** subfolder, in which existing animation scenes appear.
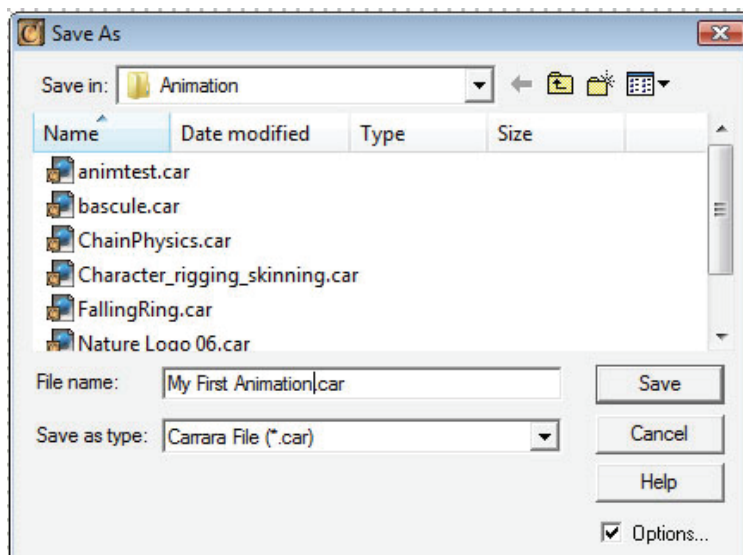
4. Click the **Options**. checkbox.
5. Click **Save**. The Carrara Export dialog appears.

6. Choose where you would like to save your texture maps. The options are:

   o **Save all internally**: Saves all textures internally with the file.

   o **Use local settings**: Uses the settings of each individual texture to determine whether they should be stored inside or outside the file.

   o **Save all externally**: Saves all textures in separate files.

7. Enable the Save Scene Preview option to generate a thumbnail for the Browser tray.

8. Check or uncheck the **Compress File** option as need to save a compressed or uncompressed version of the file.

9. Click **OK**. Your scene is saved with a .car file extension.

Browser Tray: Objects Tab

The Objects browser stores items that you can add to a scene that is already opened in your document window. There are a wide variety of objects arranged in many different folders, including, but not limited to, the following:

- Nature objects such as Deserts, Plains, Mountains, Oceans, Plants, Leaves, and Clouds.

- Particle objects.

- Basic and Advanced modeled objects created with Carrara's modeling tools.

- Formulas

- Hair

- Manmade objects such as Art, Buildings, Cities, Communication devices, and other household and industrial items

- Fashion accessories such as jewelry, belts, earrings, and more

- Transportation vehicles (land, sea, and air)

To access the Objects Browser, click the **Browser tray: Objects tab**.

To add an object from the Sequencer or the Properties tray by dragging them directly into the Browser tray.

1. Create or open a scene.

2. Click the **Objects** tab in the Browser tray.

3. Navigate through the category list until you find the object you want to add to your scene.

   - Use one of the following methods to add the object to your scene:

   - Double-click the desired Small, Large, or Name Only preview in the Objects browser tray to place the object in the center of the scene.

4. Drag the preview from the Browser tray. You can drag it into the Assemble or Storyboard room, into the Sequencer or the Instances tab of the properties. If you drop an object into the Sequencer, it appears at the origin (X=0, Y=0, Z=0). If you drag it into the Assemble or Storyboard room, it appears at the point where you release the mouse button.

To save an object to the Objects category in the Browser:

1. Click the Objects tab in the Browser tray.

2. Use the folder list to select the folder that will store your item.

3. Go to the Sequencer Tray, or to the Properties tray: Instances tab.

4. Click the name of the object that you want to add to the Browser.

5. Drag the item's name to the Browser tray.
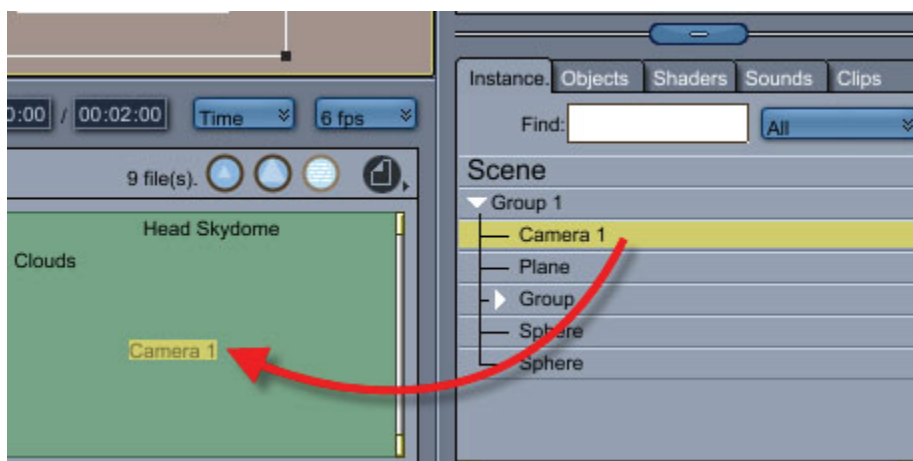
6. Carrara opens a dialog so that you can name the file containing the saved item. Enter a name and click OK. The object is saved with a .car file extension.
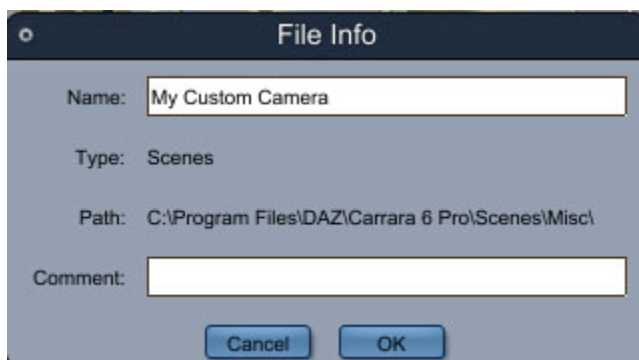
NOTE! Objects are stored in several subfolders that appear beneath the Presets folder in your Carrara installation directory.

Browser Tray: Shaders Tab

Shaders are used to define the appearance of the objects in your scene, and can range from simple colors to more complex materials that include textures, reflections, displacement and more. When you

create your own shaders (see "Creating and Editing Shader Trees"), you can save your custom presets to the library and load them from the Browser Tray.

Library shaders are stored as separate files and can be loaded into the Browser:

- When you drag a library shader from the Browser into the Sequencer, Carrara makes a copy of that shader.

- When you drag a shader from the Browser onto an object, Carrara makes a copy of that shader and applies it to the object.

- In either case, the shader becomes a part of the scene and is referred to as a **Master Shader**.

Don't overload the Browser tray with shaders. Each shader thumbnail requires some memory. Too many shaders loaded into the Browser could impact your system's performance. It's a good idea to limit the directories listed at any given time.
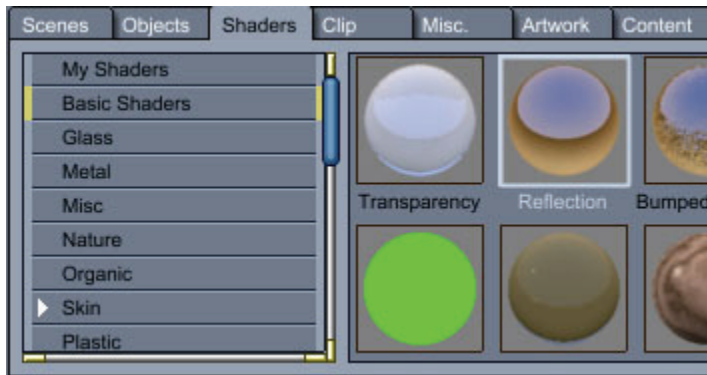
You can use the Browser to organize and store your personal shader library.

You might want to organize your shader directories as logical categories. For example, a list of directories might read: Wood, Rocks, Plastic, Glass, Marble.

You can set up the Browser to display a flat or spherical preview of the shaders. As with other items in the Browser, you can add and remove shaders from the tray.

You can also view shaders contained in a scene by clicking the Shader button in the Sequencer. Remember, the shaders you use in a scene are not automatically listed in the Browser. If you create or use a shader in a scene that you want to reuse, be sure to save it to the Browser.

To view shaders in the Browser:

- Drag open the Browser tray, then click the Shaders tab to view the currently installed shaders.

To use a shader, modifier, or constraint from the Browser:

- Drag the item you want to use from the Browser onto the object where you want to apply it.

- You can drag onto the object preview in the Assemble or Storyboard room or onto the listing in the Sequencer.

- You can also drop the item onto the appropriate section in the Properties tray: Instances tab.

Browser Tray: Clip Tab

The Clips tab stores NLA (Non-Linear Animation) clips. For more information on these types of clips, refer to "Using NLA Clips."
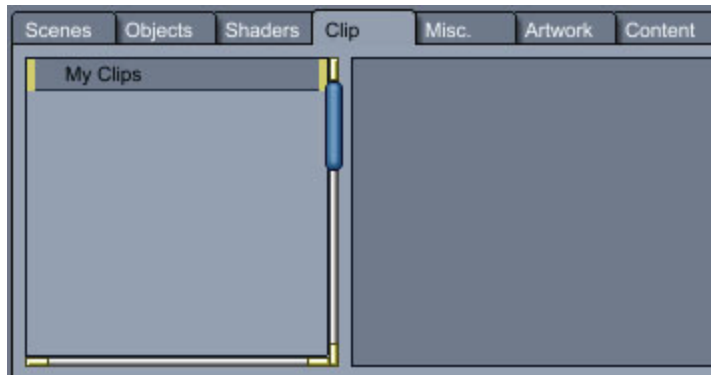
Browser Tray: Misc Tab

The **Browser tray: Misc** tab lets you save constraint settings that you can later apply to your objects.
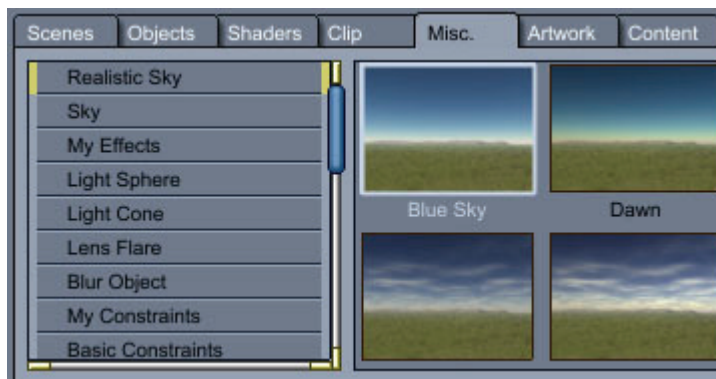
The **Browser tray: Misc** tab lets you save objects, groups, and full scenes that you can use later.

To access the Misc Browser:

- Click the **Browser tray: Misc tab**.

Browser Tray: Artwork Tab

The Artwork tab allows you to store your renders in the browser. After you render an image, save it to a folder of your choice. Use controls in the Browser menu to add the folder to the Artwork tab. Then save your images to the designated folder.
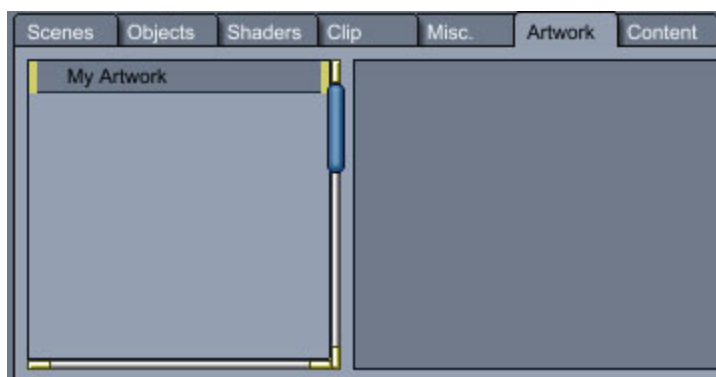
Using Content and Runtime Folders

The Content tab stores content that is compatible with DAZ Studio and Poser. Content of this type is pre-rigged and includes poseable figures, clothing, hair, poses, and more.

The Runtime folder structure originated with Poser and is also supported by DAZ Studio. All content purchased from DAZ is arranged in this manner. In order for content to appear in Poser libraries, all content has to appear beneath a folder named **Runtime**. Beneath the Runtime folder are additional library folders (Characters, Props, Hair, and so on) and Textures folders that contain their respective types of content.

When you add Runtime folders to your content browser, you can drag and drop Poser and DAZ Studio-compatible content into your scene. Items in the Characters library include pre-rigged characters and poseable clothing and hair that fit them. Clothing has to be specifically designed for the figure that wears it in order to fit and pose perfectly. Props folders typically contain scenery and non-poseable clothing like hats, glasses, and so on. The Pose folders can contain anything from preset character poses to files that apply preset material settings or textures to the items they are designed for.

Adding Runtime Folders

Carrara allows you to add Runtime content folders to your Content Browser. This provides an easy way to import DAZ Studio and Poser content into your scene. You can add as many Runtime folders to your Content Browser as you like.

To add Runtime folders in the Assemble Room:

1. Go into the Browser and click the Content tab.

2. From the Options Menu, choose **Add Runtime**. The Browse for Folder dialog appears.

3. Locate the folder that contains your Runtime files. Highlight the file and click **OK** or press **Enter**. The folder is added to the list of folders in your Content tab. You can use this procedure to add as many Runtime folders as you like. The following figure shows several different Runtimes as they appear in the Content Browser.
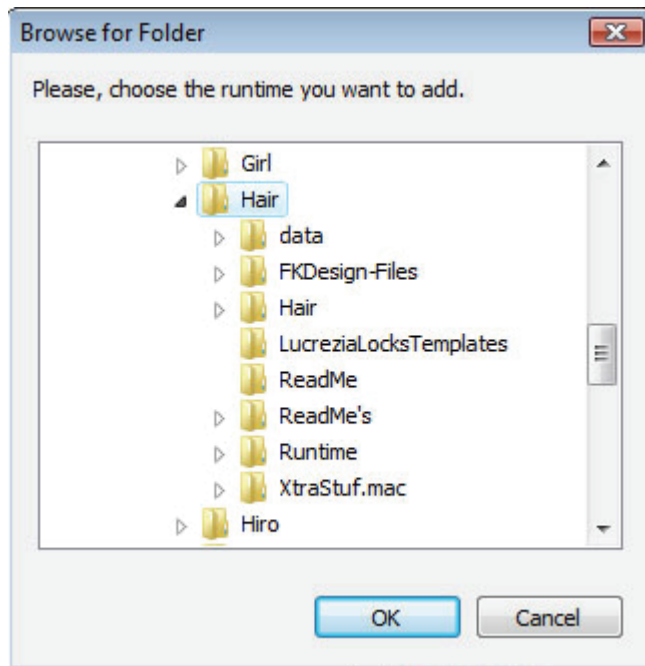
Using Content Runtime Folders

Once you have placed Runtime folders in your Content Browser you can quickly and easily place the content into your scene:

1. Use the Content Browser to expand or collapse the Runtime Folder until you locate the item you want to add to your scene. When a folder contains content, icons will appear in the Content Browser as shown in the following figure.

2. Drag an icon from the Content Browser and add it to your scene. You may be prompted to locate geometry (OBJ) files or texture maps. After you locate the appropriate files, Carrara displays a progress bar during the loading process. The content then appears in your scene.
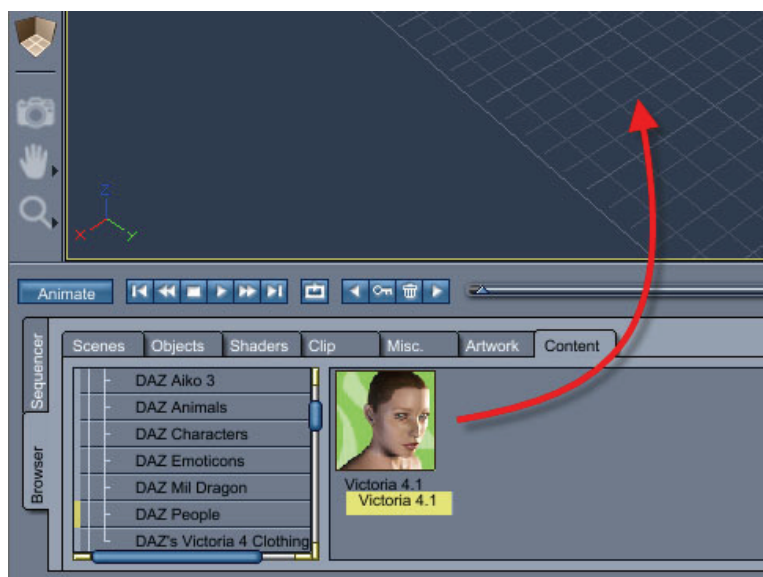
Adding Content: Special Notes

Of particular note are the following points that you should keep in mind when you add content from Runtime folders:

- **Cameras**: Cameras in Poser Runtime folders will be unavailable to add. You will have to set up your cameras within Carrara, or import Poser cameras using the **File > Import** command. For further information, see "Importing Poser Cameras (CM2/CMZ)."

- **Faces** (facial expressions and poses):
    - Drag and drop a Face pose icon over the figure in the scene that you want to apply the pose to.
    - Note that some face poses may contain injection morphs that apply special morphs to the face. See **Injection Poses** (below) for instructions on how to locate them.

- **Figures (Poseable Characters)**:
    - Poseable characters initially appear in their default poses in the default position (center) of the scene.
    - You can later pose your figure using standard Carrara tools or by applying a pre-defined pose from the Content Browser.
    - In Carrara, posing dials that are normally labeled Bend, Twist, Side-Side, Up-Down in DAZ Studio or Poser are found in the **Properties tray: Motion tab: Transform panel**. You can also use Carrara's Scale, Rotate, and Universal Manipulator tools to pose your figures.
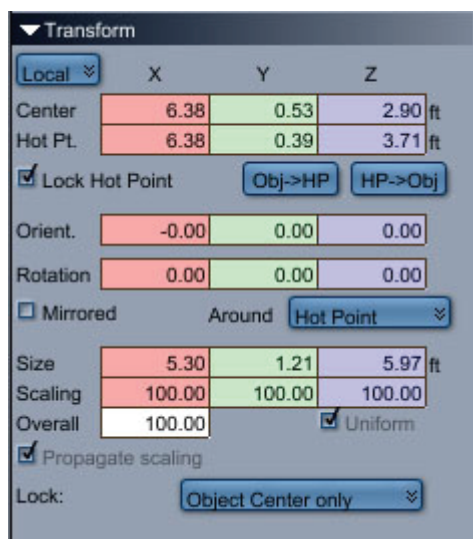
- **Figures (Conforming Clothing)**: Select the icon from the Content tab, and drag and drop the icon over the figure that you want it to conform to. The clothing automatically conforms to the selected figure.

- **Hair Props**: Hair props found in the Hair library folders in your Runtime will automatically parent to the head of the object that you drop the icon over. However, you might find that some hair props needs adjustment to properly position it on the figure. Though it is possible to adjust the position of hair props after a figure is posed, it is easier to adjust when the figure is in its default position.

- **Hand Poses**: Drag and drop a Hand pose over the figure you want to apply it to. Carrara prompts you to select the hand to which you want to apply the pose. Choose **Left** or **Right** and click OK to apply the pose.

- **Lights**: Lights in Poser Runtime folders will be unavailable to add. You will have to set up your lights within Carrara, or import Poser lights using the **File > Import** command. For further information, see "Importing Poser Lights (LT2/LTZ)."

NOTE! Poser lights may need some adjustment in intensity, color, and falloff after importing, and the amount of adjustment may also vary due to changes between Poser versions. Start by reducing the light intensity to 25-30% of the original intensity and then increase or decrease as necessary to get the desired affect.

- **Poses**: Carrara allows you to apply all types of Pose files to your scene. Use icons in the Pose folder to pose your figures, change materials, apply injection morphs, and other types of Poses that you typically also use in DAZ Studio or Poser. Keep in mind that you may be prompted to locate texture maps or injection files (see below) before the pose is applied to your object.

- **Injection Poses**: When you try to add poses that inject morphs into your figures, you may be prompted to locate injection poses on your system. For DAZ figures, these injection files are typically located in special folders that begin with !DAZ, followed by the name of the figure that they are used for. For example, injection files for Victoria 3 are located in **Delta** folders that appear beneath the **Runtime > libraries > !DAZ > Victoria 3** folder, which is further separated into **Body**, **Head** and other subfolders. To view the files in a pose folder, select **Poser Pose** from the Files of Type menu when you are prompted to locate the files.

- **Props**:
  - When loading a prop, you can drag and drop it into the Assembly window. Normally, props are designed to load in the center of the scene and on the floor.
  - Some props are designed to load in a particular place in the scene (for example, sails for a sailboat will load on the mast, or a pair of earrings will load on a figure's ears). To load props in their designed   position, drag the desired prop from the Browser tray and drop it into an empty spot in the Instances tab, or onto the word **Scene** in the Instances tab.
  - To attach or parent a prop to another object, locate the prop you want to attach in the Instance list. Then drag and drop it in the Instance list, releasing the mouse when it highlights the object or body part that you want to assign as the parent to the prop.
  - **Smart Props** (those designed to come in properly placed and parented to a figure) should be dragged and dropped onto the character in the Assembly window, or on to the main entry for the character on the Instances tab.
  - You may be prompted to locate OBJ or texture files before your prop appears in the scene in its default position.
  - After the prop appears in your scene you can use Carrara's Move, Scale, and Rotate tools (or the Universal Manipulator) to position the prop where you want it to appear.

- **Morphs**: When you add content that contains morphs, You'll find the morph dials in the Assemble room. Click the figure or body part that you want to change, and then open the General properties tab. The morphs appear near the bottom of the tab, and may be arranged in groups as shown in the following figure.
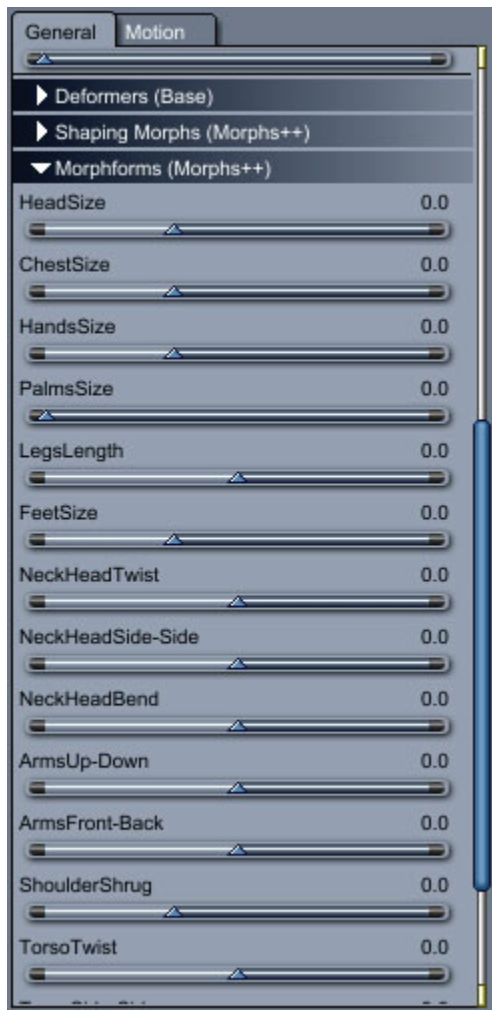
- Expand the morph group, if necessary, to display the morph dials. You can move the morph slider left or right to adjust the setting. You can also click the numerical value of the morph and type in a precise value.

Adding Level of Detail

Carrara provides the ability to use **Level of Detail (LOD)** meshes on a figure which allows the user to:

1. Use lower resolution meshes while working in the scene but render with a higher resolution mesh.

2. Use different resolution meshes depending on how far the figure is from the camera.

In order to use this feature, the figure/object must already have the LOD levels available and each mesh must have the same vertices as the other levels.

To load and use LOD:

1. Load the figure into the scene and make sure the root figure is selected in the Instance tab.

2. In the Properties tray, locate the **Level of Details** section under the Parameters heading.

3. Click on the **Add LOD** button. The **Import LOD** file dialog will come up. Locate, select, and open the first LOD mesh file.
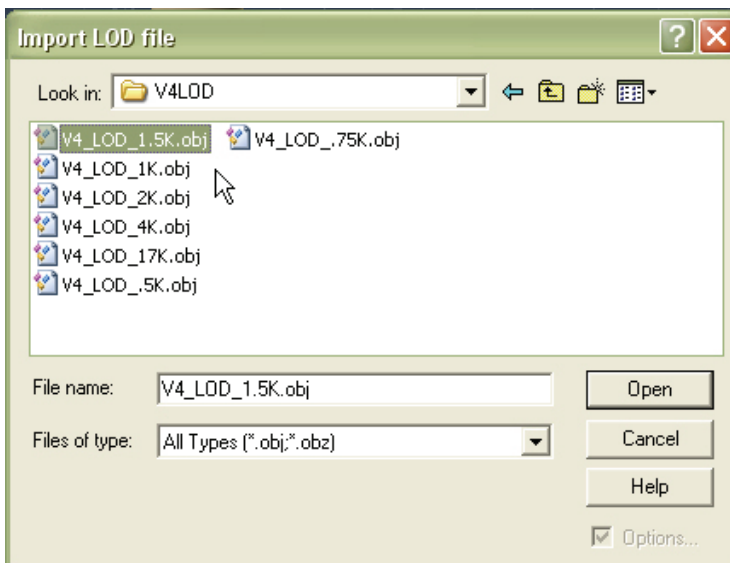
4. The **LOD name** dialog will come up. You can leave the name as is (taken from the mesh name) or rename it. Click on **OK** to accept the name.
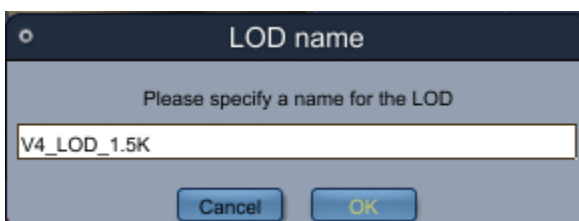
The **Calculating Level of Details** progress bar will come up.

5. If more LOD meshes need to be loaded, repeat until all meshes are loaded.

6. Once all LOD meshes are loaded, click on the **LOD Options** button to access the **Level of Detail Options** dialog.
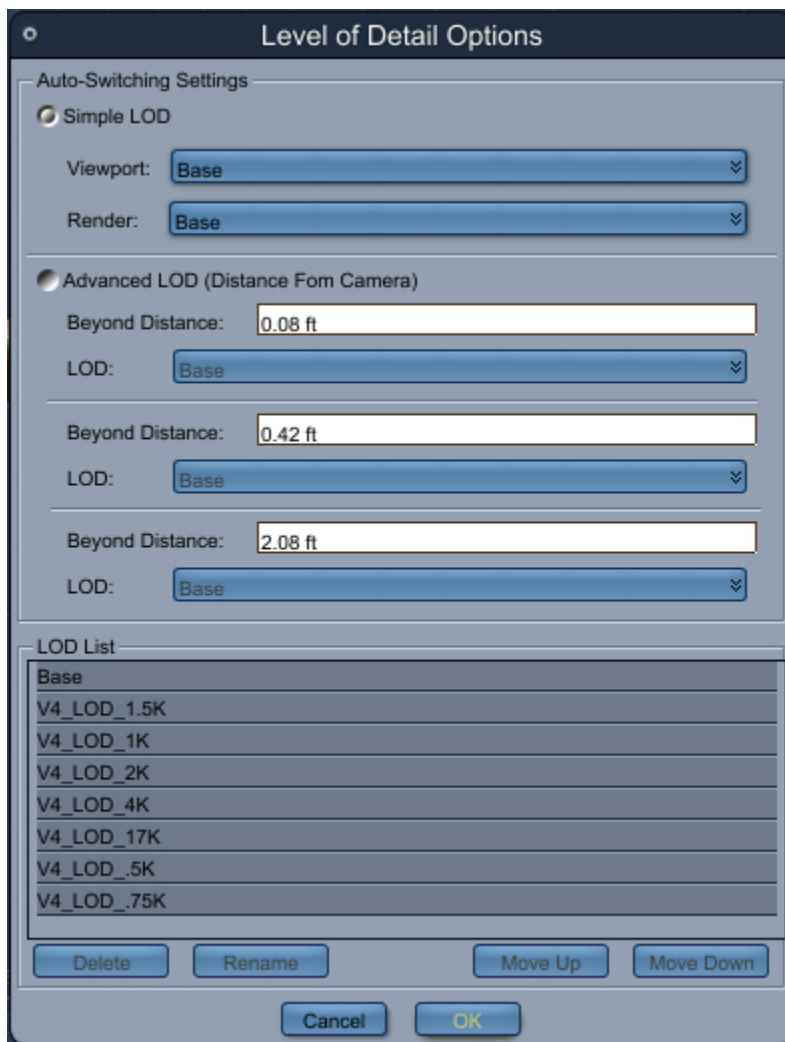
The LOD Options dialog is divided into two sections:

- Auto-Switching Settings: Allows selection of LOD levels based on viewport/rendering or on distance from camera.

- LOD List: Shows a listing of currently available LOD levels with options to delete, rename, and sort.

7. **Simple LOD** - This option is the default selection for using LOD levels. It provides options to select which mesh level to use while working in the scene and which level to use when rendering.

This allows you to set up a scene with a figure with a lower resolution mesh to save on memory usage but switch to a higher resolution mesh for final rendering.

- **Viewport**: Click on the button next to Viewport and select an LOD mesh from the listing. This is the LOD level that will be used as you are working in the scene.

- **Render**: The default render level is the Base mesh, i.e., the highest resolution mesh available. This is the LOD level that will be used when you render the scene. Unless you need to render with a lower resolution mesh, leave the Base mesh selected.
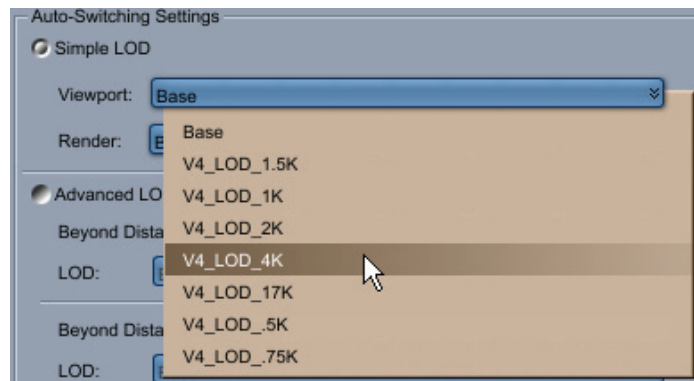
Example:

Viewport level set to use the 4K Victoria 4.2 mesh:
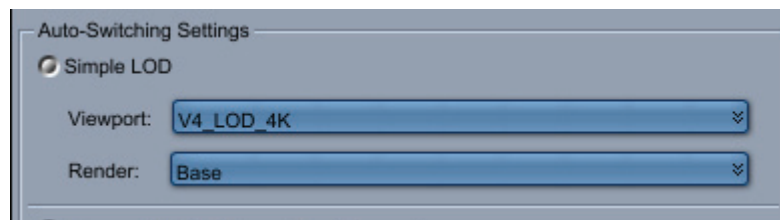
Victoria 4.2 4K mesh in the viewport:
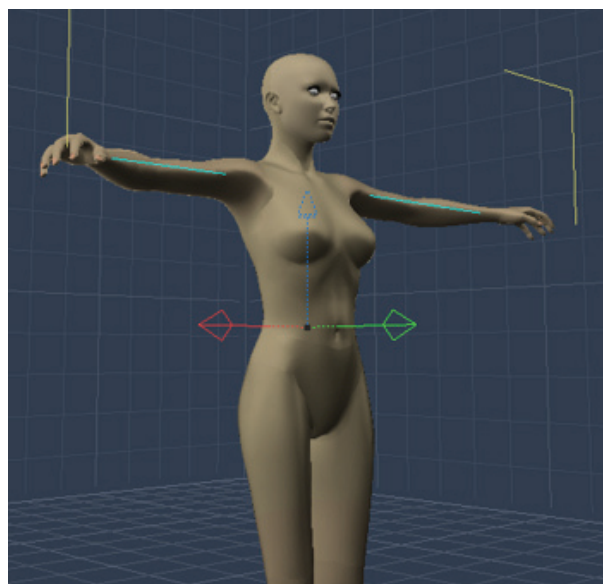
And the rendered version using the Base mesh:

8. **Advanced LOD (Distance From Camera)** - This option provides options to select which LOD level to use based on how far the figure is from the camera. This allows you to have several figures in the scene but to reduce the mesh resolution for figures that are further back in the scene and don't need as much detail available.

There are three slots available for setting distance from the camera and which LOD level to use beyond that distance.

To set the LOD levels and distance from camera:

1. Left-click on the button next to each LOD item to select an LOD level.
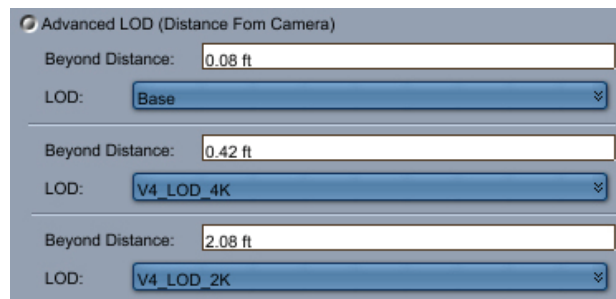
2. Left-click in each Beyond Distance field and enter the distance beyond which the figure will change to that specific LOD level. You can set the options from near to far as shown in the image above or far to near as shown below.
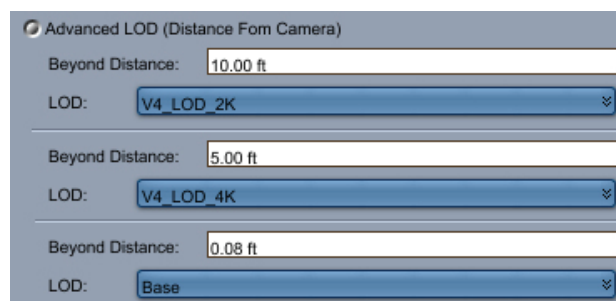
Examples:

- Base mesh before reaching 0.08 ft from camera:

- 4K mesh after passing 5 ft from camera:
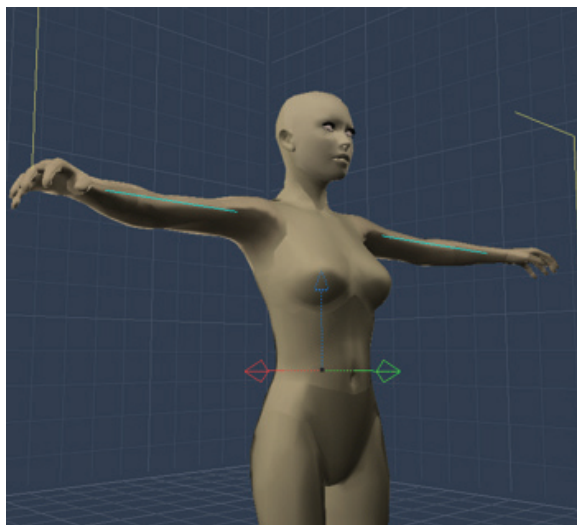
- 2K mesh after passing 10 ft from camera:
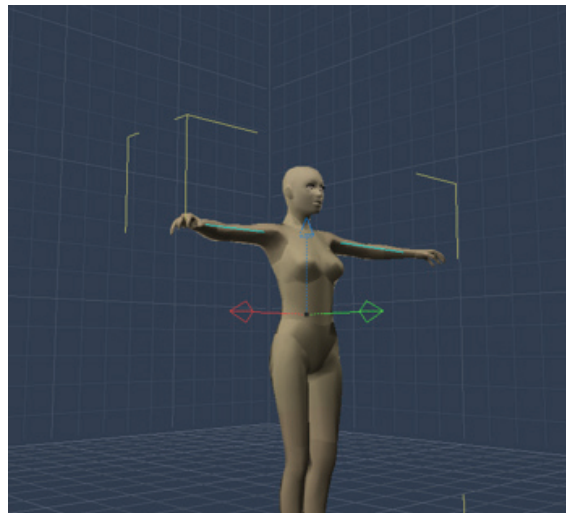
3. Click on **OK** to accept the changes or on Cancel to revert to the previous settings.

LOD List

The **LOD List** section provides options for deleting, renaming, and sorting the available LOD levels.

Click on an item in the list to select it. This will activate the buttons at the bottom of the dialog.

- To delete an item, click on the item then on the **Delete** button. In the image below, the 1.5K mesh was removed.

- To rename an item, click on the item then on the **Rename** button. The **LOD name** dialog will come up. Enter a new name and click on **OK** to rename or on **Cancel** to leave the name as is.

- To sort the list, click on an item then on the **Move Up** or **Move Down** buttons.
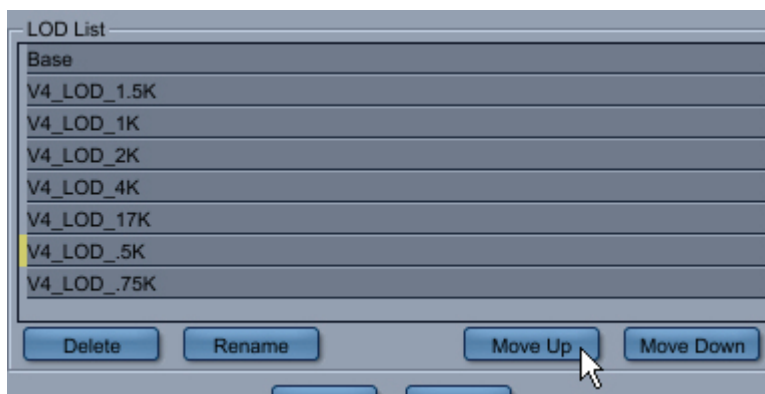
Repeat for each item to be moved until they are in the order needed. Once the items are sorted, they will be available in that order when selecting an LOD level in the Auto-Switching sections.
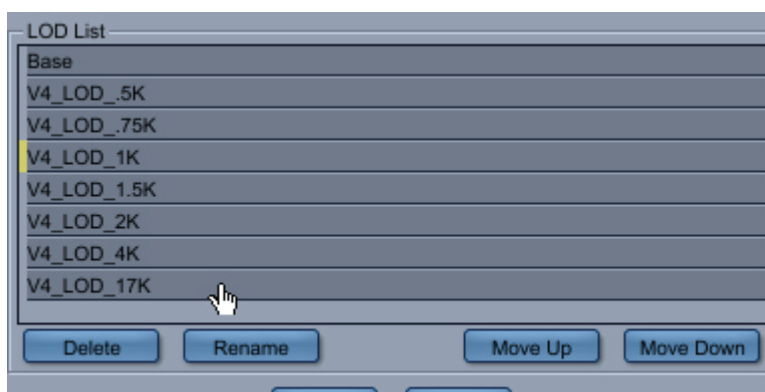
Using Environmental Primitives

Carrara offers several environmental primitives: terrains, trees, fountains, fire, fog, and clouds.

The Cloud, Fog, and Fire primitives are **volumetric**. With other objects, changing the size causes the object itself to change scale.

The terrains, the trees and the clouds are directly in the primitive toolbars. The Environmental Volumetric Primitive tools are located in a pop-up menu.

NOTE! Use the PhotoRealistic renderer to render the volumetric primitives.

Creating Terrains

The Terrain primitive simulates natural terrains. By adjusting the parameters, you can create such settings as rocky mountain ranges and desert dunes.
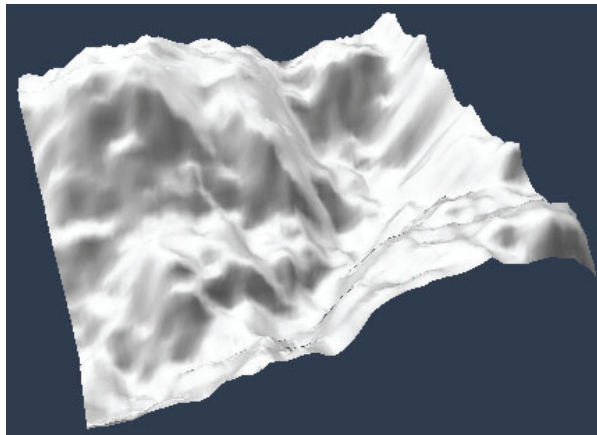
To insert a Terrain:

1. Depending on the desired location of the terrain, do one of the following:

    o   To create a terrain of default size at the center of the universe, choose **Insert >Terrain**.

    o   To create a terrain at any other location, drag the Terrain tool to the desired position in the scene window.

The Terrain editor appears.
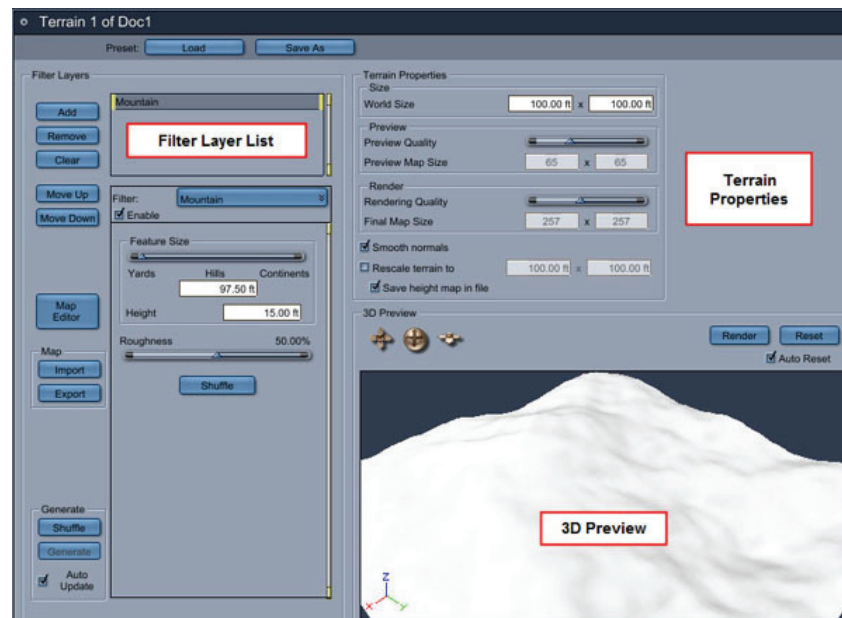
The terrain editor is divided into three parts:

- The **Filter Layers List** is the list of operations performed to generate a terrain.

- The **Terrain Properties** control the size and precision of your terrain.

- The **3D Preview** displays a preview of the terrain.

The Filter Layers List

The Filter Layers list represents the different filters that are used to generate a terrain. The top filter (or generator) is applied first, and then each of the filters is applied on the result of the previous filter. In the example shown in the following figure, the Canyon filter is applied on top of the Round Mountain filter (which means that it modifies the result of the Round Mountain filter). By combining various filters you can create any terrain you desire.

To create a new filter:

- Click on the **Add** button in the Filter Layers list to add a new filter at the end of the stack.
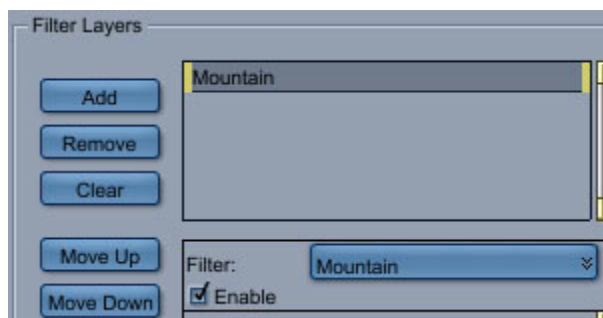
To create a terrain based on a height map:

- Click on **Import** (in the Map section) to import a texture to use as a height field.

To load a preset terrain:

- Click on **Load** (in the Preset section) to choose from a list of Wizard scenes that are furnished with Carrara and installed to your hard drive.
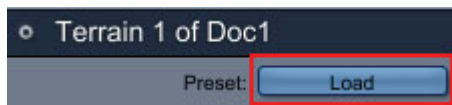
To edit a filter:

1. Select the filter by clicking on it in the list.

2. Select the type of the filter you want to use from the pop-up menu.

3. Edit the properties of the filter.
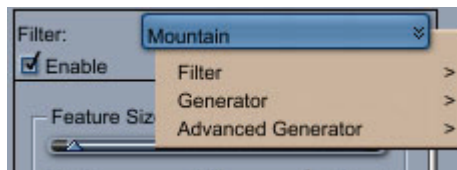
For a more detailed description of each filter, see "Filter Reference."

The Terrain Properties

The Terrain Properties control the size and precision of your terrain.
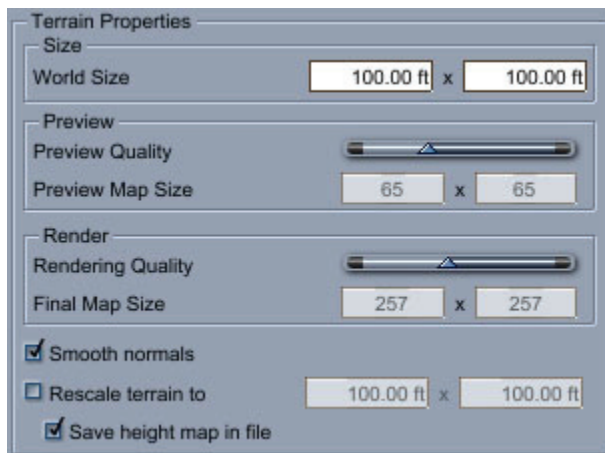
The properties are:

- **World Size:** The reference size of your terrain for the filters. This is not the size of the terrain in your scene.

- **Preview Quality**: Adjust the slider to view less or more detail in the 3D Preview.

- **Preview Map Size**: Enter the desired dimensions for the preview map. Higher values provide more detail, and lower values use less resources.

- **Rendering Quality**: Adjust the slider to view less or more detail in the rendered image in the 3D preview window.

- **Final Map Size**: Enter the desired dimensions for the map used for the final render. Higher values provide more detail, and lower values use less resources.

- **Rescale terrain to**: To change the size of the scene, check this option and enter the desired dimensions in the size fields.

- **Save height map in file:** Ensures that the terrain is stored in the file. This avoids recalculating the terrain every time you load the file. However, be careful, as this option can generate large files.

The 3D Preview (Landscapes)

The 3D preview allows you to view the terrain as it is edited. You can move and rotate the point of view using the camera tools. The Render button will raytrace the preview to create a more realistic preview of your terrain.

The preview is usually updated each time your modify your terrain. However, for complex filters, calculating the preview after each change can be too slow. In that case, uncheck Auto-Update and use the **Generate** button to update the preview.
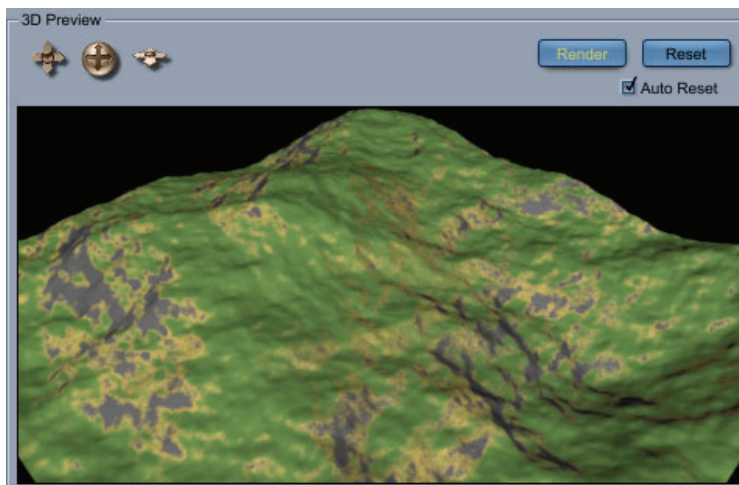
Creating a Custom Terrain

Sometimes it is more convenient to edit the terrain directly. For example, you may want to make a mountain a little lower or place a valley in the middle of your terrain.

To do so, click on the **Map Editor** button. It converts the current filter list to a static height field and opens a new editor. Note that once you have edited a terrain manually, you cannot modify the filter parameters further. You can however apply new filters on top of your terrain.

The Drawing area in the Map Editor provides a top view where you can create and edit a basic terrain shape using the Terrain tools and the Modify controls.
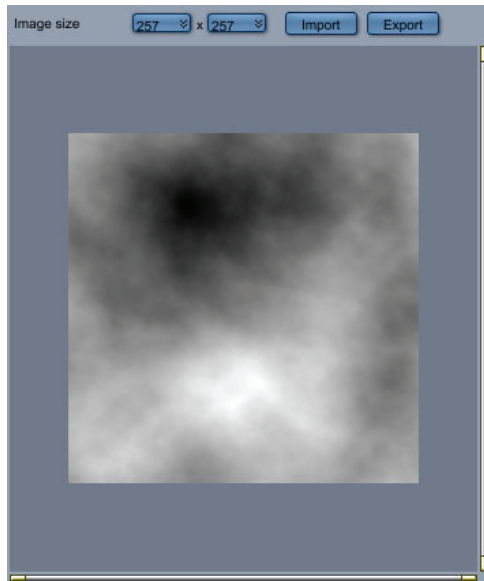
You select Terrain tools and drag in the Drawing area to directly edit the basic terrain shape. From left to right, the icons represent the **Elevation Paint**, **Raise**, **Crater**, and **Erode** tools. While you paint the terrain, note that lighter areas represent high altitudes, while darker areas represent lower altitudes.
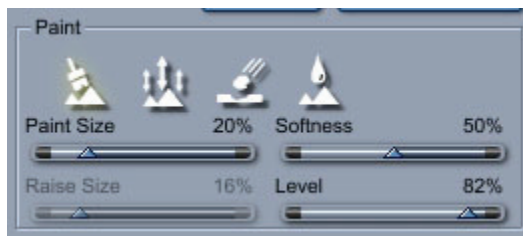
Once in the Terrain Editor, there are many different ways that you can modify your terrain.

Painting a Terrain

The Elevation Paint tool acts like a paintbrush. As you drag in the Drawing area, Carrara creates mountain ranges based on the shape you are creating and displays them in the Preview area.

When the Elevation Paint tool is active, you can adjust the brush using the following controls:

- **Paint Size:** Drag right or left to increase or decrease brush size.

- **Softness:** Drag left or right to increase or decrease brush softness.

- **Level:** Drag the slider to adjust the brightness of the paint color.

To create or add to a terrain:

- Select the Elevation Paint tool and drag in the Drawing area.

Increasing and Decreasing Heights

The Raise tool increases and decreases the height of a local area as you brush over it.

When the Raise tool is active, you can adjust the area that will be affected with the Raise Size slider.

To increase or decrease the height of an area:

1. Select the Raise tool and left-click in the Drawing area where you want to modify the terrain.

2. While left-clicking, drag up or down to increase or decrease the height.

Adding Craters

Use the Crater tool to create indentations in the terrain.

To create a crater:

1. Select the Crater tool.

2. Drag over the area you want to indent.

Simulating Erosion

Use the Erosion tool to simulate the effect of water erosion. Carrara erosion leaves sharper peaks and deeper valleys.

To erode an area:

1. Select the Erosion tool.

2. Drag over the areas you want to erode.

Filter Reference

The terrain editor provides a wide range of powerful filters to create a wide variety of terrains.
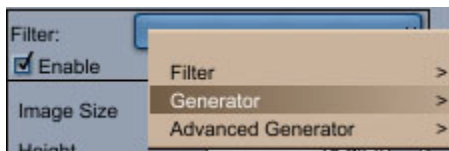
There are three categories of filters in the Terrain Generator.

- **Generators** are generally used at the top of the filter list. They can create a landscape without being applied to another filter. Note that a generator is still a filter, meaning that it will be applied on top of the previous filter in the list (if there is one).

- **Advanced generators** provide additional options such as displacement, fractals, gradients, and more.

- **Filters** are generally used after a generator or another filter. They are used to modify an existing terrain.

Generators

The main generators that you can select from the Generator menu are:

- The **Crater** filter generates a flat terrain with craters. Use it to create moon-like landscapes. Because it generates a flat terrain, you may want to combine it with another generator to make a more realistic terrain.

- The **Hill** filter creates a gentle rolling hill on the terrain.

- The **Mesa** filter creates a desert with some mesas.

- The **Mountain** filter creates mountains that have the same rocky aspect everywhere. To increase the realism of this terrain, use other filters such as the glaciate filter or the canyon filter.

- The **Round Mountains** filter creates terrains with some mountains and some flat areas.

- The **Smooth Mount** filter creates a terrain with smooth raised and lowered areas.

- The **Smooth Plain** filter creates a terrain with smooth hills.

- The **Straight Mountain** filter creates straight mountains.

- The **Valley And Mountain** filter creates very realistic landscape that includes rocky mountains and smooth valleys.

- The **Valley and Straight Mountain** filter creates straight rocky mountains and smooth valleys.

After you select a terrain generator, its properties appear in the terrain editor. The following figure shows an example of the properties that are available for the Hill generator.
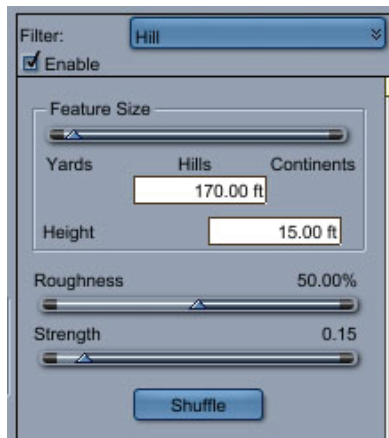
The following settings are common to many of the generators:

- **Feature Size**: Controls the scale of the feature over the existing terrain. Decrease the feature size to create a larger number of hills or mountains on the terrain. Increase the feature size to create a fewer number of hills or mountains. As you adjust the slider, a numeric indicator displays the relative size of the feature.

- **Height**: Enter the desired maximum height for your feature.

- **Roughness**: Adjust the slider to affect the smoothness or roughness of the terrain.

- **Strength**: Adjust the slider to increase the amount of strength that is applied by the terrain map.

Filters

The Filter menus in the Terrain editor also provide a variety of filters that create the following modifications:
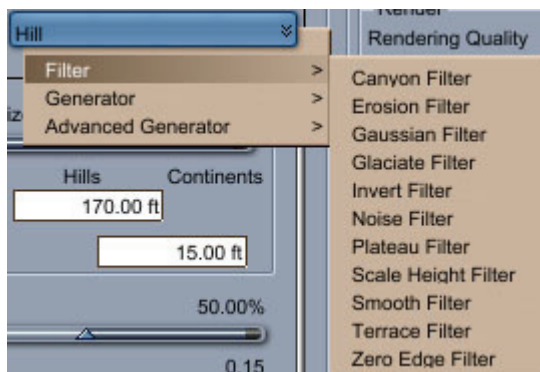
- The **Canyon** filter creates a canyon in your landscape. You will have some flat areas at the bottom and at the top and between cliffs.

- The **Erosion Filter** can create a realistic terrain by applying erosion to the terrain. You can apply four types of erosion: rain, thermal weathering, wind, and smooth erosion. The most important parameter is the Erosion Cycle; it controls how many cycles of erosion will be applied. Each type of erosion that you enable is applied at each cycle. Note that a high value of Erosion Cycle will lead to a very slow terrain generation.

- The **Gaussian Filter** creates a terrain with a border that has a height close to zero. Use it to create a single mountain (See also the Zero Edge Filter).

- The **Glaciate Filter** creates flat valleys similar to the valleys that result from the erosion by a glacier.

- The **Invert Filter** inverts the map of the terrain. You can use this filter, for example, to create all kinds of rivers by using the level parameter.

- The **Noise Filter** adds visual noise to your terrain. The location of the noise can be controlled by the local shape of the landscape (altitude, slope, and orientation). Use visual noise to create an area that looks like a forest on your terrain.

- The **Plateau Filter** adds a midland plateau to your landscape.

- The **Scale Height Filter** changes the height of a terrain.

- The **Smooth Filter** smoothes the terrain.

- The **Terrace Filter** creates terraces on the landscape.

- The **Zero Edge Filter** modifies the terrain so that all borders have a height of zero. It can be used to create a single mountain.

Advanced Generators

Finally, the Filter menu provides advanced generators that allow you to further customize the appearance of your terrains.
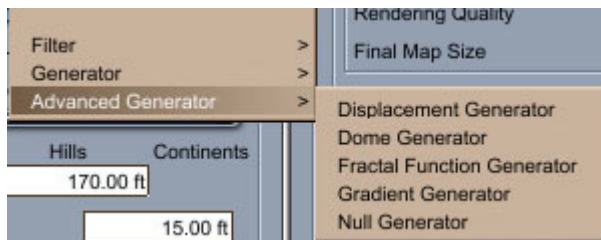
- The **Displacement** filter is very similar to the Mountain generator. Its main advantage is that it is very fast.

- The **Dome** filter creates a dome centered on your terrain.

- The **Fractal Function** filter can be used to make most types of terrain. It provides you control of all parameters. This is a complex generator with more options and can be challenging to use. The standard generators will meet most of your needs.

- The **Gradient Generator** creates a gradient of heights.

Terrain Shading

You can use any kind of shader to shade a terrain. The Terrain shader is specifically designed for this purpose and makes it easier to shade a terrain.

When you switch to the Texture room from the Terrain Editor, or when a terrain is selected in your scene, you see the shader properties that are currently assigned to the terrain.

About Terrain Layers

Terrain layers allow you to build up different types of ground cover on your terrain. The terrain shader is composed of a number of different layers. Each layer in the list corresponds to a type of surface. The

layers are applied on top of each other (for example, the snow layer covers the grass layer, which covers the land layer, and so on). The last layer covers the global shader.

When you expand a Terrain Layer in the Shading tab, you will see a **distribution** and a **shader** surface associated with it.

When you create your terrain layers, you should consider the order in which they appear in the Shading hierarchy. In other words, snow usually appears at the top of a mountain, with rocks, land, and grass below. Wherever none of the other layers are visible, you will see the global shader

Distribution Types

The distribution type determines the type of covering that is distributed on the selected terrain layer. There are different types of distribution, described in the following sections. Choose None if you do not want to apply a distribution layer.

There are eight different types of distribution layers:

- **None**: No distribution layer.

- **Custom**: The Custom distribution type provides access to all the parameters that control a distribution. This allows you to create your own customized distribution layers.

- **Everywhere**: The Everywhere distribution type puts the distribution layer everywhere. This means that no layer will be visible under it. There are no properties to set.

- **Grass:** Grass distribution layers are used to distribute a grass or tree layer. The layer will be visible only between two altitude values and on relatively flat areas (grass does not grow on vertical cliffs).

- **Land**: Land distribution layers are used to put a layer of land on the terrain. It is located only in areas that are flat enough (minimum slope).

- **Rock**: Rock distribution layers are usually located in areas where the slope is steep. It is useful to position rocks on your terrain.

- **Simple:** The Simple distribution layer distributes the layer properties randomly.

- **Snow**: The Snow distribution layer distributes the layer only at the top of the mountain and in areas where the slope is not too steep.

When you choose one of the distribution types, its properties appear in the right pane of the terrain shader editor. The properties you can set are arranged in five categories. Though these categories do not apply to all of the distribution layer types, the purpose of the settings are common to all layer types:

Bump Properties

When you choose the Custom, Grass, Land, or Rock distribution layer type, you'll find the following typical settings that adjust various bump properties:
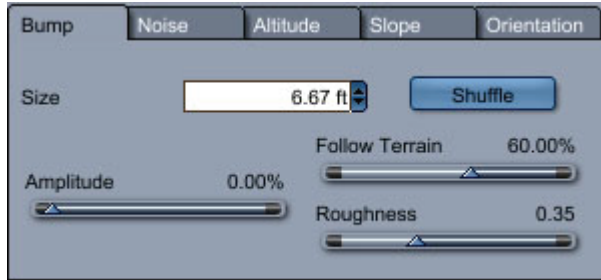
- **Size** (Custom only): Determines the size of the noise features you can see in the bump.

- **Amplitude** (all): Determines the amplitude of the bump.

- **Follow Terrain** (Custom and Rock): Adjusts the bump altitude depending on the steepness of the terrain. Where the terrain is steep, the bump has a bigger amplitude. This parameter controls how strong the variation should be.

- **Roughness** (Custom and Rock): Determines the roughness of the noise used in the bump. A rough surface has a lot of variation. Set this value lower for smoother surfaces.

- **Shuffle** (Custom only): Click this button to randomly generate bump for the terrain.

Noise Properties

When you choose the Custom, Grass, Land, Rock, or Simple distribution layer type, you'll find the following typical settings that adjust various noise properties:
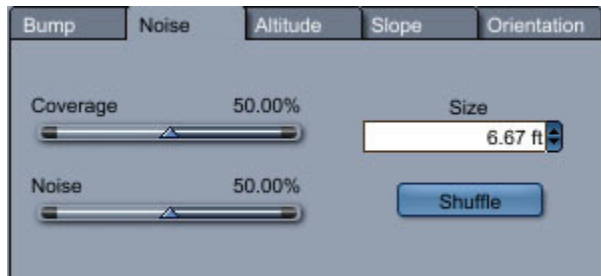
- **Coverage** (all): Determines how much this layer covers the terrain. At 0% the layer is not visible on the terrain. At 100% it completely covers the terrain.

- **Noise** (all): Determines the amount of noise in this layer. The variation between the area covered and the area not covered can be more or less noisy, based on the setting of this parameter.

- **Size** (all): Determines the size of the noise features.

- **Shuffle** (all): Click this button to randomly generate noise for the terrain.

Altitude Properties

When you choose the Custom, Grass, or Snow distribution layer type, you'll find the following typical settings that adjust various altitude properties:
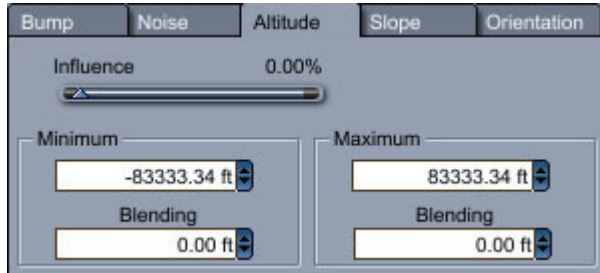
- **Influence** (Custom and Grass): Allows you to control where the layer should be visible, depending on the altitude. For example, you might want to put a layer of snow above a certain altitude.

- **Minimum** (all): This layer will not cover any part of the terrain when the altitude is below this value.

- **Maximum** (Custom and Grass): This layer will not cover any part of the terrain when the altitude is above this value.

- **Blending** (all): .

Slope Properties

When you choose the Custom, Grass, Land, Rock, or Snow distribution layer type, you'll find the following typical settings that adjust various slope properties:
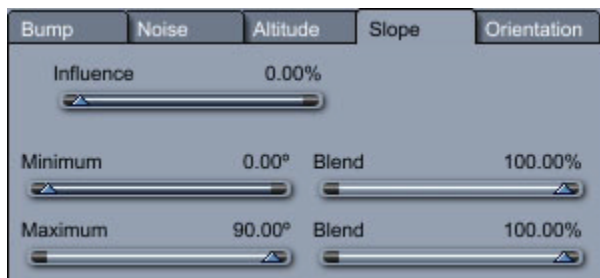
- **Influence** (Custom only): Allows control of the layer repartition depending on the slope. Using a snow layer as an example, you might not want to put snow on steep parts of the terrain. This parameter controls how strongly the slope limits need to be taken into account.

- **Minimum** (Custom and Snow): This layer will not cover any part of the terrain when the slope is below this value.

- **Maximum** (Custom, Grass, Land, and Snow): This layer will not cover any part of the terrain when the slope is above this value.

- **Blend** (all): This parameter controls the fuzziness of the transitions.

Orientation Properties

When you choose the Custom or Snow distribution layer type, you'll find the following typical settings that adjust various orientation properties:

- **Influence** (all): Allows control of the layer repartition depending on its orientation. Again using snow as an example, you might not want to put snow on the part of the terrain that is oriented toward the sun. This parameter controls how strongly the orientation limits need to be taken into account.

- **Direction** (all): Defines a vector in 3D space that is used as a reference orientation. The layer will be visible only if the surfaces face this orientation.

- **Pitch** (all): Defines a vector in 3D space that is used as a reference orientation. The layer will be visible only if the surfaces match this pitch setting.

- **Blend** (all): This parameter controls the fuzziness of the transitions.

Creating Plants

The Plant primitive simulates real plants with branches and leaves, such as trees. Even if you only adjust a few parameters you can create many different types of trees. The number of parameters might seem a little bit daunting at first. However a lot of the parameters are the same for each generation.

That is why we recommend that you always start from one of the preset trees and use this dialog as an exploration tool. It is usually a good idea not to modify too many parameters at the same time since the results can be difficult to understand.

It is important to understand the concept of **generation.** The branches of the tree are hierarchical with a concept of generation: the trunk is generation 0, branches born from the trunk are generation 1, and so on. Some parameters depend on the generation of the branch they apply to, so that branches at different levels in the tree hierarchy can have different properties.

To insert a Plant:

1. Depending on the desired location of the tree, do one of the following:
   - To create a tree of default size at the center of the universe, choose Insert **>Plant**.
   - To create a plant at any other location, drag the Plant tool to the desired position in the scene window.
2. Carrara will move to the model room where the Plant editor opens.

Basic Plant Editing

While editing your plant, you can at any time click on the **Render** button to see a raytraced rendering of your tree using its current shader.

The Plant editor contains five tabs, each of which contains different parameter settings for your plants. These tabs are **Trunk**, **Branches**, **Leaf**, **Tree Shape**, and **Experts**.

Additionally some parameters are provided to control the appearance of the tree in the manipulation window.

The Plant editor allows you to create your own plant types. Creating plants can be a challenging task so it is recommended to start from one of the presets delivered with the application, and to read the following documentation to understand the meaning of each parameter.
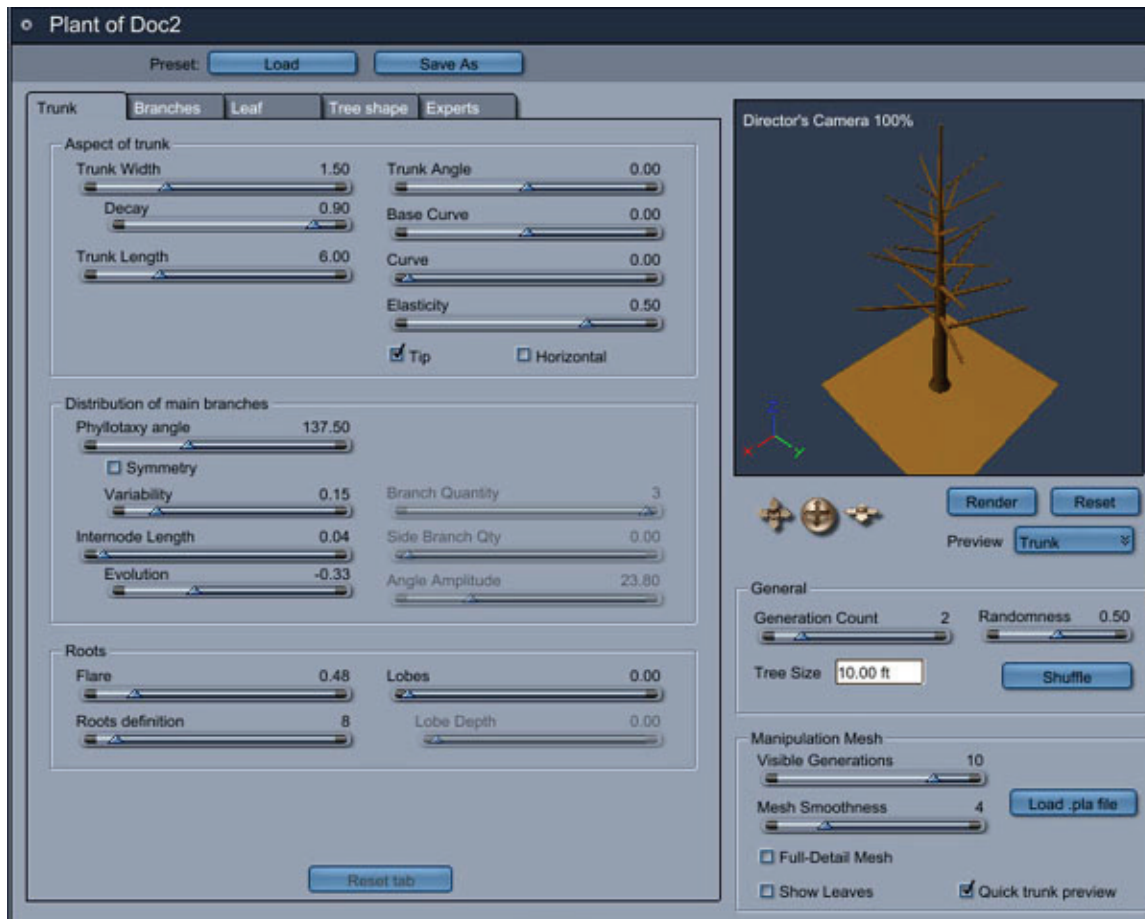
Further information about the options in the Plant Editor are discussed in the following sections:

General Properties

The general properties appear directly beneath the tree preview render area. The settings available in this section control the general appearance of the tree.
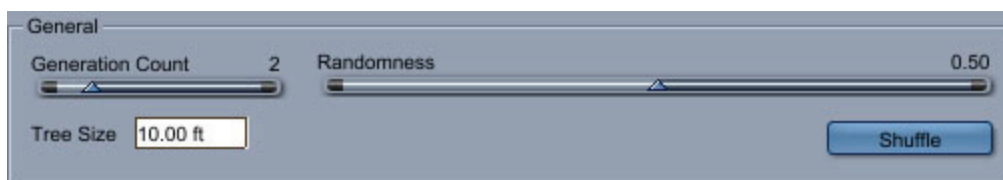
- **Generation Count**: This parameter controls the maximum number of generations used for a tree. The idea is that each branch has child branches and each series of child branch constitutes a generation. Setting this parameter to 0 will create a simple trunk.

TIP! Keep the number of generation as low as possible while modeling to speed up the preview redraw. Be careful when increasing this number as it can require a lot of time and memory to generate a very complex tree.

- **Randomness**: This parameter controls the distribution of the branches and leaves in the tree. A value of zero will lead to an even distribution whereas a bigger value will create a more random tree.

- **Tree Size**: Controls the size of the tree.

- **Shuffle**: Clicking on this button generates a different tree using the same parameters.

Manipulation Mesh Settings

The manipulation mesh is the mesh you see in the interactive preview (3DView in the assemble room). This mesh is also the mesh that will be used if you apply a modifier to a tree, if you export your tree or if you convert the tree to another modeler. It is recommended to use a low resolution mesh to get good performance when you edit a scene containing trees. (The full resolution mesh will always be used for final rendering).
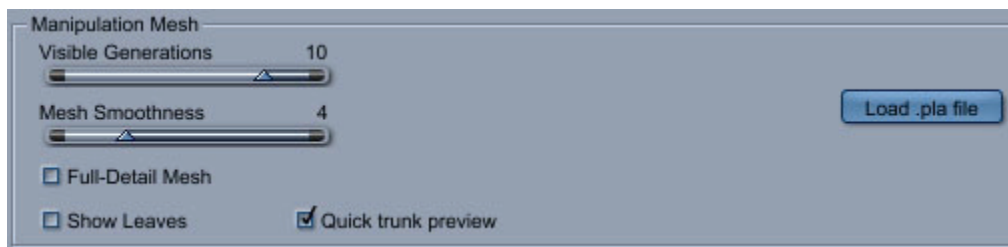
- **Visible Generations**: The maximum number of generations that are used for the manipulation mesh.

- **Mesh Smoothness**: The number of triangles used for the mesh. (a smoother mesh requires more triangles)

- **Full-Detail Mesh**: Check this if want to see the most detailed mesh in the assemble room (this is useful if you want to export a tree to another program and want to apply a modifier to a tree). Keep in mind that the complete mesh is much more complex than the default one and will require a lot of memory and will redraw a lot slower.

- **Show Leaves**: Check this if you want to see the tree leaves in Assemble Room. (leaves will be symbolized by triangles).

- **Quick Trunk Preview**:

- **Load .pla file**: A Tree Type is a particular set of parameters which defines a specific plant species. Tree types are stored as presets files on your hard disk (*.pla files). You can browse the installed tree types by opening the popup-menu, or save your own tree types by clicking the Save button - the new preset will then appear in the menu.

Trunk Properties

You can control the size and complexity of the trunk using the following parameters:

Aspect of Trunk
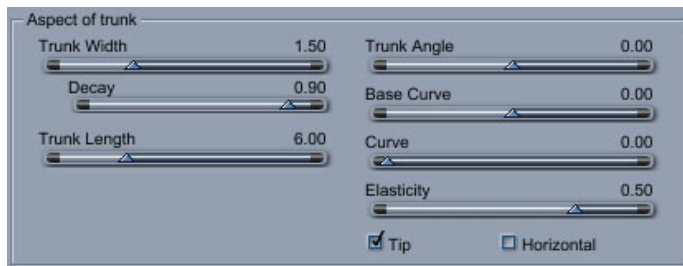
- **Trunk Width**: The width of the trunk. Note that the width of child branches is defined relative to this width.

- **Trunk Angle**: The angle between the trunk and the vertical direction. (Can be negative to lean the tree backwards).

- **Decay**: Controls how fast the size of the trunk decreases with the height.

- **Base Curve**: The curvature of the trunk. A value of 0 will produce a straight trunk, a positive value will bend it forwards, a negative value will bend it backwards.

- **Trunk Length**: The length of the trunk. Note that the lengths of child branches are defined relative to this length.

- **Curve**: Randomly curves the branches in a direction orthogonal to growth axis. A curve of 0 produces straight branches, and a curve of 1 produces twisted branches.

- **Elasticity**: Controls how a branch is sensible to tropisms - that is, gravity or light. When Elasticity is zero, the branch will not be changed by the Attraction parameter, while when it is 1, the effect of Attraction is maximized.

- **Tip**: Allows the generation of a child at the end of the mother branch, in a continuing direction. Most plant species do have this extra child at the end of every parent branch. (used by the centered generator).

- **Horizontal**: Forces the child branches to grow with an horizontal direction while still complying with the Start Angle constraint (centered and hybrid generators).

Distribution of Main Branches

- **Phyllotaxy Angle**: Controls the rotation of successive child branches around and along a same mother branch. For ramified generators it sets the angle between child branches, and for the centered generator it controls the way child branches spiral around the mother-branch. A good value for this angle is 137.5 degrees.

- **Symmetry**: Forces the generation of symmetric child branches in the centered generators.

- **Variability**: Controls how much the angle varies for this generation.

- **Branch Quantity**: Controls the maximum number of branches per junction (only used for hybrid or ramified generators).

- **Internode Length**: Represents the distance between two child branches.

- **Side Branch Qty**: Controls the quantity of side branches (by opposition to a junction in a ramified generator).

- **Evolution**: Increase/decrease of internode length (distance between two consecutive child branches on a mother branch).

- **Angle Amplitude**: Controls the amplitude of angles that can occur along the branches of a ramified or hybrid tree (see Angle Rate).

Roots
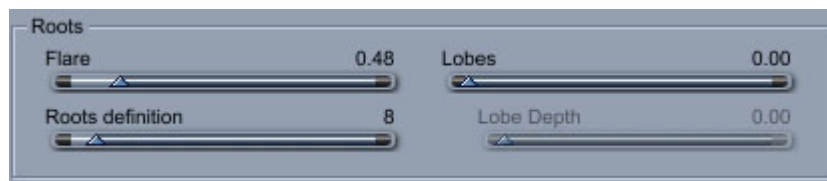
- **Flare**: Controls the expansion of the trunk at the bottom. A flare of 0 will prevent any expansion

- **Lobes**: Lobes is the number of roots in the plant.

- **Roots Definition**: Represents the number of polygons used to approximate the roots.

- **Lobe Depth**: Controls the depth of the roots.

Branch Properties

The Branches tab controls the properties of the branches on your plant. The settings are as follows:
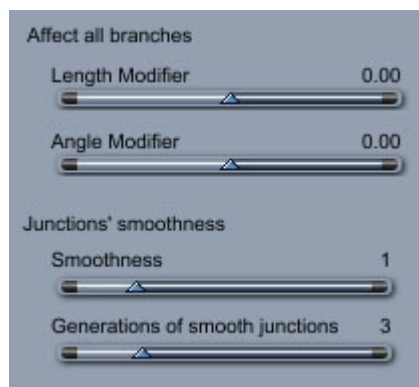
Affect all branches

- **Length Modifier**: This parameter controls the length of all branches on the tree. A positive value will yield longer branches, whereas a negative value will shorten all branches.

- **Angle Modifier**: This parameter controls the angles between all branches in the tree. A positive value will increase all angles while a negative value will decrease all angles.

Junctions' smoothness

- **Smoothness**: The number of points used to approximate a junction.

- **Generations of smooth junctions**: The number of junctions that fully approximate with polygons.

Leaf Properties

You can control the foliage of your tree by using the following parameters:
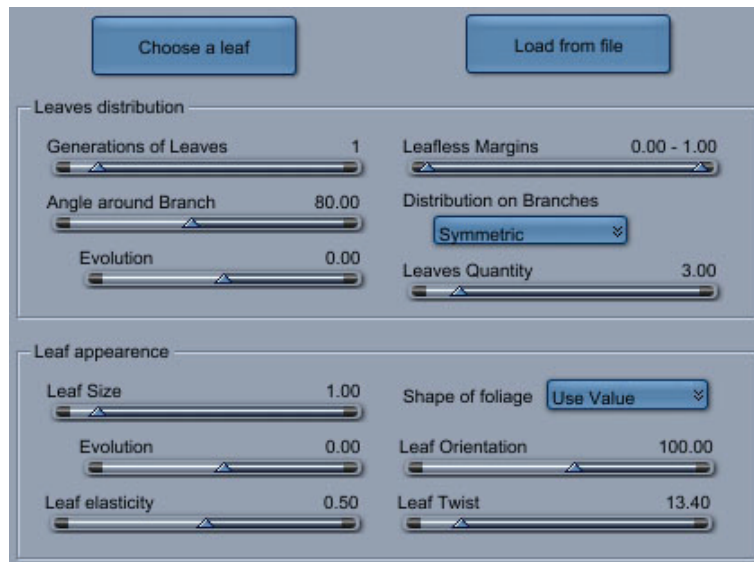
- **Choose a Leaf**: Click this button to open the Scene Wizard, which allows you to select one of many different types of leaves. After you select your leave, click OK to load the leaf shader (if desired) and return to the Plant Editor.

- **Load from File**: Click this button to load a leaf file from your hard disk. A leaf is defined by a Carrara file in which the first object is assumed to be the leaf. In this file the x-axis is the length axis for the leaf, the y-axis its width axis, and the z-axis points out of the upper face of the leaf. The plug-in will automatically compute the base point of the leaf from its shape and orientation. The popup-menu allows you to browse through the available leaf models (*.car files). You can also create your own leaf models, they will automatically appear in the menu if you place the files in the appropriate folder (in your Carrara folder, look for the Data folder, then for a Plants subfolder).

Leaves Distribution

Settings in this section control the distribution of leaves on the tree.

- **Generations of Leaves**: The number of generations of branches bearing leaves (usually one)

- **Leafless Margins**: Determines the length at the beginning and end of a branch that remains without leaf expressed as percentages of the branch length. Since every branch grows, as a default behavior, a leaf at their end, you can use a non-null End Margin parameter to prevent branches from having a leaf at their ends.

- **Angle around Branch**: Controls the increase/decrease of Leaf Angle (angle between leaf and bearing branch) along a branch.

- **Evolution**: Controls the increase/decrease of the leaf size along a branch.

- **Leaves Quantity**: The average number of leaves along a branch segment.

Leaf Appearance

- **Leaf Size**: The size of the leaves.

- **Shape of foliage**: Controls the shape of the foliage in changing the size of leaves along the branch - exactly the way the tree shape parameter changes the lengths of the branches along the trunk. A special mode, named "Use Value", selects no particular shape but rather uses the Size Evolution parameter to compute leaf sizes.

- **Evolution**: Determines how the leaf size changes compared to the previous generation of branches.

- **Leaf Orientation**: Controls the angle between the leaf and the vertical (an orientation of 0 makes the leaf point up, an orientation of 180 makes it point down).

- **Leaf elasticity**: Controls the distribution of the leaves.

- **Leaf twist**: The amplitude of random perturbations to leaf angle and leaf orientation, so the leaves on a branch do not share exactly the same orientation.

- **Leaf Angle**: The angle between the branch and the main axis of the leaf. A low angle will make leaves point upwards, a big angle will make them bend downwards relatively to the branch they grow on.

Tree Shape Parameters

You can control the global shape of your tree using the following parameters.
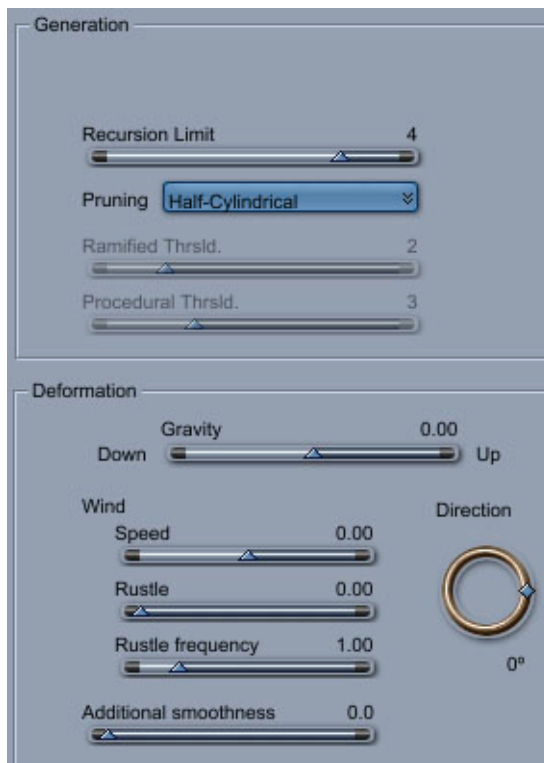
Generation

- **Recursion Limit**: Limits the number of times the procedural generator (centered generator or hybrid generator) is called recursively. This avoids generating by error hugely complex trees.

- **Pruning**: Determines what shape the tree is constrained to.

- **Ramified Threshold**: The generation number until which the tree will grow in a ramified way.

- **Procedural Threshold**: The generation number from which a segment will spawn children procedurally (that is, spiraling all along the segment). Note that if Procedural Threshold is superior to Ramified Threshold, no child branch will be spawned using the procedural process.

Deformation

- **Gravity**: The strength of the gravity that is applied to the branches.

- **Wind**: Set the Speed, Rustle, and Rustle Frequency settings as desired to control how the leaves react to wind forces in your scene. Use the Direction slider to control the direction that the wind comes from.

- **Additional Smoothness**: Controls the smoothness of the animation.

Experts Plant Parameters

Aspect of Branch

- **Width Ratio**: Controls the ratio of the width of a branch, relatively to its parent branch. Usually these ratios should remain under 1 since branches grow shorter and thinner generation after each generation.

- **Variability**: Variability parameters control the randomness of the tree. They control how much the randomized values can vary from the statistical average of these kind of characteristic (lengths, widths, angles).

- **Evolution**: Increase/decrease of width ratio between child branch and mother branch.

- **Decay**: Controls how fast the size of the tree branches decrease with the length.

- **Length Ratio**: Controls the ratio of the length of a branch, relatively to its parent branch. Usually these ratios should remain under 1 since branches grow shorter and thinner generation after each generation.

- **Variability**: Controls how much the angle varies around the value entered.

- **Evolution**: Increase/decrease of length ratio between child branch and mother branch.

- **Start Angle**: Control the angle between a child branch and its mother branch. The differences between Start Angle and End Angle determines the curvature of the branch parallel to the growth direction.

- **Evolution**: Increase/decrease of the angle under which the child branches are grown along the mother branch.

- **End Angle**: Control the angle between a child branch and its mother branch. The differences between Start Angle and End Angle determines the curvature of the branch parallel to the growth direction.

- **Evolution**: Increase/decrease of the angle between the end of the child branch and the mother branch.

- **Curve**: Controls the amount of curve in this generation of branch.

- **Elasticity**: Controls the elasticity of the branch.

- **Tip**: Check to add a tip to the branch.

- **Horizontal**: Changes the orientation of the branches.

Distribution of Subbranches

- **Phyllotaxy angle**: Controls the rotation of successive child branches around and along a same mother branch. For ramified generators it sets the angle between child branches, and for the centered generator it controls the way child branches spiral around the mother-branch. A good value for this angle is 137.5 degrees.

- **Symmetry**: Forces the generation of symmetric child branches in the centered generators.

- **Variability**:

- **Internode Length**: Represents the distance between two child branches.

- **Evolution**: Increase/decrease of internode length (distance between two consecutive child branches on a mother branch).

- **Margin Length**: The size of the base of the branch where no child will be spawned.

- **Branch Quantity**: The average number of branches generated. at the end of each branch.

- **Side Branch Quantity**: Represents the probability that a branch will give birth to a side branch.
- **Angle Rate**: Represents the probability that a branch will show an "angle" - that is, a discontinuity in direction. The value of the angle is controlled by the Angle Amplitude parameter.

Creating Clouds

Carrara 6 provides several ways to add clouds to your scenes. This section takes a look at the several options available for creating many different types of clouds.

Note that you can also add cloud layers in the Sky Editor. To learn more about the sky editor see "Sky."

Further information about Carrara's cloud primitive tools are discussed in the following sections.

Volumetric Clouds (Standard, Pro)

The volumetric cloud primitive allows you to create and place 3D clouds in your scene. Those clouds scatter light realistically. When used in conjunction with the realistic sky they can give very realistic clouds.

- Choose **Insert > Volumetric Clouds** or click and drag the Volumetric Clouds tool into the scene.

Once you insert a cloud in your scene, the cloud editor opens in the modeling room. The editor is split in several sections: the Light to Use panel, the Shape panel, the Properties panel and the 3D Preview.

The following sections discuss the settings for Volumetric Clouds.

Light to Use

The **Light to Use** panel allows you to control which light will affect the cloud.
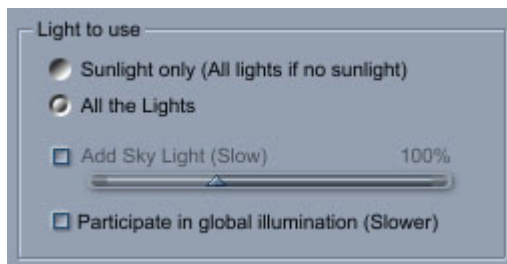
You can use **only the sun light** (which is recommended if you are doing outdoor scenes) or **all the lights** in your scenes.

You can also **add the sky light** to get more realistic lighting effects. This is especially useful if you are creating sunsets. Note the rendering will be slower with this option.

The Shape Panel

The shape panel allows you to edit the shape of the cloud. Several types of clouds are provided.
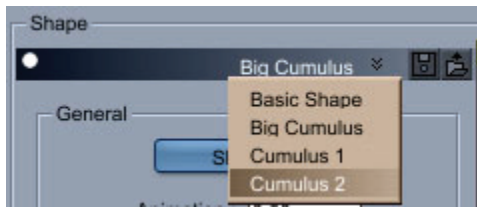
Several cloud shapes are available to choose from.

Basic Shape

Basic Shape fills simple shapes (cube, sphere, etc.). This does not create realistic clouds.
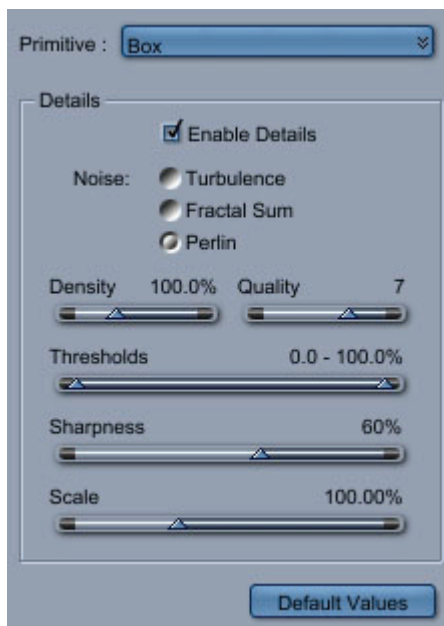
Big Cumulus

Big Cumulus creates large clouds.
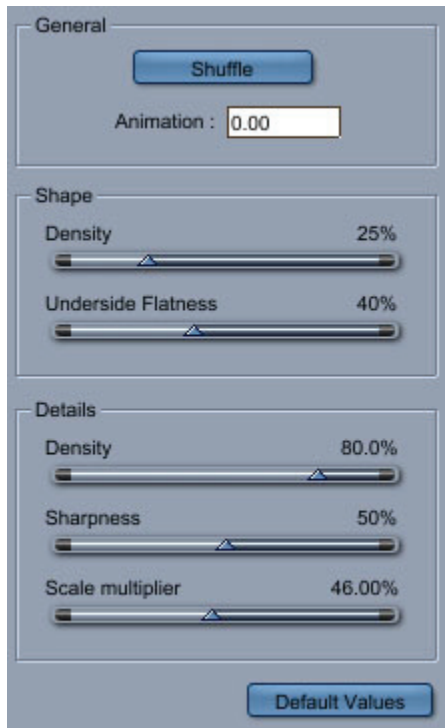
Cumulus 1 and 2 Clouds

Cumulus 1 creates generic clouds. Cumulus 2 generates shapes that are less complex than cumulus 1. The settings are similar for both types:
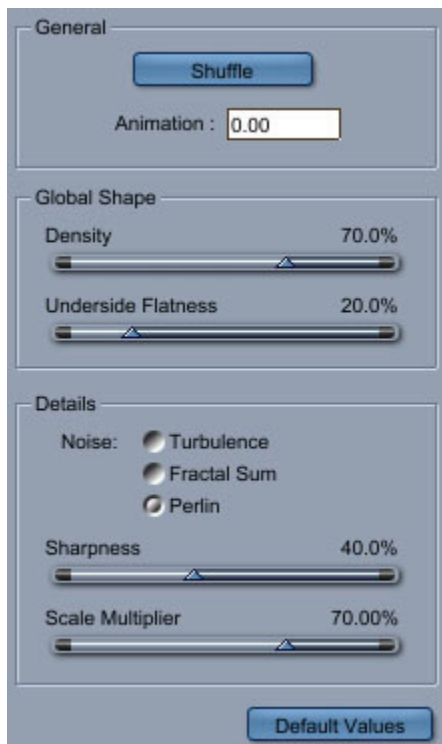
General Properties Panel

This panel allows you to edit the size of the clouds as well as some basic lighting properties.

A couple of remarks:

- The **silver lining** controls how the clouds receives light depending on the direction. A high value will create a more directional lighting which leads the silver lining effect when the cloud is lit from behind.

- The size of the cloud is very important. To obtain realistic effects, the size of the atmosphere and the clouds have to be realistic in relation to your scene.
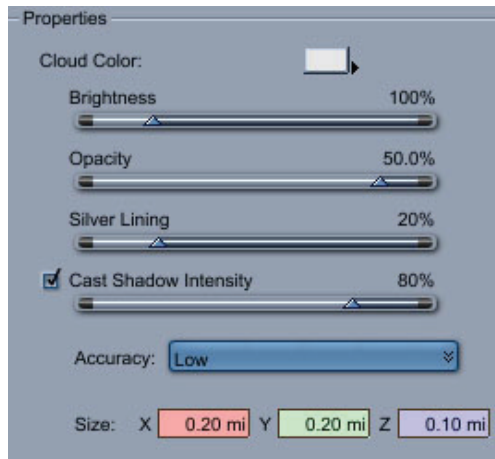
The 3D Preview (Clouds)

The 3D preview allows you to visualize your clouds from any angle you want (use the camera tools to change the angle of view).

One useful option is that you can either render the preview using a Draft Mode or Full Accuracy. It is usually a good idea to create the shape of your cloud in Draft Mode and then use Full Accuracy to tweak the lighting properties.
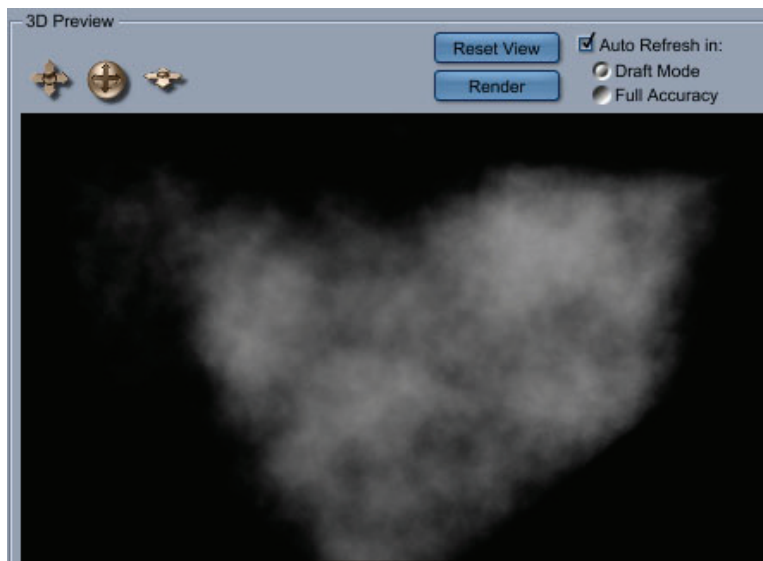
Volume Clouds

The Cloud primitive creates a 3D volume filled with clouds. You can use this primitive to quickly add a sky to any scene.

There is no on-screen preview of the cloud except its bounding box, which indicates the cloud's size and location. You cannot see the cloud until it's rendered.

Tip! You can do a spot check of the cloud using the Test Render tool. Select the tool and drag over any area of the cloud.

You can place objects inside the cloud or partially within the cloud. Cloud attributes can also be animated.

To create a cloud object:

1. Depending on the desired location of the cloud, do one of the following:

    *   To create a cloud of default size at the center of the universe, choose Insert **> Clouds**.

    *   To create a cloud at any other location, drag the **Clouds** tool to the desired position in the scene window.

2. Click the Model button to switch to the Model room. The Cloud Primitive controls appear.
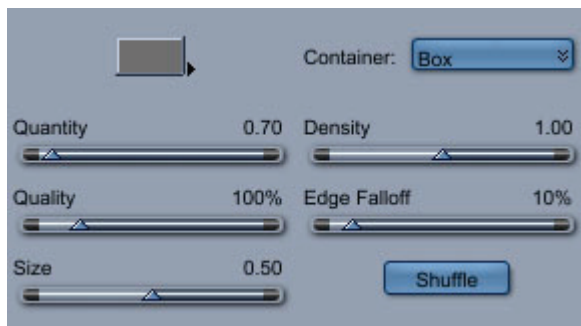
3. Set the Cloud Primitive controls to achieve the desired effect.

    *   **Color** sets the color of the cloud.

    *   **Container** determines the general shape of the cloud. Choose Box, Cylinder, or Sphere from the list.

    *   **Quantity** controls the number of clouds included in the bounding box.

    *   **Quality** controls the quality of the clouds as they are rendered. The higher the quality, the longer the render time.

    *   **Size** controls the size of clouds included in the bounding box.

    *   **Density** controls the density of clouds. A low setting creates almost transparent clouds while a high setting creates almost opaque clouds.

    *   **Edge Falloff** controls the appearance of the edges of the clouds. A low setting results in a sudden change, or shorter falloff. A high setting results in gradual change, or longer falloff.

    *   **Shuffle** randomizes the order in which cloud swirls are generated. For example, if you duplicate a cloud object, the two objects are identical, which looks artificial. To make them look different while leaving all other settings identical, you can shuffle one cloud for

a more natural look.

NOTE! The random order is expressed as the Seed value in the cloud properties.

4. When you are finished, close the Cloud controls and go back to the Assemble or Storyboard room.

Old Volumetric Clouds (Standard, Pro)

The Old Volumetric Clouds feature creates animated volumetric clouds. There are a few tricks to getting them to look right. For example, if bands are appearing in your clouds, you probably need to increase the number of sections.

To create Old Volumetric Clouds:

Choose **Insert > Old Volumetric Cloud** and set the following options:
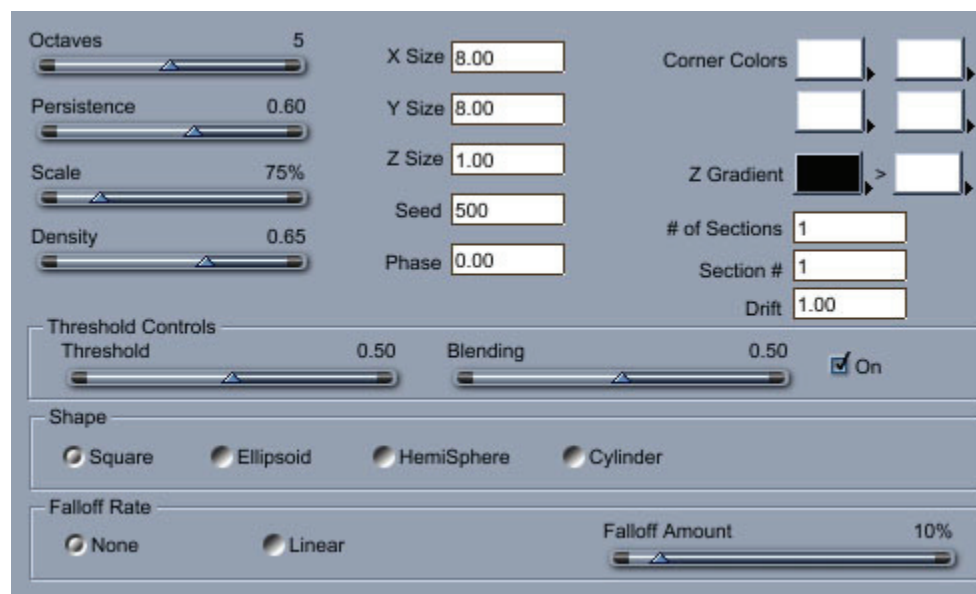
- **Octaves:** Increases the complexity of the clouds. Perlin noise algorithms are used to generate, among other things, clouds, marble, wood, and jitter effects. A universal parameter of Perlin noise is the octave parameter. Each successive octave of noise has a frequency twice that of the previous octave. Therefore, each time you increase the octave parameter, the Perlin Noise becomes more detailed and intricate.

- **Persistence:** Increases the granularity of the clouds. Another parameter that is common to Perlin noise, is persistence. This parameter defines the rate at which the amplitude decreases for each successive level of noise/order. Higher values of persistence lead to greater levels of noise for both regular and reverse Perlin noise.

- **Scale:** Scales the cloud.

- **Density:** Controls how dense the clouds are.

- **Size Controls:** These control the cloud's size in local coordinates. Increasing these will cause the cloud size to increase without stretching, as scaling it in the properties panel would do.

- **Seed:** This value "seeds" the random number generator.

- **Phase:** Controls the animation of the clouds. Start with a phase of 0, move the slider to the end of the animation, and set it to the final value. For slowly changing clouds, increase this by about 10 per second of your animation, and 100 per second for rapidly changing clouds.

- **Color Controls:** Change the color of the clouds at the corners, and the gradient from top to bottom.

- **Section Controls:** If your clouds look banded, you will need multiple sections. Alternatively, you can increase the sizes, then scale the cloud in the Properties panel. This may not always work.

- **Drift:** Set this to zero at the start of your animation, and a higher value at the end. This controls how far the cloud moves.

- **Threshold Controls:** These controls behave identically to the threshold controls in Noise Factory (see "Noise Factory"). The threshold is a middle value, that when enabled, any numbers falling below it will be set to 0, any above it to 1.

- **Shape:** Controls the basic shape of the cloud. Choices are Square, Ellipsoid, HemiSphere, and Cylinder.

- **Falloff Rate:** Higher values will cause the cloud to become increasingly less dense towards its boundary. Using the falloff controls can help reduce banding.

Tips

If the banding is bad, do the following:

1. Get your cloud looking about how you want it to. (Ignore the banding).

2. Set all the animation parameters.

3. Change the number of sections to a higher value. If you are using a lot of falloff, you can keep this at 2 or 3. If you are trying to make very large clouds, you may need as many as ten.

4. Exit the modeler, duplicate your object, edit it.

5. Choose "create new master."

6. Increase the section number.

7. Repeat steps 4 - 6 as necessary.

Fire Primitive

The Fire primitive creates a 3D or volumetric fire. You use this primitive to set a scene or object on fire.
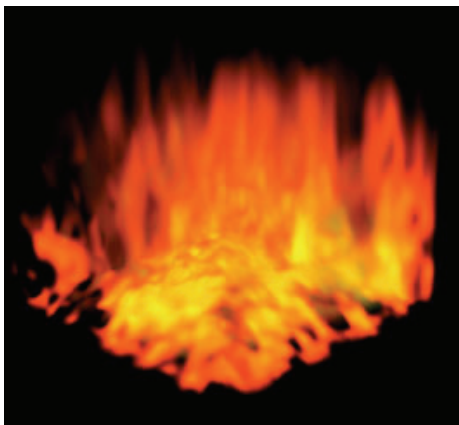
There is no on-screen preview of the fire except its bounding box, which indicates the fire's size and location. You cannot see the fire until it is rendered.

**Tip!** You can do a spot check of the fire using the Test Render tool. Select the tool and drag over any area of the fire.

You can place objects inside the fire or partially within it. Fire attributes can also be animated.

Changing the size of the fire by dragging a larger or smaller bounding box does not change the size of the flames. It only changes the area that the fire covers.

To create a fire object:

1. Depending on the desired location of the fire, do one of the following:

   - To create a fire object of default size at the center of the universe, choose Insert **>Fire**.

   - To create a fire object at any other location, drag the Fire tool to the desired position in the scene window.

2. Click the Model button. The Fire Primitive controls appear.
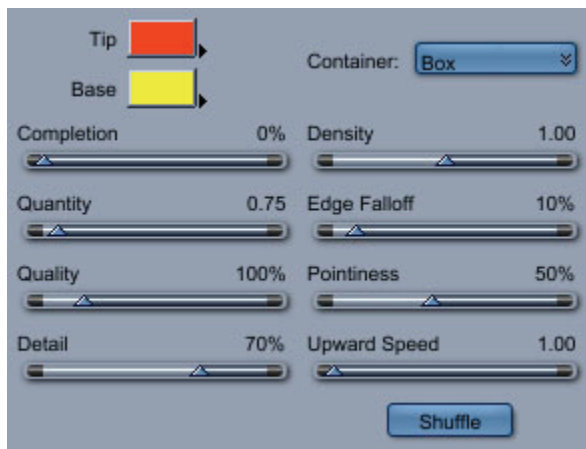
3. Set the Fire Primitive controls to achieve the desired effect.

   - **Tip Color** sets the color of the tips of the flames.

   - **Base Color** sets the color of the base of the flames.

   - **Container** determines the general shape of the fire. Choose a container from the list.

   - **Completion** controls the animation of your fire. Set it to 0% at the start of your animation, and at 100% at the end.

   - **Quantity** controls the quantity of flames that appear in your fire object.

   - **Quality** controls the quality of the fire as it is rendered. The higher the quality, the longer the render time.

   - **Detail** controls the degree of detail in the fire primitive.

   - **Density** controls the density of the fire. A low setting creates fire that is almost transparent. A high setting creates fire that is almost opaque.

   - **Edge Falloff** controls the appearance of the edges of the fire. A low setting results in a sudden change, or shorter falloff. A high setting results in gradual change, or longer falloff.

- **Pointiness** controls the percentage of your fire primitive that is points or tips of flames. A low setting results in few, less-tapered points, while a high setting results in many, more-tapered points.

- **Upward Speed** controls the speed at which the flames appear to shoot up during an animation.

- **Shuffle** randomizes the order in which flames are generated. For example, if you duplicate a fire object, the two objects will be identical, which looks artificial. To make them look different while leaving all other settings identical, you can shuffle one fire object for a more natural look.

NOTE! The random order is expressed as the Seed value in the Fire Properties.

4. When you are finished, close the Fire controls and go back to the Assemble or Storyboard room.

Creating Fountains

The Fountain primitive is a particle system you can use to create objects like geysers or tornadoes. The fountain's attributes let you change the density of its particles and rate of animation.

To create a fountain object:

1. Depending on the desired location of the fountain, do one of the following:

   - To create a fountain of default size at the center of the universe, choose Insert **>Fountain**.

   - To create a fountain at any other location, drag the Fountain tool to the desired position in the scene window.

2. Click the Model button. The Fountain Primitive controls appear.
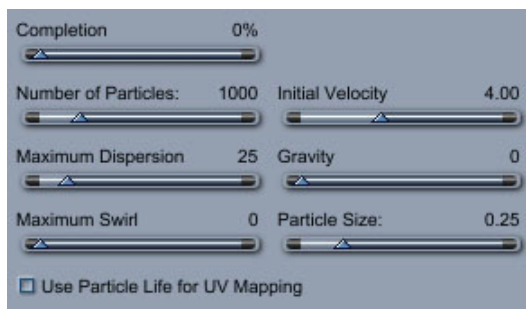
3. Set the Fountain Primitive controls to achieve the desired effect.

- **Completion** controls the fountain's animation. Set it to 0% at the beginning of the animation and 100% at the end.

- **Number of Particles** controls the number of particles in your fountain.

- **Maximum Dispersion** controls the angle of the spray from the fountain. When it is set to 0° the spray goes straight up; at 180° it radiates outward from the fountain in all directions.

- **Maximum Swirl** controls how much the particles rotate parallel to the ground plane as they fall. A low setting causes very little rotation while a high setting causes particles to rotate a great distance as they fall.

- **Initial Velocity** controls the force of the fountain. A low setting creates a slow-spouting fountain while a high setting creates a fast-spouting fountain.

- **Gravity** controls the amount of gravity applied to the fountain. When the setting is low, the particles shoot upward and float. When the setting is high, the particles fall down quickly.

- **Particle Size** controls the size of the particles in your fountain.

- **Use Particle Life for UV Mapping** determines how the shader is mapped to the fountain primitive. When enabled, the shader is mapped over the entire fountain, with the particles near the bottom of the fountain looking different from those nearer the top (depending on the shader content). When disabled, all the particles are mapped individually, so they all appear identical.

4. When you are finished, go back to the Assemble or Storyboard room.

Creating Fog

The Fog primitive creates a 3D, or volumetric fog.

There is no on-screen preview of the fog except its bounding box, which indicates the fog's size and location. You cannot see the fog until it is rendered.

TIP! You can do a spot check of the fog using the Test Render tool. Select the tool and drag over any area of the fog.

You can place objects inside the fog or partially within the fog. You'll probably want your fog object to be fairly large so you can place a portion of your scene inside it. Fog attributes can also be animated.

Scaling the fog object by dragging a larger or smaller bounding box does not scale the swirls of fog themselves. It only changes the area that the fog covers.

NOTE! The Fog primitive is best for creating distinct local areas of fog, while the Cloudy Fog and Distance Fog controls in the **Properties tray: Effects tab: Atmosphere panel** are best for applying fog to the whole scene.

To create a fog object:

1. Depending on the desired location of the fog, do one of the following:

    - To create a fog object of default size at the center of the universe, choose Insert **>Fog**.

    - To create a fog object at any other location, drag the **Fog tool t**o the desired position in the scene window.

2. Click the Model button. The Fog Primitive controls appear.
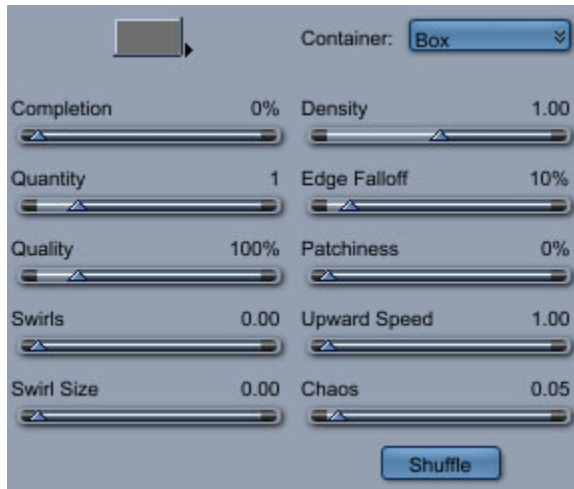
3. Use the Fog Primitive controls to create fog.

4. Set the Fog Primitive controls to achieve the desired effect.

    1. **Color** sets the color of the fog.

    2. **Container** determines the general shape of the fog. Choose a container from the list.

    3. **Completion** controls the animation of the fog. Set this value to 0% at the start of your animation and 100% at the end.

    4. **Quantity** controls how many patches of fog are contained in the bounding box.

    5. **Quality** controls the quality of the fog as it is rendered. The higher the quality, the longer the render time.

    6. **Swirls** controls how the fog swirls or rotates as it rises.

    7. **Swirl Size** controls the size of the swirls.

    8. **Density** controls the amount of light that penetrates the fog.

    9. **Edge Falloff** controls the appearance of the edges of the fog. A low setting results in a sudden change, or shorter falloff. A high setting results in gradual change, or longer falloff.

10. **Patchiness** controls the regularity of the fog throughout the bounding box. A low setting creates a blanket of fog. A high setting creates patches of fog.

11. **Upward Speed** controls the rate at which the fog rises during an animation. Use a higher setting for longer animations.

12. **Chaos** controls the uniformity of the fog. A low setting creates more uniform fog, while a high setting creates fog with random patches and swirls.

13. **Shuffle** randomizes the order in which fog swirls are generated. For example, if you duplicate a fog object, the two objects are identical, which looks artificial. To make them look different while leaving all other settings identical, you can shuffle one fog object for a more natural look.

NOTE! The random order is expressed as the Seed value in the fog properties.

5. When you are finished, close the Fog controls and go back to the Assemble or Storyboard room.

Creating Oceans (Pro only)

To create an ocean:

Depending on the desired location of the terrain, do one of the following:

- To create an ocean of default size at the center of the universe, drag the Ocean tool into the Hierarchy, or choose Insert **> Ocean**.

- To create an ocean at any other location, drag the Ocean tool to the desired position in the scene window.
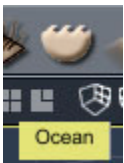
Ocean Properties

The General tab in the Assemble Room displays the following properties for oceans:
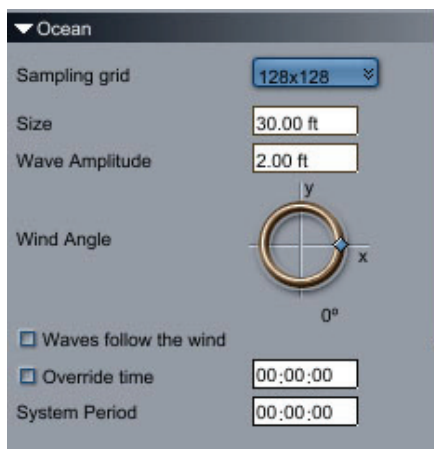
- **Sampling Grid**: Determines the amount of detail that you see in the ocean. Smaller values display less detail than larger values.

- **Size**: Sets the length and width of the ocean (which is a square).

- **Wave Amplitude**: Sets the height of the waves in the ocean.

- **Wind Angle**: Determines the direction from which the wind blows. Drag the indicator along the indicator to change direction, or enter a numerical value between -180 and 180 in the numerical field.

- **Waves Follow the Wind**: Check this option to make the waves respond to the direction of the wind.

- **Override Time**: When checked, you can set the time of the ocean. This allows you to animate the ocean at a different speed from the scene. Otherwise, the scene time is used.

- **System Period**: Defines the time it takes for the ocean's animation to loop.

Import/Export

This chapter describes how you can import/export 3D objects and scenes from/to various file formats. It will show you how you can use Carrara to create content for the web and other applications.

Importing 3D Objects

You may directly import 3D objects from other applications into Carrara using the **File > Import** or the **File > Open** commands. With Import, the object will be imported into the current scene. With Open, the file opens in a new document.

Note Most importers create compact Carrara objects that cannot be edited right away. To edit, select it, go to the modeling room, use the Edit menu: Convert to Other Modeler command, choose Vertex modeler.

3DS Import (Standard, Pro)

To import a 3DS File:

1. Use the **File > Open** command to create a new document, or use the **File > Import** command to add the objects to the current scene.

2. Choose **3DS** for the file format. The 3D Studio Import dialog comes up.

3. Set the following options as desired:

- **AutoPosition: When checked, the imported object is centered on zero in X and Y, and its bottom is set to Z= 0. When unchecked, the values from the file are used.**

- **Disable Auto-scaling**: When checked, use the **One 3DS Unit Equals** field to set the unit conversion factor between the imported file and Carrara. When unchecked, the imported object is scaled to fit in the default working box.

- **Smoothing Angle**: Calculates the normals for the imported objects using this value. A low value will keep most edges sharp, whereas a higher value will create smoother objects.

- **Hide BackFaces**: Speeds up the interactive renderer (works on closed objects like spheres).

- **Show BackFaces**: Displays all the facets of the imported objects in the interactive renderer (needed for surfaces, or objects with incoherent normals).

4. Press **OK**.

3DMF Import (Standard, Pro)

To import 3DMF data:

1. Use the **File > Open** command to create a new document, or use the **File > Import** command to add the objects to the current scene.

2. Choose **3DMF** for the file format. The 3DMF Import dialog comes up.

3. Set the following options as desired:

- **AutoPosition**: When checked, the imported object is centered on zero in X and Y, and its bottom is set to Z= 0. When unchecked, the values from the file are used.

- **Disable Auto-scaling**: When checked, use the **One 3DMF Unit Equals** field to set the unit conversion factor between the imported file and Carrara. When unchecked, the imported object is scaled to fit in the default working box.
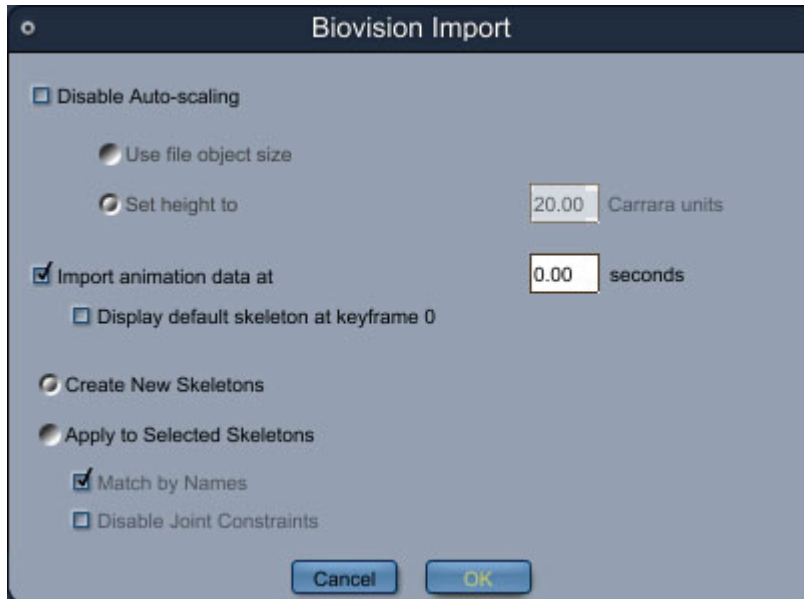
4. Click **OK**.

Biovision BVH Import (Standard, Pro)

To import BVH data:

1. Use the **File > Open** command to create a new document, or use the **File >** Import command to add the objects to the current scene.

2. Choose **BVH** for the file format. The Biovision Import dialog comes up.

3. Set the following options as desired:

    *   **Disable Auto-scaling**: When checked, you can choose to use the original object size described in the file, or set the height of the imported object. When unchecked, the imported object is scaled to fit in the default working box.

    *   **Import animation data at**: When "Create New Skeletons" (described below) is selected, you have the option of importing the animation data to the skeleton in the file. You can also set the time you want the animation to begin, in seconds.

    *   **Display default skeleton at keyframe 0**: When checked, the geometry of the imported object is loaded beginning at keyframe 0. The motion data of the file is loaded starting at keyframe 1.

    *   **Create New Skeletons**: When checked, the file is imported as a new skeleton.

    *   **Apply to Selected Skeletons:** When checked, animation data (optionally, from a different file) is imported to the selected skeleton beginning at the specified time, in seconds.

    *   **Match by Names**: TBD

    *   **Disable Joint Constraints**: TBD

4. Click **OK**.

To apply animation to the skeleton:

1. Select the skeleton.

2. Use the **File > Import** command to import animation from a file.

3. Choose the BVH file with the animation you want to use.

4. Check the **Apply to Selected Skeletons** option.

5. Click **OK**.

Collada Import (Std, Pro)

To import a COLLADA file:

1. Use the **File > Open** command to create a new document or use the **File > Import** command to add the objects to the current scene.

2. Choose **COLLADA (.dae)** for the file format.

3. Locate and select the file to import.

4. Click **OK**.

DAZ Studio (DAZ format)

Carrara allows you to import DAZ Studio (.daz) files into your scene.

To import DAZ Studio files:

1. Use the **File > Open** command to create a new document, or use the **File > Import** command to add the objects to the current scene.

2. Choose DAZ Studio **(*.daz)** for the file format. The DAZ Studio Import dialog comes up.

3. In the Constraints section, check **Use Joint Limits** if you want all poses in your figures to obey the limits set for each joint. Uncheck this option if you are not sure if the existing poses exceed the limits.

4. Click **OK**. The objects appear in your scene.

DXF Import (Standard, Pro)

To import DXF data:

1. Use the **File > Open** command to create a new document, or use the **File > Import** command to add the objects to the current scene.

2.   Choose **DXF** for the file format. The DXF Import dialog will come up.
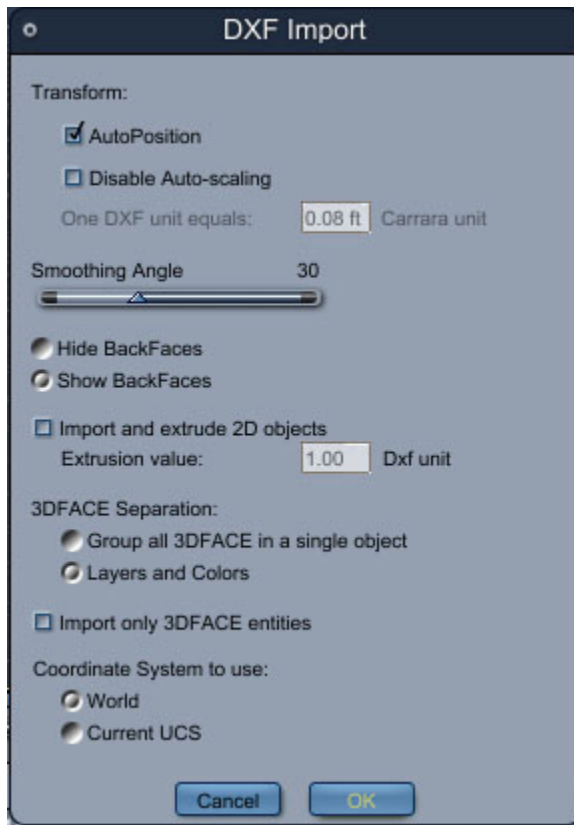
3.   Set the following options as desired:

- **AutoPosition**: When checked, the imported object is centered on zero in X and Y, and its bottom is set to Z= 0. When unchecked, the values from the file are used.

- **Disable Auto-scaling**: When checked, use the **One DXF Unit Equals** field to set the unit conversion factor between the imported file and Carrara. When unchecked, the imported object is scaled to fit in the default working box.

- **Smoothing Angle**: Calculates the normals for the imported objects using this value. A low value will keep most edges sharp, whereas a higher value will create smoother objects.

- **Hide BackFaces**: Speeds up the interactive renderer (works on closed objects like spheres).

- **Show BackFaces**: Displays all the facets of the imported objects in the interactive renderer (needed for surfaces, or objects with incoherent normals).

- **Import and extrude 2D objects**: When checked, 2D objects from the DXF file are imported and given a thickness specified by the Extrusion value. When unchecked, 2D objects are ignored.

- **3DFACE Separation**: You can choose to import all the DXF 3DFACE(s) in a single Carrara object, or into multiple Carrara objects that are separated by layers and colors.

- **Import only 3DFACE entities**: When checked, any DXF object that is not a 3DFACE is ignored.

- **Coordinate System to Use**: You can choose to use World or Current UCS coordinate system.

4. Click **OK**.

DirectX Import (Pro only)

To import DirectX files:

1. Use the **File > Open** command to create a new document, or use the **File > Import** command to add the objects to the current scene.

2. **Choose DirectX (*.x)**for the file format. The DirectX Import dialog comes up.

3. Set the following options as desired:

- **Disable Auto-scaling**: When checked use the One DirectX Unit Equals field to set the unit conversion factor between the imported file and Carrara 6. When unchecked, the imported object is scaled to fit in the default working box.

4. Click **OK**.

Filmbox FBX Import (Pro only)

To import FBX files:

1. Use the **File > Open** command to create a new document, or use the **File > Import** command to add the objects to the current scene.

2. Choose **FBX** for the file format. The FBX Import dialog comes up.

3. Set the following options as desired:

    - **AutoPosition:** When checked, the imported object is centered on zero in X and Y, and its bottom is set to Z= 0. When unchecked, the values from the file are used.

    - **Disable Auto-scaling:** When checked you can set the unit conversion factor between the imported file and Carrara 6. When unchecked, the imported object is scaled to fit in the default working box.
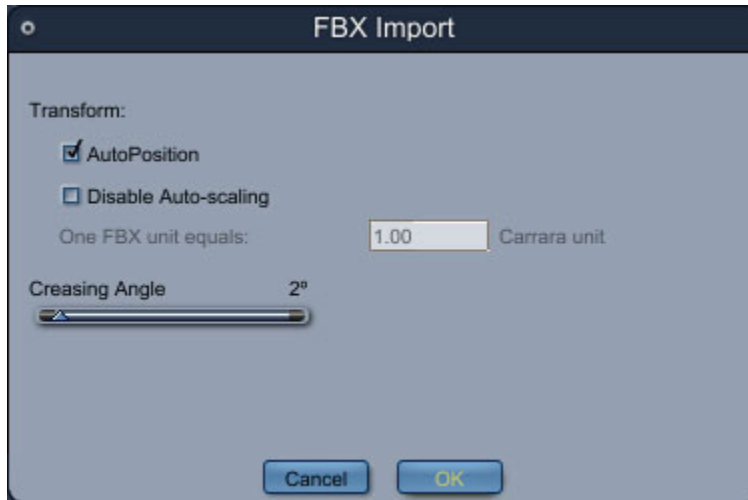
4. Click **OK**.

What is Imported

- Skeletons are imported with their motion data.

- A vertex primitive is created for each model in the FBX file (patches and Nurbs are converted to polygons). It is automatically attached to its skeleton (if any)using the weights from the file.

Takes

- Sometimes, an FBX file contains several animations or takes. In this case, a dialog enables you to select the take you want to import. After you make your selections, choose OK to import the file.

Limitations

Some FBX files may contain skeletons made of several pieces, or models influenced by something else than a bone. With such structures, the skinning information is not imported.

- Additive skinning mode is not supported.

- Morph targets and shading are not imported.

- UV coordinates are not imported.

Characters with Control Rigs

In Motion Builder, animating a character is generally done by editing a control rig instead of directly manipulating the bones. As this is not supported by the FBX importer, you need to "Plot" your characters in Motion Builder before you save the FBX file. Please refer to the documentation of Motion Builder for more information on plotting characters.

Lightwave (LWO Import) (Standard, Pro)

To import LWO files:

1. Use the **File > Open** command to create a new document, or use the **File > Import** command to add the objects to the current scene.

2. Choose **FBX** for the file format. The Lightwave Import dialog comes up.
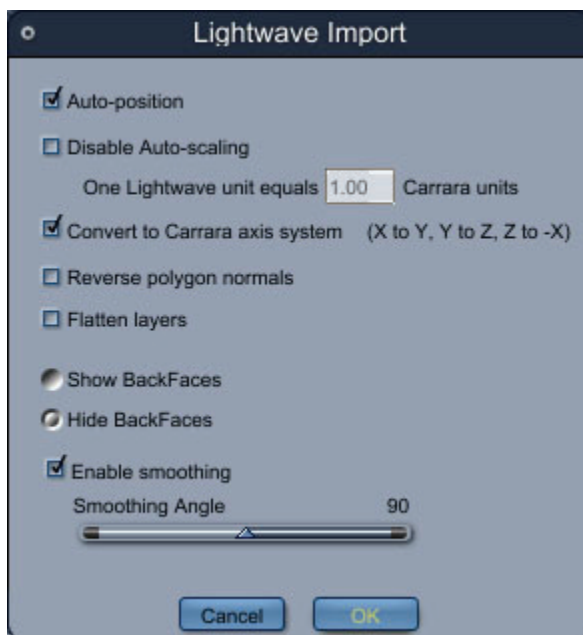
3. Set the following options as desired:

- **AutoPosition:** when checked, the imported object is centered on zero in X and Y, and its bottom is set to Z=0. When unchecked, the values from the file are used.

- **Disable Auto-scaling:** when checked, you can set the unit conversion factor between the imported file and Carrara 6. When unchecked, the imported object is scaled to fit in the default working box.

- **Convert to Carrara axis system:** when checked, transforms the coordinates of the imported object so that its bottom is horizontal in Carrara. When unchecked, the coordinates are imported without modification.

- **Reverse polygon normals:** Some object files are stored in such a way that their normals are backwards, facing the inside of a convex object. Checking this box will cause Carrara to reverse the normals of the imported object.

- **Flatten layers:** Checking this box causes the importer to disregard layer information in Lightwave files, creating objects all in one group, rather than creating a group for each Lightwave layer.

- **Show/Hide Backfaces:** Toggles whether the backfaces of the object will be displayed by default in the 3D view.

- **Enable Smoothing:** When checked, allows the user to select the maximum angle between faces to be smoothed. An angle of 0 results in sharp transitions between all polygon edges. Note that subdivision surfaces are always imported with smoothing, regardless of setting.

4. Click **OK**.

Wavefront OBJ Import

To import OBJ files:

1. Use the **File > Open** command to create a new document, or use the **File > Import** command to add the objects to the current scene.

2. Choose **OBJ** for the file format. The OBJ Import dialog comes up.
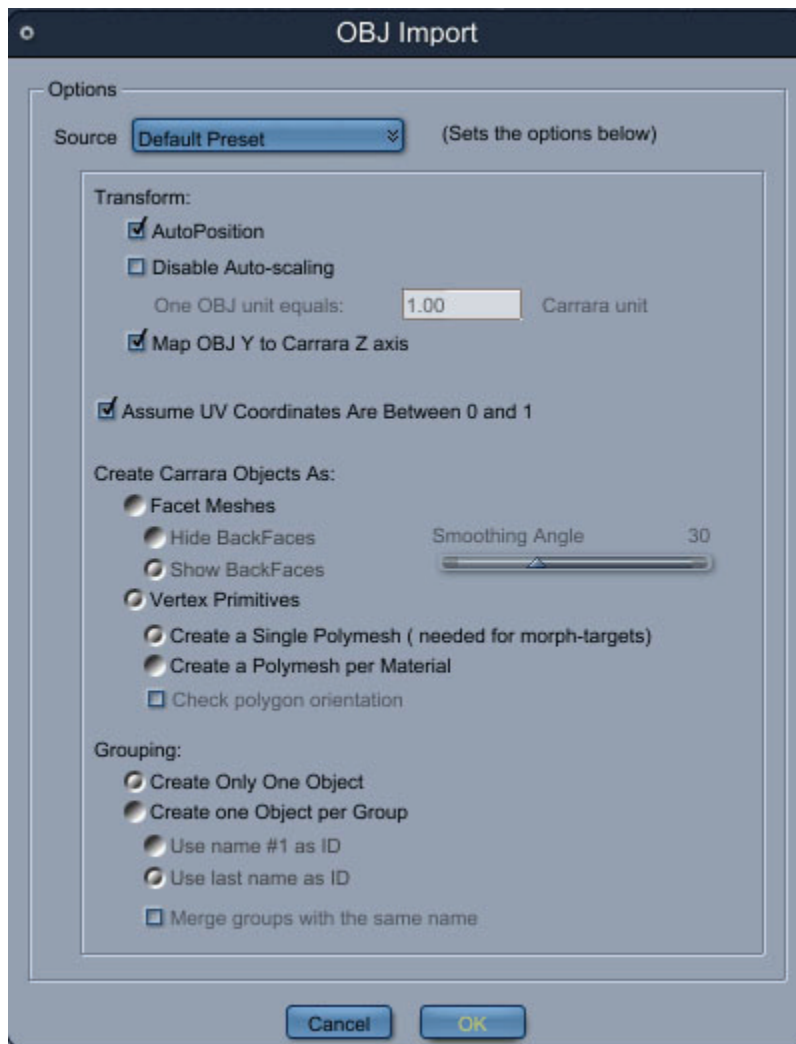
3. Set the following options as desired:

- **AutoPosition**: when checked, the imported object is centered on zero in X and Y, and its bottom is set to Z= 0. When unchecked, the values from the file are used.

- **Disable Auto-scaling**: when checked, you can set the unit conversion factor between the imported file and Carrara. When unchecked, the imported object is scaled to fit in the default working box.

- **Map OBJ Y to Carrara Z axis**: when checked, transforms the coordinates of the imported object so that its bottom is horizontal in Carrara. When unchecked, the coordinates are imported without modification.

- **Assume UV Coordinates Are Between 0 and 1**: In most cases, you shouldn't need to change this option. When checked, the object's texture coordinates are imported without modification. When unchecked, the texture coordinates are transformed so they fit between 0 and 1.

- **Create Carrara Objects As**: You can choose to import your objects as:

- **Facet Meshes**: use less memory but cannot be edited without conversion. The import options used in this case are:

- **Hide BackFaces**: speeds up the interactive renderer (works on closed objects like spheres).

- **Show BackFaces**: displays all the facets of the imported objects in the interactive renderer (needed for surfaces, or objects with incoherent normals).

- **Smoothing Angle**: If needed, calculates the normals for the imported objects using this value. A low value will keep most edges sharp, whereas a higher value will create smoother objects.

- **Vertex Primitives**: can be edited in the Vertex modeler. You can choose to :

- **Create a Polymesh per Material**: creates a polymesh and a shader for every material in the OBJ file. This is needed to properly import OBJ files with their shading information.

- **Create a Single Polymesh**: this will create only one polymesh and is what should be used if you are doing morph targets for Poser.

- **Create Only One Object**: When selected, a single Carrara object will be used to store the imported geometry.

- **Create one Object per Group**: When selected, a Carrara object will be created for each group in the imported file. As groups can have several names, you can choose to use the first or the last ones to name the new Carrara objects.

4. Click **OK**.

Poser® Native Import

You can now directly import Poser content without the need for the TransPoser plug-in: Poser cameras (CM2/CMZ), characters (CR2/CRZ), Poser prop hair (HR2/HRZ), Poser lights (LT2/LTZ), Poser props (PP2/PPZ) and Poser scenes (PZ3/PZZ) and also all other content linked poses or clothes.

When using the Poser Native Importer, you can, in Carrara, edit the imported objects (adjust morph targets, move the objects, move the joints, import a pose or a cloth, edit a shader, etc.). For instance, if you import a Poser character into Carrara, it will create the model of the character and also attach a skeleton to it and import all morph targets. You can edit all this directly in Carrara. The Poser Native Importer does not support dynamic clothes and dynamic hair.

Note You will find it much easier to use the Content Browser to add DAZ and Poser content into your scenes. See "Using Content and Runtime Folders."

Carrara allows you to import Poser content in the formats that you commonly find in your Poser libraries. Refer to the following sections for specific information:

Importing Poser Cameras (CM2/CMZ)

Poser camera files are imported with Poser scenes, but you can also import them individually.

To import a Poser camera file:

1. Choose **File > Import.** The Open dialog appears.

2. Poser camera files are typically located in **Runtime > Libraries > Camera** subfolders and have a .CM2 or .CMZ extension. Locate the folder that contains the camera file that you want to open. Highlight the file and click Open. The Poser Camera Import dialog appears.

3. Carrara prompts you to select the path to the Runtime folder that your camera appears in. Click the file selector button to locate the path to the runtime folder.

4. When you return to the Poser Camera Import dialog, click OK. The camera appears in your scene.

5. To switch to the new camera view, select the desired camera from the menu in the upper-left corner of the document window.


Importing a Poser Character (CR2/CRZ)

You can also open Poser content such as Poser Characters. The available options are similar to those used when importing a scene (see "Importing a Poser Scene (PZ3/PZZ)." However, a Poser character does not contain any animation.

Poser character files are poseable figures, most often seen as human figures, animals, and poseable clothing. When you import a character, you are given the option to import morphs and deformers that are associated with the figure. The figures will import in their default pose, and in the default position.

To open or import a Poser character file:

1. Choose **File > Open** or **File > Import.** The Open dialog appears.

2. Poser character files are typically located in **Runtime > Libraries > Character** subfolders and have a .CR2 or .CRZ extension. Locate the folder that contains the character file that you want to open. Highlight the file and click Open. The Poser Character Import dialog appears.
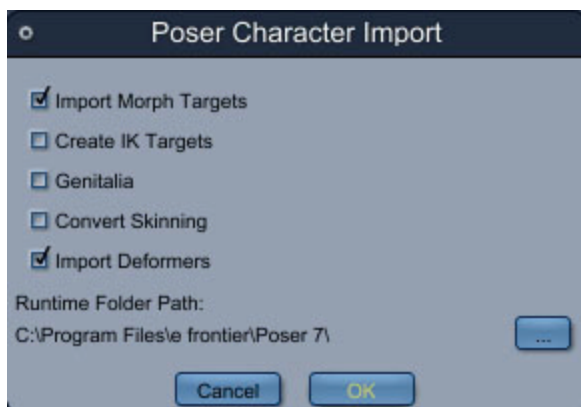
3. Check or uncheck the following options as desired:

   • **Import Morph Targets**: Check this option if you want to include morph targets with your Poser content when you open or import the file.

   • **Create IK Targets**: Check this option if you want to import the inverse kinematic settings for the character's hands and feet.

   • **Genitalia**: Check this option if your figure uses separate geometry for genitalia and you want to import the figure with the alternate genitalia geometry.

   • **Convert Skinning**: Converts the Poser skinning to the standard Carrara skinning using a single weight map.

   • **Import Deformers**: Check this option if you want to import deformers (magnets or wave deformers) that are attached to your figure.

4. Click the Browse for Folder button to select your Poser Runtime Path. If you are running Poser 7 on a Windows operating system, the default path to the Poser executable file is **C:\Program Files\e frontier\Poser 7**. After you select the file click OK to return to the Poser Character Import dialog.

5. Click OK to start the import process. A progress bar appears while Carrara imports the file into a new scene.
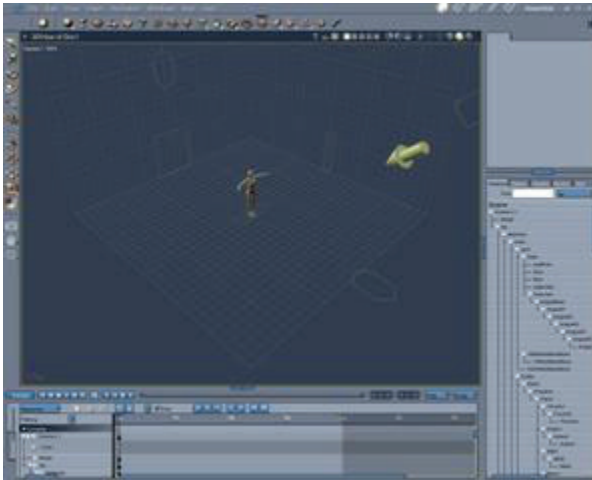
6. Then you can select the character in your scene to access the character properties. Those properties are available in the **Properties tray: General tab**.
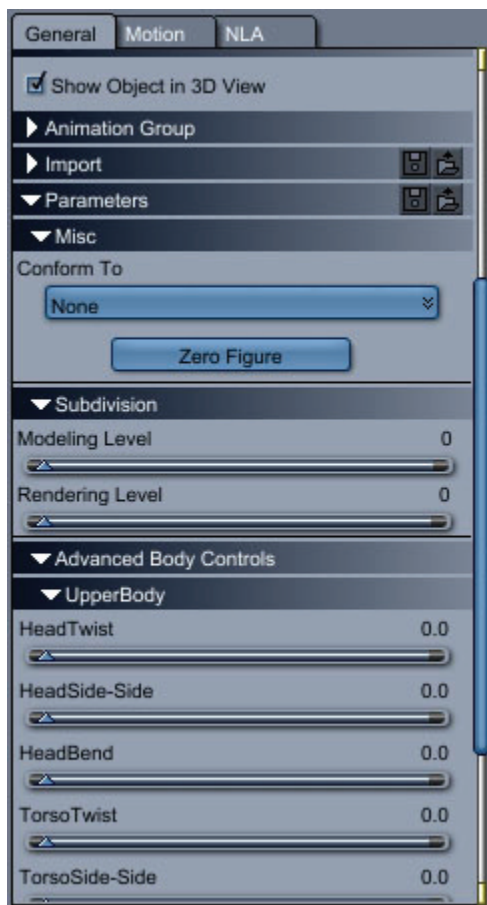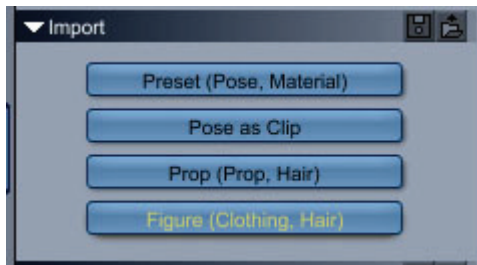

Importing Options in the Properties Tray

Once you have imported a character, you can also use the **Properties tray: General tab: Import panel** to import poses, props, hair, clothing, and figures.

With the main character selected, open the **Properties tray: General tab: Import panel**. There you will see four buttons:


Poser import options in the Properties tray.

- **Preset (Pose, Material)**: Use this button to import pose files that can be applied to the current figure.

- **Pose as Clip**: Use this button to import an animated pose as an NLA animation clip.

- **Prop (Prop, Hair)**: Use this button to import props or prop hair.

- **Figure (Clothing, Hair)**: Use this button to import conforming clothing or hair.

Conforming Clothing

Conforming clothing automatically follows a poseable figure, but you have to conform the clothing to the figure in order for this to happen. In some cases, the clothing conforms automatically. However, if you find that you need to manually conform it, you can find an option in the General tab to accomplish this.

After you load your clothing (which should be designed for the character you are conforming it to), open the **Properties tray: General tab: Misc panel**. A Conform To button appears there. Click the button and choose the desired character from the list.
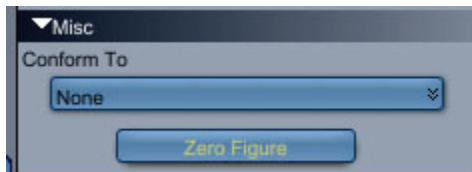
TIP! You can remove conformed clothing using the **Unconform** button.

Importing Poser Hair (HR2/HRZ)

Poser hair objects are prop files that are typically located in a **Runtime > Libraries > Hair** folder.

1. Choose **File > Open** or **File > Import**. The Open dialog appears.
2. Poser hair files are typically located in **Runtime > Libraries > Hair** subfolders and have a .HR2 or .HRZ extension. Locate the folder that contains the hair file that you want to open. Highlight the file and click Open.
3. You may be prompted to locate the geometry file that is associated with the HR2 file. If so, a dialog will display the name of the OBJ file that it needs. The geometry files are typically located in a subfolder beneath the **Runtime > Geometries** folder, in a folder named by the content creator. Highlight the OBJ file and click Open. The hair appears in your scene.
4. In the Assemble tab, position the hair if necessary. Then go to the Instance list and locate the hair that you imported. Drag the hair upward until you locate the figure's Head body part, and drop the

hair onto the head. The hair will then attach to the head so that it moves along with the figure when you pose it.

Importing Poser Lights (LT2/LTZ)

Poser light files are imported with Poser scenes, but you can also import them individually.

To import a Poser light file:

1. Choose **File > Import.** The Open dialog appears.

2. Poser camera files are typically located in **Runtime > Libraries > Camera** subfolders and have a .LT2 or .LTZ extension. Locate the folder that contains the light file that you want to open. Highlight the file and click Open. The Poser Light Import dialog appears.

3. Carrara prompts you to select the path to the Runtime folder that your light appears in. Click the file selector button to locate the path to the runtime folder.

4. When you return to the Poser Light Import dialog, click **OK**. The light appears in your scene.

NOTE! Poser lights may need some adjustment in intensity, color, and falloff after importing, and the amount of adjustment can also vary due to changes between Poser versions. Start by reducing the light intensity to 25-30% of the original intensity and then increase or decrease as necessary to get the desired affect.

Importing Poser Props (PP2/PPZ)

To import a Poser character file, follow these steps:

1. Choose **File > Open** or **File > Import.** The Open dialog appears.

2. Poser prop files are typically located in **Runtime > Libraries > Prop** subfolders and have a .PP2 or .PPZ extension. Locate the folder that contains the prop file that you want to open. Highlight the file and click Open. The Poser Prop Import dialog appears.

3. Check or uncheck the following options as desired:

- **Create IK Targets**: Check this option if you want to import the inverse kinematic settings for the character's hands and feet.

- **Convert Skinning**: Converts the Poser skinning to the standard Carrara skinning using a single weight map.

- **Import Deformers**: Check this option if you want to import deformers (magnets or wave deformers) that are attached to your figure.

4. Click the Browse for Folder button to select your Poser Runtime Path. If you are running Poser 7 on a Windows operating system, the default path to the Poser executable file is **C:\Program Files\e frontier\Poser 7**. After you select the file click **OK** to return to the Poser Character Import dialog.

5. You may be prompted to locate the geometry file that is associated with the prop file. If so, a dialog will display the name of the OBJ file that it needs. The geometry files are typically located in a subfolder beneath the **Runtime > Geometries** folder, in a folder named by the content creator. Highlight the OBJ file and click **Open**. The prop appears in your scene.

Importing a Poser Scene (PZ3/PZZ)

Poser scene files can contain one or several characters, props, clothing, poses, and other content that is already posed. After you import a Poser scene into Carrara, you can add animation and other enhancements to the scene that you import. In addition, you can add and conform clothing, apply pose files, and make other scene changes by adding additional content in the Content Browser.

To import a Poser scene:

1. Choose **File > Open** or **File > Import.** The Open dialog appears.

2. Locate the folder into which you saved your Poser scene file. The file will have a PZ3 or PZZ extension.

3. Highlight the file and click Open. The Poser Scene Import dialog appears.

4. Check or uncheck the following options as desired:

- **Import Morph Targets**: Check this option if you want to include morph targets with your Poser content when you open or import the file.

- **Create IK Targets**: Check this option if you want to import the inverse kinematic settings for the character's hands and feet.

- **Import Animation**: Check this option if the Poser scene file contains an animation and you want to import all of the animation frames. If you uncheck this option, Carrara will only import the first frame.

- **Genitalia**: Check this option if your figure uses separate geometry for genitalia and you want to import the figure with the alternate genitalia geometry.

- **Convert Skinning**: Converts the Poser skinning to the standard Carrara skinning using a single weight map.

- **Import Deformers**: Check this option if you want to import deformers (magnets or wave deformers) that are attached to your figure.

5. Click the Browse for Folder button to select your Poser Runtime Path. If you are running Poser 7 on a Windows operating system, the default path to the Poser executable file is **C:\Program Files\e frontier\Poser 7**. After you select the file click OK to return to the Poser Character Import dialog.

6. Click Open to start the import process. A progress bar appears while Carrara imports the file into a new scene.

7. You may be prompted to locate files that Carrara cannot find. You are given the option to stop looking for these files which will cause Carrara to import only the files that it was able to locate.

Realviz XML Import (Pro only)

You can import Realviz XML data (RZML file format). This format is used to transfer camera motion data calculated from a movie in a match moving software. Note that only the motion of the camera, 3D trackers and basic geometry are imported.

To import a Realviz File:

1. Use the **File > Open** command to create a new document, or use the **File > Import** command to add the objects to the current scene.

2. Choose Realviz for the file format. The Realviz XML Import dialog comes up.
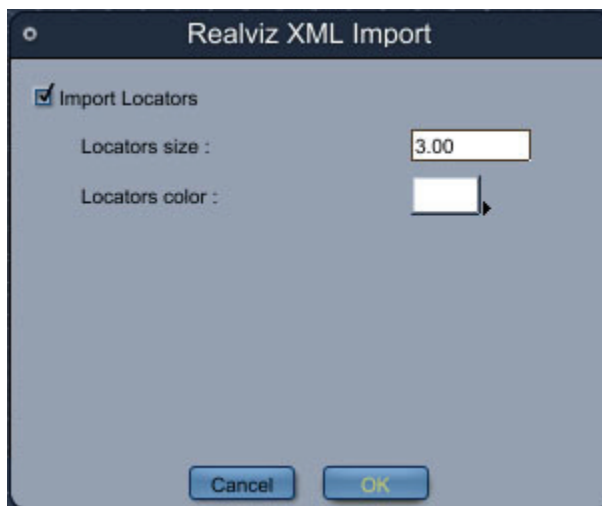
3. Set the following options as desired:

- **Import Locators**: when checked, the 3D trackers are imported as target helper objects.

- **Locators size**: determines the display size of the created helper objects.

- **Locators color**: determines the display color of the created helper objects.

4. Click **OK**.

SynthEyes™ Import (Pro only)

You can import SynthEyes data (.syncar file format). This format is used to transfer camera motion data calculated by the SynthEyes match moving software.

Note that only the motion of the camera and 3D trackers are imported.

To download the latest Carrara 6 export script for SynthEyes, please go to: http://ssontech.com/

TrueSpace Import (Pro only)

To import trueSpace data (COB file format):

1. Use the **File > Open** command to create a new document, or use the **File > Import** command to add the objects to the current scene.

2. Choose **COB** for the file format. The TrueSpace Import dialog comes up.
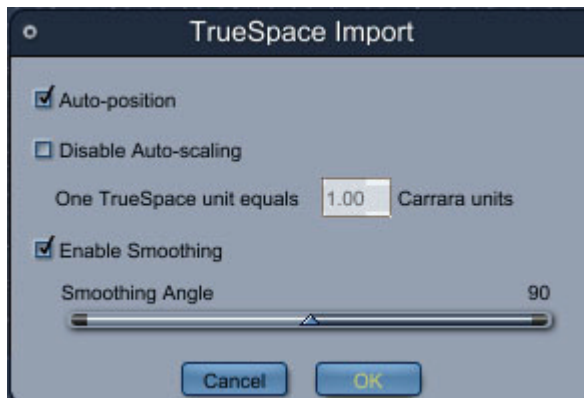
3. Set the following options as desired:

   - **AutoPosition:** when checked, the imported object is centered on zero in X and Y, and its bottom is set to Z = 0. When unchecked, the values from the file are used.

   - **Disable Auto-scaling:** when checked, you can set the unit conversion factor between the imported file and Carrara 6. When unchecked, the imported object is scaled to fit in the default working box.

   - **Enable Smoothing:** calculates the normals for the imported objects using the smoothing angle. A low value will keep most edges sharp, whereas a higher value will create smoother objects.

4. Click **OK**.

CAD Formats (Pro only)

Carrara 6 also allows you to import the following CAD formats. The import options are similar for all of these formats:

- IGES (.igs, .iges)
- OpenNurbs (.3dm)
- SAT (.sat)
- STEP (.stp, .step)

- VDAFS (.vda, .vdafs)

To import CAD format files:

1. Use the **File > Open** command to create a new document, or use the **File > Import** command to add the objects to the current scene.

2. Choose any of the CAD file types for the file format. The corresponding import dialog comes up.
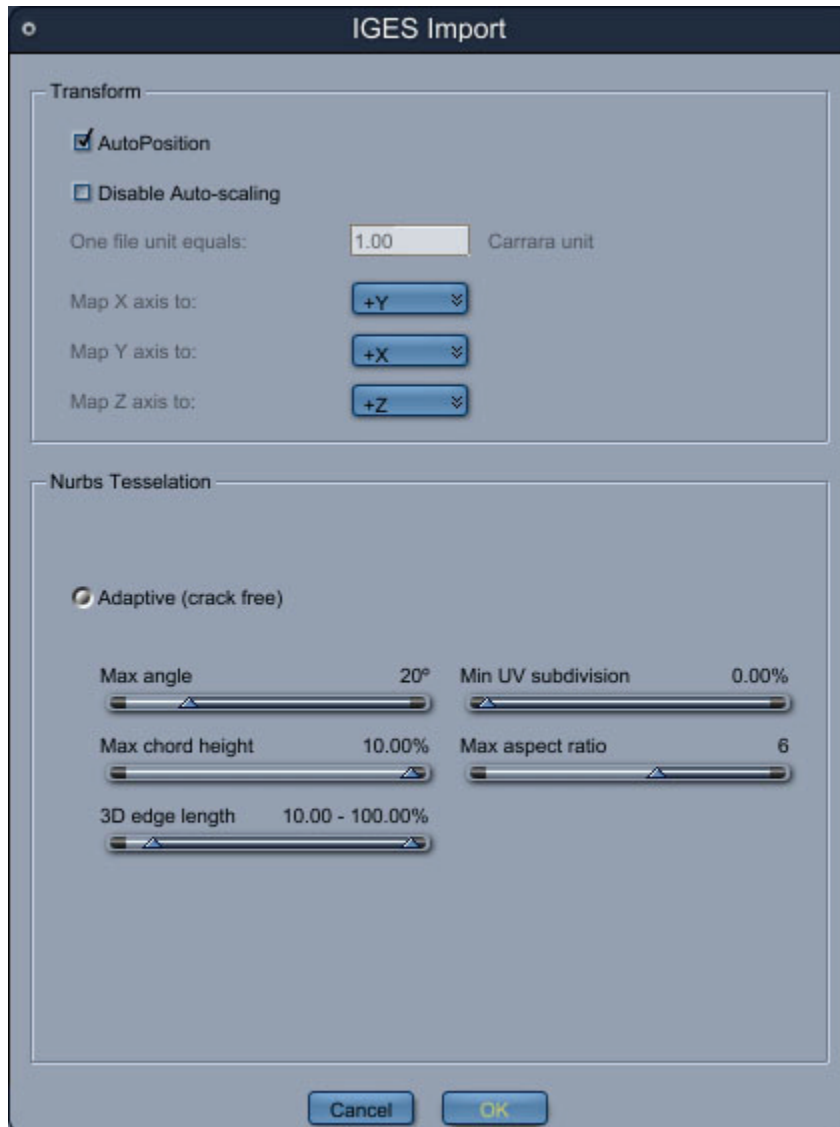
3. Set the following options as desired:

   - **AutoPosition**: when checked, the imported object is centered on zero in X and Y, and its bottom is set to Z= 0. When unchecked, the values from the file are used.

   - **Disable Auto-scaling**: when checked, you can set the unit conversion factor between the imported file and Carrara. When unchecked, the imported object is scaled to fit in the default working box.

   - **Map X, Y or Z Axis to**: TBD

   - **Nurbs Tesselation**: TBD

- **Adaptive (crack free)**: TBD
- **Max angle**: TBD
- **Max chord height**: TBD
- **3D edge length**: TBD
- **Min UV subdivision**: TBD
- **Max aspect ratio**: TBD

4. Click **OK**.

Other Formats

The following formats can also be imported by Carrara:

- Infini-D 3.0-4.5 (.id4, .ids, .id0)
- RayDream 3 (.rd3, .d3d)
- Ray Dream 4 (.rd4)
- Ray Dream 5 (.rds, .rdd)

Exporting 3D Objects

You can export 3D objects to a variety of different file formats using the **File > Export** or the **File > Save As** commands. When you use export, only the current selection is exported. When you choose Save As, the whole scene is exported.

Exporting 3D objects allows you to interact with other 3D applications. It also allows you to create 3D content for the Web.

The following sections provide additional information on specific export file types.

NOTE! Wavefront OBJ format is the only export format available for Carrara 3D Express.

3DS Export (Standard, Pro)

To export to 3DS:

1. Use the **File > Save As** command to export the whole scene, or use the **File > Export** command to export the current selection.

2. Choose the 3DS file format, type a file name, and click on **Save**. The 3D Studio Export dialog comes up.

3. Check on the options controls to change the export settings.

- **Scaling**: Rescales all the exported objects based on the value entered in the **One Carrara Unit Equals** field.

- **Surface Fidelity**: For objects with multiple levels of detail (like Splines), sets the precision of the export. A 100% value will export highly detailed meshes, while a lower value will export fewer facets.

- **Textures Format**: Selects the image format to use when saving texture maps.

- **Convert Procedural Shaders to Textures**: When checked, a texture map is calculated for each procedural shader used in the Carrara scene. This option can slow down the export quite a bit, especially if you select a high resolution.

4. Press **OK.**

3DMF Export (Standard, Pro)

To export to 3DMF:

1. Use the **File > Save As** command to export the whole scene, or use the **File > Export** command to export the current selection.

2. Choose the 3DMF format, type a file name, and click on **Save**. The 3DMF Export dialog comes up.
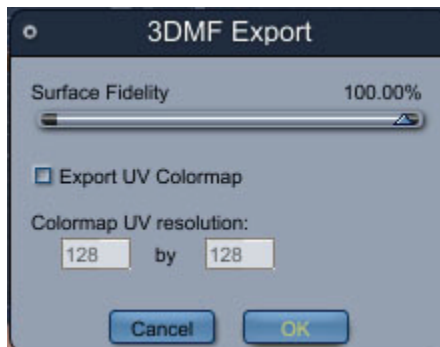
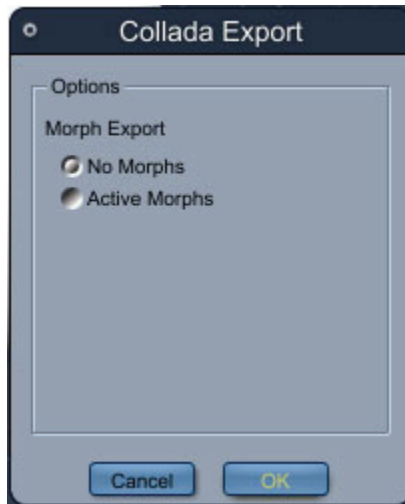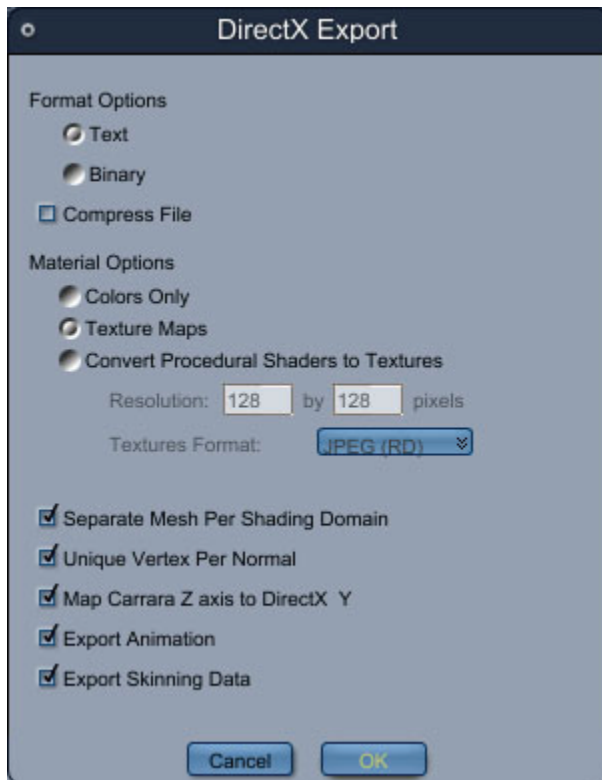3. Check on the options controls to change the export settings.

- **Surface Fidelity**: for objects with multiple levels of detail (like Splines), sets the precision of the export. A 100% value will export highly detailed meshes, while a lower value will export fewer facets.

- **Export UV Colormap**: when checked, a texture map is calculated for each procedural shader used in the Carrara scene. Enter the desired width and height resolution in the fields provided. This option can slow down the export quite a bit, especially if you select a high resolution.

4. Press **OK**.

Adobe® After Effects® Export (Pro only)

The After Effects® exporter saves the animation of cameras, lights (distant, bulb and spot), and planes into a ".cae" file. The generated file can then be imported into After Effects by using the Carrara plug-in for After Effects (5.5 and above on Windows, 6.5 and above on Mac OS X).

1. Use the **File > Save As** command to export the whole scene, or use the **File > Export** command to export the current selection.

2. Choose the **After Effects (*.cae)** format, type a file name, and click on **Save**. The AE Export dialog comes up.
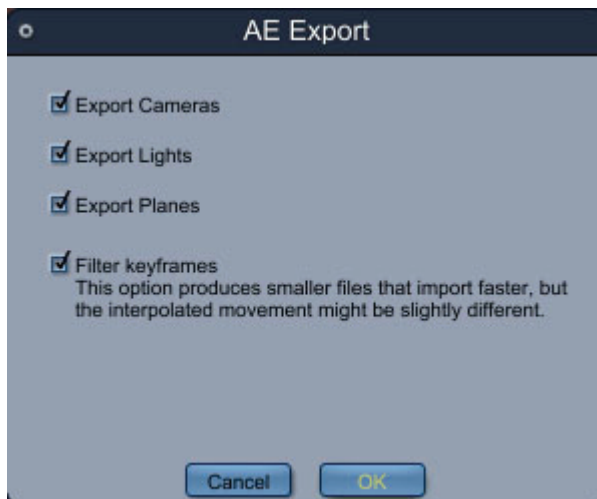
3. Check on the options controls to export cameras, lights, and planes, and to filter keyframes (if desired).

4. Press **OK**.

COLLADA Export

To export to COLLADA:

1. Use the **File > Save As** command to export the whole scene or use the **File > Export** command to export the current selection.

2. Choose the **COLLADA (.dae)** file format, type a file name, and click on **Save**. The DXF Export dialog comes up.

3. To export morphs as part of the COLLADA file, click in the radio button next to **Active Morphs**. To not export morphs, click in the radio button next to **No Morphs**.

4. Press **OK**.

DXF Export (Standard, Pro)

To export to DXF:

1. Use the **File > Save As** command to export the whole scene, or use the **File > Export** command to export the current selection.

2. Choose the **DXF** file format, type a file name, and click on **Save**. The DXF Export dialog comes up.

3. Set the **Surface Fidelity**: for objects with multiple levels of detail (like Splines), sets the precision of the export. A 100% value will export highly detailed meshes, while a lower value will export fewer facets.

4. Press **OK**.

DirectX Export (Pro only)

To export to DirectX:

1. Use the **File > Save As** command to export the whole scene, or use the **File > Export** command to export the current selection.

2. Choose **DirectX** format, type a file name, and click on **Save**. The DirectX Export dialog comes up.

3. Check on the options controls to change the export settings.

    - **Format Options**: Choose Text or Binary format.

    - **Compress File**: When checked, exports the DirectX object in compressed format.

    - **Colors Only**: Exports basic colors only. Does not export texture maps.

    - **Texture Maps**: Choose this option to convert texture maps during the export. Enter the desired size in the **Resolution** fields, and select a file format using the **Textures Format** popup menu.

    - **Convert Procedural Shaders to Textures**: When checked, a texture map is calculated for each procedural shader used in the Carrara scene. This option can slow down the export, especially for high resolutions

    - **Separate Mesh Per Shading Domain**: TBD

    - **Unique Vertex Per Normal**: TBD

    - **Map Carrara Z Axis to DirectX Y**: TBD

    - **Export Animation**: TBD

    - **Export Skinning Data**: TBD

4. Press **OK**.

Kaydara FBX Export (Pro only)

You can export geometry, bones and skinning in the FBX format.

1. Use the **File > Save As** command to export the whole scene, or use the **File > Export** command to export the current selection.

2. Choose **Kaydara FBX** file format, type a file name, and click on **Save**. The FBX Export dialog comes up.
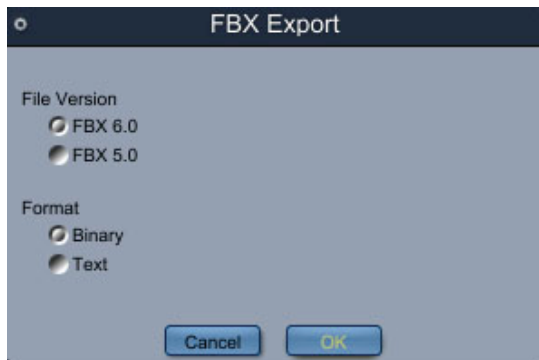
3. In the options dialog, you can choose between **FBX 5** or **FBX 6**, as well as **binary** or **text** mode.

4. Press **OK**.

Shockwave 3D Export (Standard, Pro)

To export Shockwave 3D format:

1. Shockwave 3D allows you to create 3D content for Macromedia Director.

2. Use the **File > Save As** command to export the whole scene, or use the File > Export command to export the current selection.

3. Choose **Shockwave** file format, type a file name, and click on **Save**. The Shockwave Export dialog comes up.
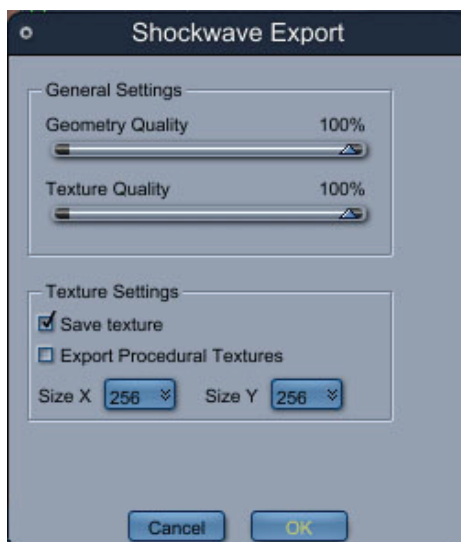
4. Check on the options controls to change the export settings.

- **Geometry Quality**: Controls the accuracy of the geometry export.

- **Texture Quality**: Adjust the slider to increase or decrease the quality of the exported texture.

- **Save Texture**: When checked, exports textures with the file. Choose the desired size for the texture in the X and Y popup menus.

- **Export Procedural Textures**: When checked, a texture map is calculated for each procedural shader used in the Carrara scene. This option can slow down the export, especially for high resolutions.

5. Press **OK**.

VRML Export (Standard, Pro)

To export to VRML:

1. Use the **File > Save As** command to export the whole scene, or use the File > Export command to export the current selection.

2. Choose the **VRML** file format, type a file name, and click on **Save**. The VRML 2.0 Export dialog comes up.
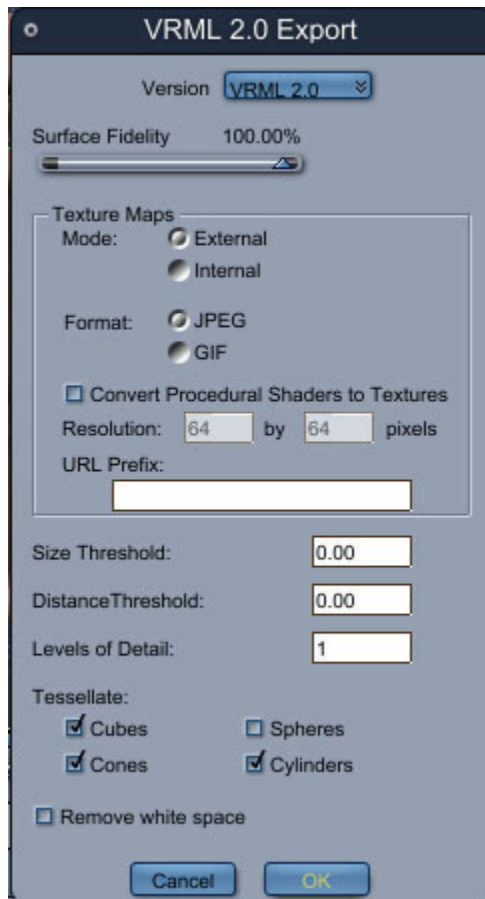
3. Check on the options controls to change the export settings.

- **Version**: Exports to VRML 1.0 or 2.0.

- **Surface Fidelity**: For objects with multiple levels of detail (like Splines), sets the precision of the export. A 100% value will export highly detailed meshes, while a lower value will export fewer facets.

- **Texture Maps**:

    - **Mode**: You can choose to save the texture map inside the VRML file (Internal mode) or outside it (External mode).

    - **Format**: The available formats for the texture map files are JPEG and GIF.

    - **Convert Procedural Shaders to Textures**: When checked, a texture map is calculated for each procedural shader used in the Carrara scene. This option can slow down the export, especially for high resolutions.

    - **URL Prefix**: this string will be added before each texture map file name appearing in the exported VRML file.

- **Size Threshold**: TBD

- **Distance Threshold**: TBD

- **Levels of Detail**: TBD

- **Tessellate**: you can choose to export basic primitives (Cubes, Spheres, Cones, etc.) as faceted objects by checking the control with the name of the primitive. Otherwise, default VRML primitives are used.

- **Remove white space**: when checked, the exported VRML file is not indented (and is smaller). When checked, the VRML file is indented for easier human reading in a text editor.

4. Press **OK**.

Wavefront OBJ Export

NOTE! This is the only export format available for Carrara 3D Express.

To export to Wavefront (OBJ):

1. Use the **File > Save As** command to export the whole scene, or use the **File > Export** command to export the current selection.

2. Choose the **OBJ** file format, type a file name, and click on **Save**. The OBJ Export dialog comes up.
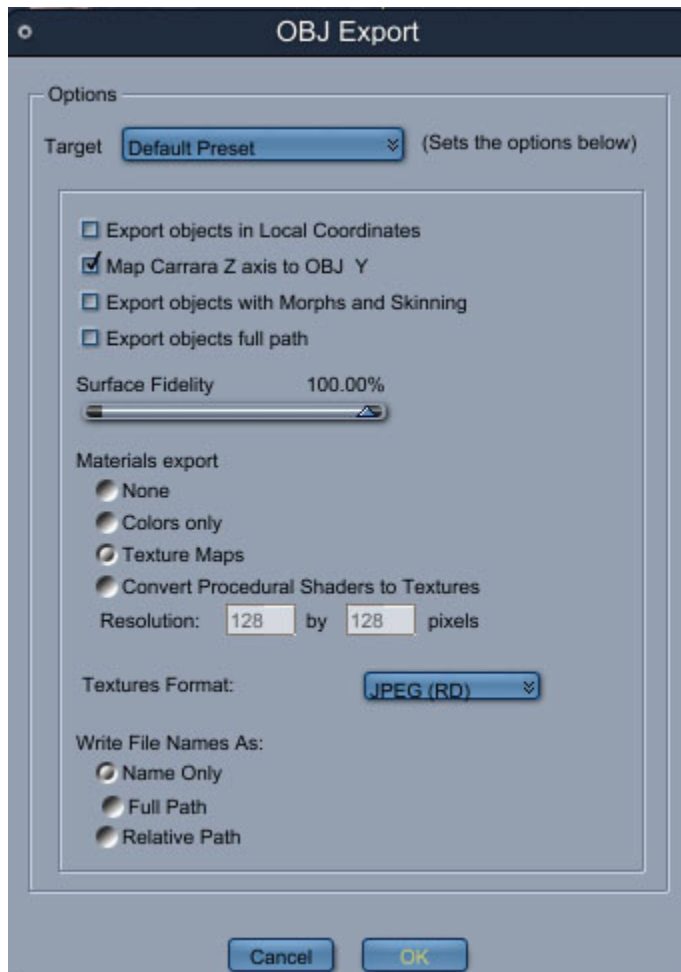
3. Check on the options controls to change the export settings.

    - **Target**: Select the target program for the export. Based on the target selected, different export options may be unavailable (grayed out).
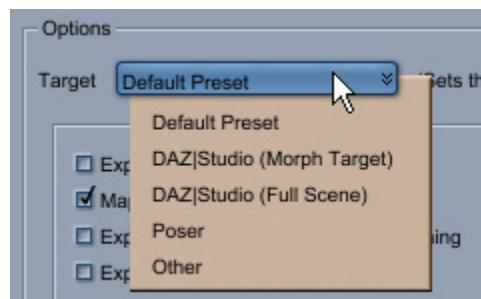
    - **Export Objects In Local Coordinates**: when unchecked (default), the Carrara objects are exported with the positions and scaling they have in the scene. When checked, the geometry of Carrara's instances is exported without taking into account their scaling and position (useful for creating Poser morph targets).

    - **Map Carrara Z axis to OBJ Y**: when checked, transforms the coordinates of Carrara objects so that their bottom (XY plane) is horizontal in the OBJ file. When unchecked, the coordinates are exported without modification.

- **Export objects with Morphs and Skinning**: TBD

- **Export objects full path**: TBD

- **Surface Fidelity**: for objects with multiple levels of detail (like Splines), sets the precision of the export. A 100% value will export highly detailed meshes, while a lower value will export fewer facets (and generate smaller OBJ files).

- **Materials export**:

  - **None**: when selected, no materials will be exported.

  - **Colors only**: when selected, only the color of the objects will be exported.

  - **Texture Maps**: when selected, texture maps and object colors will be exported.

  - **Convert Procedural Shaders to Textures**: when checked, a texture map is calculated for each procedural shader used in the Carrara scene. This option can slow down the export quite a bit, especially if you select a high resolution.

- **Textures Format**: selects the format to use when saving texture maps.
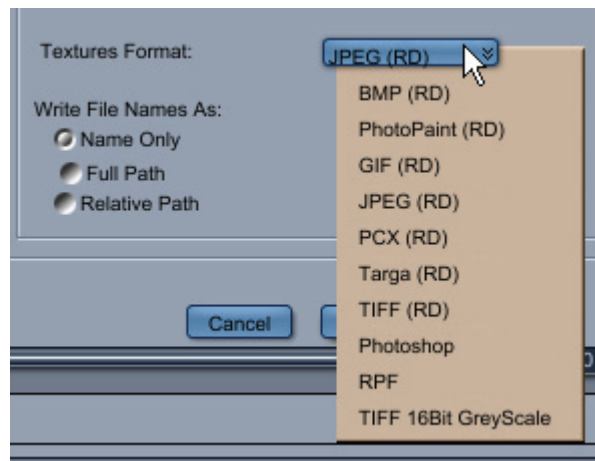
- **Write File Names As**: sets how file references are written in the OBJ and MTL files (as file name only, full path, or relative path).

4. Press **OK**.

Viewpoint Experience Technology Export (Standard, Pro)

You can use Carrara 6 to create 3D content for your web site using VET (Viewpoint Experience Technology). It's as simple as exporting your scene. Carrara converts the geometry, lighting and materials of your objects to VET file format. It can also embed code in an HTML page using special tags.
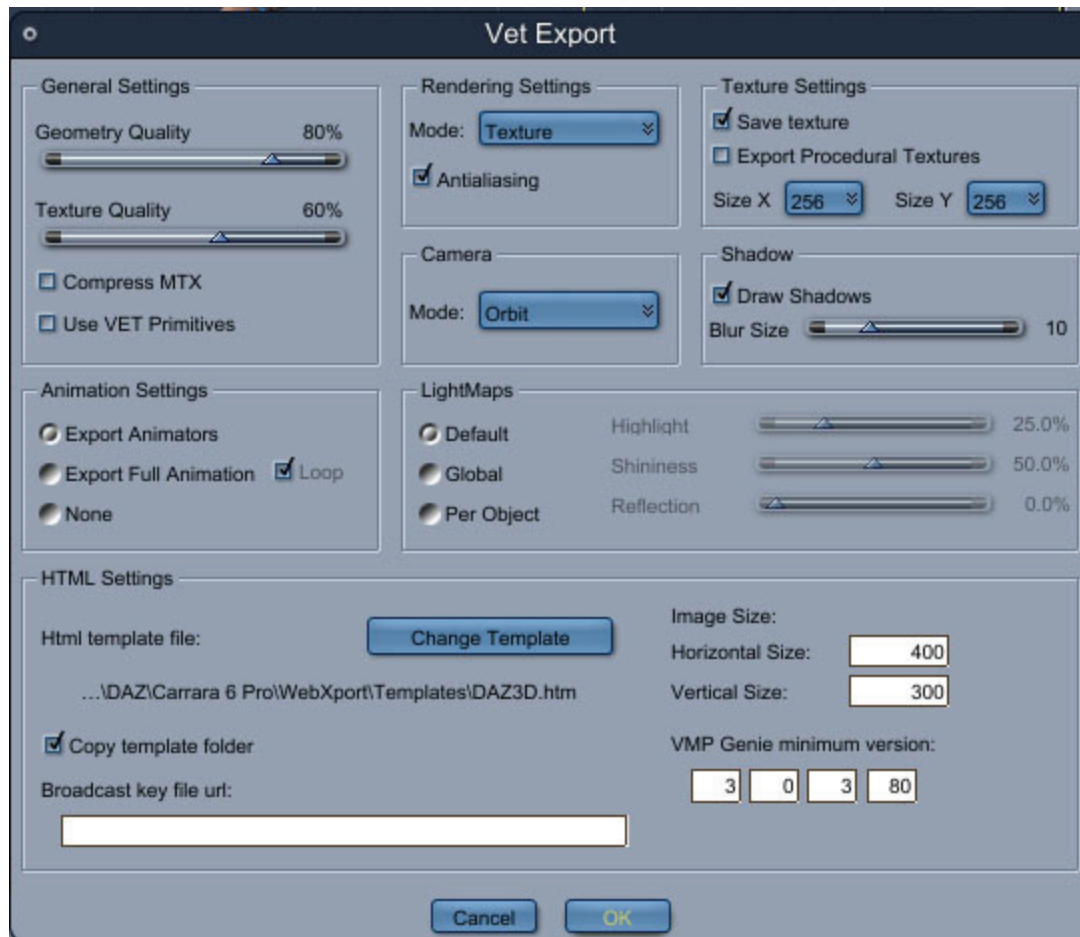
You can also add interactivity to your scene using VET-dedicated tools in Carrara.

The following topics give more detail about Viewpoint Experience Technology export.

Exporting a Scene

To export a VET scene:

1. Choose **File Menu> Save As** and select **VET** as the type of the file to export.
2. If you selected the **options** checkbox in the file dialog, an option dialog will open and you can select the options you want. Carrara will create three files:
    *   **filename.html** which is the actual web page in which the scene will be embedded
    *   **filename.mtx** which contains the XML code of your file.
    *   **filename.mts** which contains the description of the 3D objects and materials.
3. Simply double-click on the HTML file and the scene will open in your web browser.

For information on the many export options, see "Export Options."

Previewing a Scene

You can also preview your scene from inside Carrara, useful as you are building your scene to make sure things are looking like you want. The options in the VET Preview dialog are similar to those described below under Export Options.

To preview a scene:

1. Choose **Edit Menu> VET preview.**
2. Choose the export options you need in the dialog that opens. The preview will open and you can rotate around your scene with the mouse.

VET Export Options

The option dialog allows you to control how your scene will be converted to the VET format. Here is a description of each option and how to use them:

General Settings

- **Geometry quality** controls how the geometry is compressed. (100% for the best quality)
- **Texture quality** controls how the texture maps and light maps are compressed. (100% for the best quality)
- **Compress Mtx**: The MTX file is usually small but if it becomes too big, you can use this option to compress it.
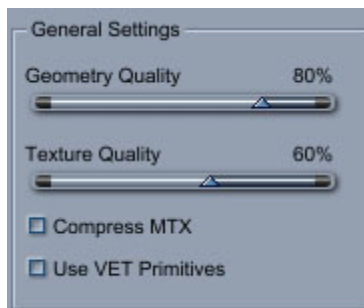
Rendering Settings

- **Rendering mode** selects the rendering modes that will be used by the VET player. Options are Wire, Point, Flat, Texture, and Sketch.
- **Antialiasing:** If checked the object edges will be antialiased.
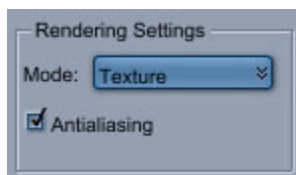
Camera Settings

The camera can be in two different modes:

- **Orbit:** The camera turns around the scene, the center of rotation is the origin of the scene. Use this mode if you want the user to be able to rotate the scene.

- **Actual:** The camera is at the exact same position as the camera in Carrara. In this mode the animation of the camera is also exported. Use this mode if you want to make a small movie-like animation.
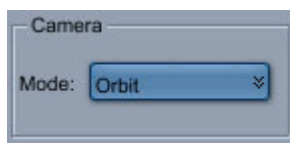
Texture Settings

- **Save texture** means that the texture maps are exported with your scene

- **Export Procedural Textures:** By default only shaders containing simple texture maps are exported. If you select this option, the procedural shaders such as marble or cellular are sampled and exported as texture maps. This conversion process can be slow so we advise that you only use it when doing the final export.

- **Size** controls the size of the texture maps used for procedural textures. A bigger size means better quality but also more memory and a slower export.
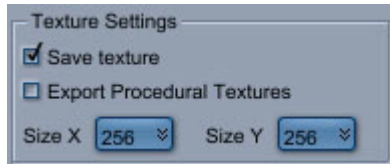
Animation Settings

- **Export Animators** will cause the animation defined with VET Animators to be exported (see "VET Animators").

- **Global Animation** will cause the animation of the scene to be exported (VET Animators will be ignored).

- **Loop** continually repeats the animation once it is finished.

- **None** means that no animation will be exported.
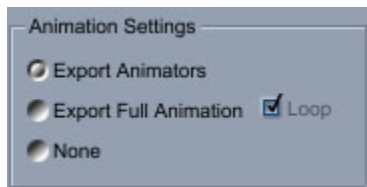
Shadow Settings

- **Draw Shadows** decides whether drop shadows are drawn.

- **Blur Size** controls the size of the blur of the shadows. A blur of 0 gives hard shadows, a bigger blur means softer shadows.

LightMaps Settings

- **Light Maps** are used to simulate the lighting and reflections on your objects. There are three different modes for Light Maps.

- Note that in all these modes you can always override the light map for a given shader by placing a 256x256 texture map in the reflection channel of this shader. (see "Light Maps" for more information on light maps).

- **Default:** A default light map is used for all the objects. You should use this mode when you just want to check the position and animation of your objects. This mode is fastest because it does not need to render any light maps.

- **Global:** A single light map is used for all the objects. You can select the properties of the lighting you want using the highlight, shininess and reflection sliders (reflection will reflect the current background of the scene)

- **Per Object:** In this mode, a light map is calculated for each shader that differs significantly from the others (only the highlight, shininess and reflection intensities are taken into account).Calculating light maps can be fairly slow so we recommend that you only use this mode once your objects are well positioned and animated.
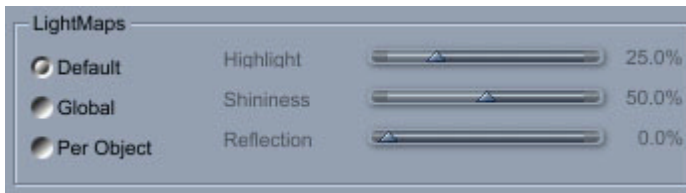
HTML Settings

- **Html Template file**: This allows you to use a different HTML template. The HTML template is the file in which your scene will be embedded.

- **Copy Template Folder**: When this option is selected, Carrara looks for a folder with the same name as the HTML template file and copies it next to the HTML file that is exported. Usually this folder contains all the images necessary for your HTML page to display properly.

- **Image Size** controls the size in pixels of the images that will be displayed in the HTML file.

- **VMP Genie minimum version** is the minimum number of the version of the Viewpoint Media Player that needs to be used to display this scene. Normally, there is no reason to change this.

- **Broadcast key file** URL: The URL of the broadcast key needed to publish Viewpoint content on your web site. (You need to contact Viewpoint to get a broadcast key).
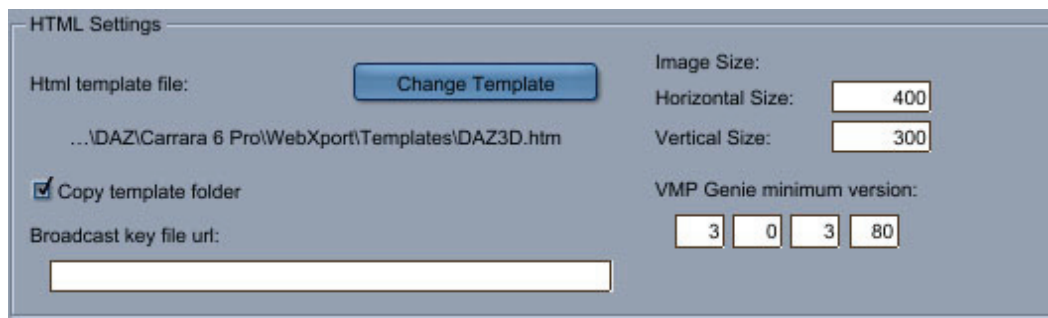
Getting Results with VET

Light Maps

In the VET technology, lighting is simulated using a texture called light maps. Carrara automatically generates light maps that match the lighting of your scene. You can also override the light maps used for a given object.

To get good results, it is necessary to understand how light maps are built. Light maps can also allow you to simulate reflections (environment mapping).

How to create good light maps:

- We recommend that you only use distant lights in your scenes because lights maps only simulate distant lights. However, Carrara will also convert your other lights into light maps but the results may not be what you expect.

- Objects that have the same shader or similar shading properties will use the same light maps. The properties that influence the light maps are the following: shininess, highlight and reflection. To save memory, avoid using too many different shaders.

- Light maps will include a reflection of the background for objects where the reflection is not set to 0.

- You will need to set a background on your scene so as to have reflections on your objects.

Setting your own light maps:

- A light map is basically a 256 by 256 texture map. If you apply a texture map to the reflection channel of a shader, this texture will be used as a light map. You can create new light maps easily with Carrara. You just need to render a sphere in 256x256 so that the edges of the sphere touch the edges of the image. Adding lights, a background, backdrops and changing the shader of the sphere will all affect the properties of the light map.

- It is very important that the texture is 256 by 256 pixels or it will not work.

Maps (Texture, Bump, Transparency)

The texture maps, bump maps and transparency maps that you may assign to shaders in Carrara will be exported. There is nevertheless a few constrains to respect in order to make them correctly.

If you want to export a transparency map, the shader must also contain a texture map of the same dimensions. If the dimensions of the transparency map are different from the texture map ones, the

transparency map will be ignored and the transparency of the material set to the first value encountered on the map.

A bump map can be exported with or without the existence of a texture map, but if there is one, its dimensions should be identical to the bump map's.

A texture map can be a traditional picture but it can also be a flash movie file (SWF files). However, flash movies cannot be displayed inside Carrara. The shader will appear to be white in Carrara but the flash movie will be visible in the preview or after exporting the VET file.

Backdrop

The backdrop of a Carrara scene will also be exported. As for the texture maps, it is possible to assign a flash movie to the backdrop. The flash movie won't be visible inside Carrara but it will be in the preview or in the exported VET file.

Animation

There are some limitations to the kind of animations that can be exported. Only key-frame based animations are properly exported. Only the animations using object transform, the color of a light map or the deformation applied to an object with the Morpher Deformer can be exported. You can move, rotate, scale objects in your scene. Any other kind of animation will not be exported correctly.

Add Interactivity to Your Scene

Carrara 6 gives you the possibility to add interactivity to your scene for the VET export. You can define interactions such as triggering an animation when clicking on an object or displaying two dimensional text when the cursor slides over an invisible hot spot. Two objects in Carrara can help you to define such interactions: the VET Animator object and the VET Interactor.

VET Animators

An animation can be defined as a collection of smaller animations, each of them independent from the others, in order to be played separately and to be triggered at different moments. The VET Animator is an object that defines a piece of an animation.

To create a new VET Animator:

1. Choose **Edit Menu>VET Animator**. A dialog displays the VET animator options.
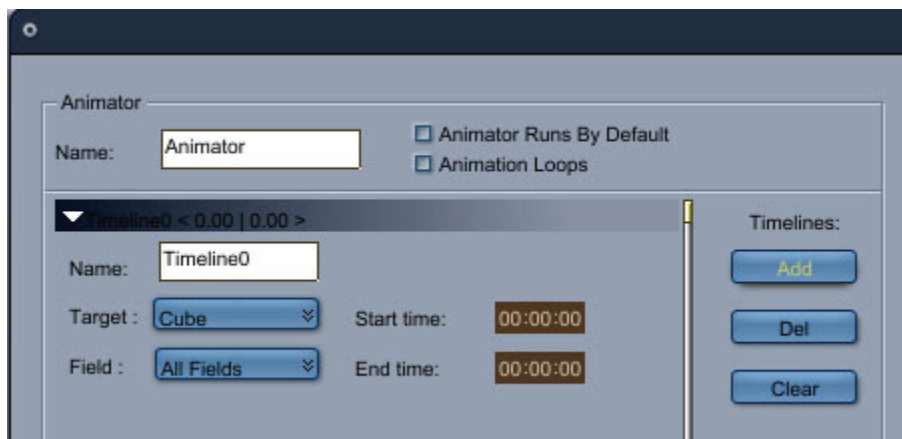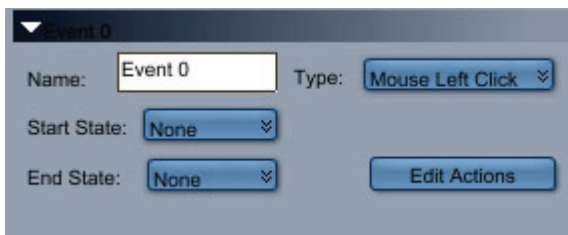
2. Set options as follows:

- **Name**: Type the name of your animator (This is the name that appears in the object hierarchy). It is important that all the objects in the scene have a unique name in order to properly export your scene as a VET scene.

- **Animator Runs by Default**: Check this option to run the animation by default (at the opening of the scene in a web browser). Uncheck the option to trigger the animation with a VET Interactor.

- **Animation Loops**: Check this option to repeat the animation continuously. Uncheck the option to stop the animation once it is finished.

- **Add**: Click this button to add one or several animations (timelines) to the list.

- **Del**: Click to remove the last animation in the list.

To edit each animation:

1. **Name** your animation in order to find it more easily later.

2. Choose the object or group of objects the animation will be applied to using the **Target** popup menu.

3. Choose the field of the target you want to animate. (All Fields means both the transform and the color of the object.)

4. Enter the start and end time in the object time line that will be used for the animation.

To modify an Animator:

1. Select the animator in the object hierarchy.

2. Click on the Edit button of the Properties tray.

NOTE! Do not forget to check the Export Animations option in the Export dialog.

VET Interactors

An Interactor defines a list of events (mouse left click, mouse over, ...) attached to an object of the scene and the actions triggered by those events.

There are three types of actions:

- Actions applied to an Animator to start, stop, rewind, ... the animations it contains.

- Actions that modify a parameter of an object, like its scaling.

- Actions that trigger VET specific actions, such as loading a MTX file or displaying 2-dimensional text.

Interactors are implemented as state machines. A state machine is a collection of states and transitions between those states. At a given time only one state is active (current state). The state machine determines a transition map of how to get from one state to another based on input (events). The scene transitions from state to state by means of actions.

New Interactor

To create a new Interactor:

1. Choose **Edit > VET Interactor**. The following dialog will come up.
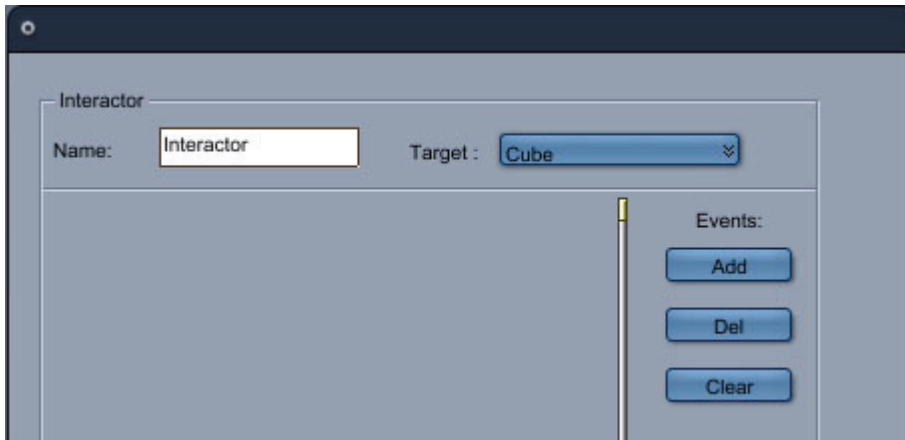
2. Type the name of the **Interactor** (this is the name that appears in the object hierarchy). It is important that all the objects in the scene have a unique name in order to properly export your scene to VET.

3. Next, choose the **Target** of the interactor. This target can be either a visible or an invisible geometry in the scene. An invisible geometry is called a **Hot Spot**.

4. Add one or several events.

New Event

To add an event:

1. Click on the Add button. (Del removes the last event of the list).

2. Set the following options as needed:

   - **Name**: Assign a name to the event.

   - **Type**: Choose an mouse event option from the popup menu. This mouse movement will trigger the event.

   - **Start State** and **End State**: Choose None to assign no state. Choose Add to assign a name for the state The Start State of an event acts as a condition on the activation of the event. An event is considered only if the Current State matches the Start State of an event when this event occurred. Otherwise the event is ignored.

   - **Edit Actions**: Click this button to add, delete, or clear the associated actions.

3. After the execution of the action, the Current State of the scene is set to the End State of the Event (if there is no End State, the Current State stays untouched). Once an event is defined, the actions associated with it can be described.

To define the actions associated with an event:

1. Click on the **Edit Actions** button.

2. Click on the **Add** button to add an action.

3. Name the action.

4. Choose the target of the action.

5. Choose the property of the target that will be affected by the action from the **Field** popup.

6. Adjust the desired value for that property.

Special Actions

These actions do not modify the state of a current object but will add something to your scene. There are four special actions:

- **Load MTX File** allows you to load an existing VET scene. You must select the path of the MTX file you want to load.

- **2D Text** is very useful to add extra information to a scene. When the user points to the target of the interactor, the user will typically see descriptive text.

- **2D Picture**: Same as 2D text but displays an image instead of text.

- **2D Flash**: Same as 2D text but displays a Flash movie instead of text.

Hot Spots

A Hot Spot is the name given to the target of an interactor. This target can be a classical geometry but it can also be an invisible or still geometry.

To make a geometry invisible, unchecked the **Visible** check box in the **Properties** of an object. The object will still exist in the VET scene and interactors will be able to use it as a target, but its geometry won't be visible.

To make a still geometry, you have to define this geometry as a child of the camera. For this, click and drag the object to the Camera in the hierarchy. The object will then be motionless to the user. Be sure to choose the Actual camera when exporting your scene to keep the same relative position of the object and the camera.

To modify an Interactor:

1. Select it in the object hierarchy.

2. Click on the **Edit** button of the Properties tray.

Current Limitations

There are several important limitations as to what can be exported. It is important to keep these limitations in mind when you are designing a scene for VET or you will not get the results you might expect.

- **Camera animation:** The camera is not animated in the preview but the animation of the camera works in the exported file (you need to select **Actual** as the export mode for the camera).

- **Animation defined by VET Animators** are not visible in the preview but they are exported correctly.

- **Animation:** Only key-frame based animation are currently supported. Animations using motion paths, inverse kinematics or physics will not export properly. Also note that animated hot points are not supported. Morph animations are not supported on the Mac platform.

- **Tweeners:** Only linear and Bézier tweeners are supported, all other tweeners will be exported as linear tweeners. Bézier tweeners are exported with their default values. Modifying the Bézier curve in order to modify the exported animation will not have any effect.

- **Texture Map Size:** To get optimal results, it is advised to use only the power of 2 (1, 2, 4, 8, 16, 32, 64, 128, 256, 512 ...) for the size of the texture maps.

- **Object Naming:** It is important that objects are properly named with unique names.

Modeling Objects

Using Primitive Tools

Carrara provides a number of different types of primitive objects that you can add to your scene. These primitives are found in the Insert menu, and also have tool icons that appear directly beneath the menu commands.

In this section you'll learn how to work with the basic geometric and particle primitives. Refer to "Using Environmental Primitives" to learn more about the primitives that simulate natural objects such as terrains, clouds, and plants.

Using Geometric Primitives

If you are creating an object from scratch, the first thing is to break down the object to its basic components. What those basic components turn out to be will determine how to build it.

If those components turn out to be all simple geometric shapes, you can create an object using the geometric primitives in the Assemble or Storyboard room.

For example, when the skyscraper shown in the following figure is reduced to its basic components, you can see that it's made up of cylinders, cones and cubes. Therefore you could use the geometric primitives in the Assemble room to create the object.

Below is a complex object on the left and the primitives used to make it on the right.
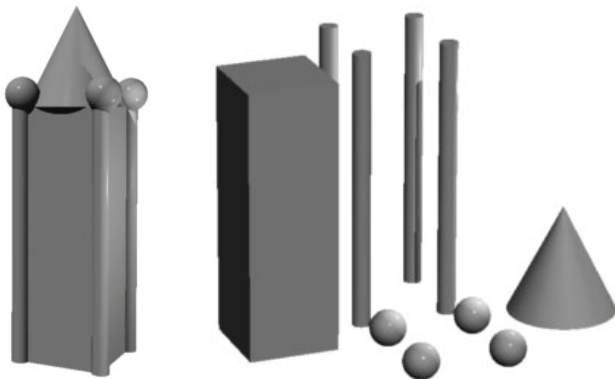
In the Assemble room, you can create five 3D geometric primitives (spheres, cubes, cones, cylinders, and icosahedra) three 2D geometric primitives (planes, infinite planes, and Splat).

When you insert a geometric primitive object, it simply appears in the scene window, where you can move and resize it as necessary.

The Geometric Primitive tools are located on a single pop-up menu. You can also use commands in the **Insert** menu to insert a primitive object.
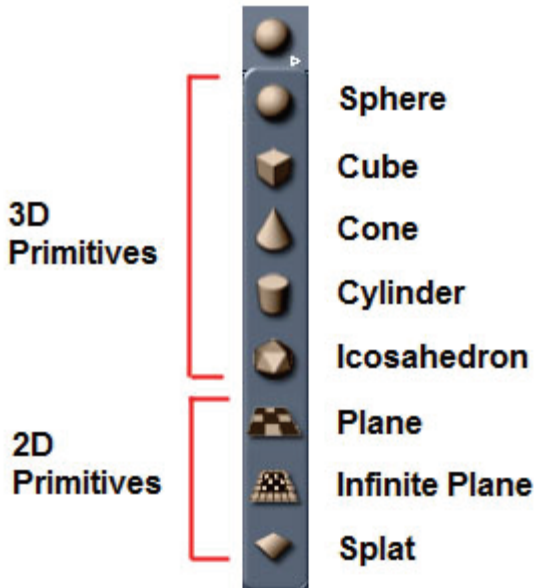
To insert a basic geometric primitive:

1. In the Assemble or Storyboard room, choose **Insert menu**, then choose the >type of object.

2. Or, choose a Geometric Primitive tool from the pop-up menu and do one of the following:

   - To create an object of default size, click in the scene window.

   - To create an object of a custom size, drag in the scene window.

3D Primitive Tools

Often, the easiest way to create an object is by combining a number of simple geometric primitives. Carrara provides a number of 3D primitive tools that you can use to build your objects and which you'll learn more about in this section.

# Sphere Primitives

To insert a sphere primitive:

In the Assemble or Storyboard room, do one of the following:

- To create a sphere of default size, choose **Insert > Sphere**, or click the **Sphere** tool and then click in the document window at the desired location.

- To create a sphere of a custom size, do one of the following:

  o Click the **Sphere** tool. Click in the document window to set the location and then drag outward to size the sphere.

o Select any Sphere primitive in your scene and enter a new size in the **Properties tray: General tab: Sphere panel**.
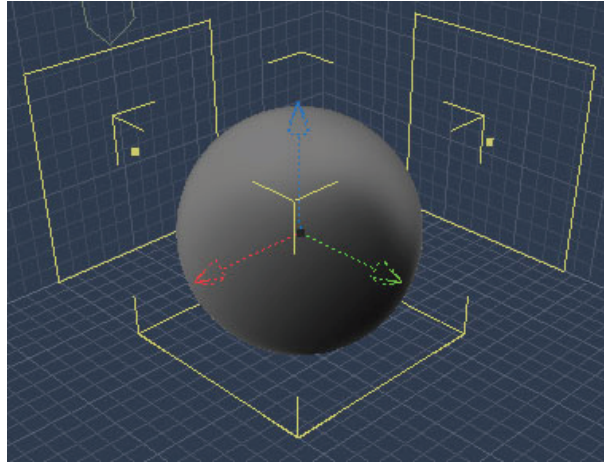
Cube Primitives

To insert a cube primitive:

In the Assemble or Storyboard room, do one of the following:

- To create a cube of default size, choose **Insert > Cube**, or click the **Cube** tool and then click in the document window at the desired location.

- **To create a cube of a custom size,** do one of the following:

  o Click the **Cube** tool. Click in the document window to set the location and then drag outward to size the cube.

  o Select any Cube primitive in your scene and enter a new size in the **Properties tray: General tab: Cube panel**.
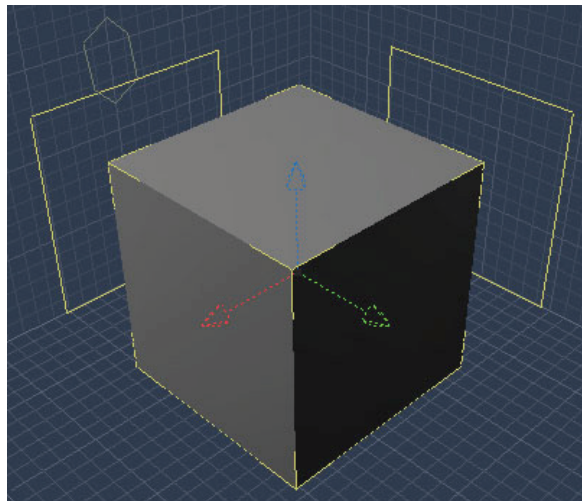
Cone Primitives

To insert a cone primitive:

In the Assemble or Storyboard room, do one of the following:

- To create a cone of default size, choose **Insert > Cone**, or click the **Cone** tool and then click in the document window at the desired location.

- To create a cone of a custom size, do one of the following:

    o Click the **Cone** tool. Click in the document window to set the location and then drag upward to size the cone.

    o Select any Cone primitive in your scene and enter a new size in the **Properties tray: General tab: Cone panel**.
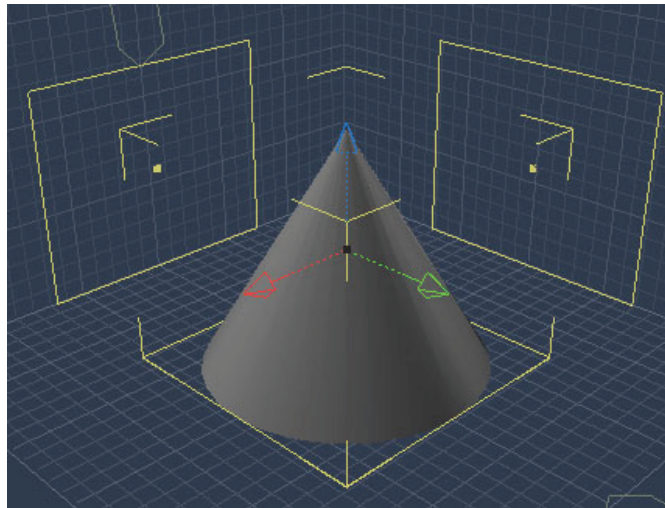
Cylinder Primitives

To insert a cylinder primitive:

In the Assemble or Storyboard room, do one of the following:

- To create a cylinder of default size, choose **Insert > Cylinder**, or click the **Cylinder** tool and then click in the document window at the desired location.

- To create a cylinder of a custom size, do one of the following:

    o Click the **Cylinder** tool. Click in the document window to set the location and then drag upward to size the cylinder.

    o Select any Cylinder primitive in your scene and enter a new size in the **Properties tray: General tab: Cylinder panel**.
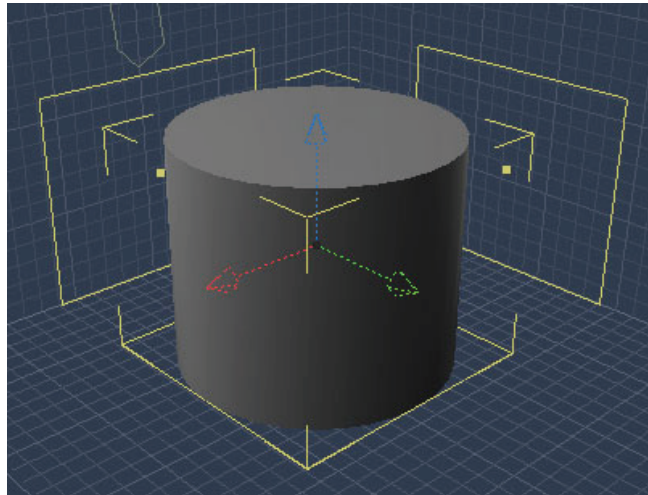
Icosahedron Primitives

To insert an icosahedron primitive:

In the Assemble or Storyboard room, do one of the following:

- To create an icosahedron of default size, choose **Insert > Icosahedron**, or click the **Icosahedron** tool and then click in the document window at the desired location.

- To create an icosahedron of a custom size, do one of the following:

  o Click the **Icosahedron** tool. Click in the document window to set the location and then drag upward to size the icosahedron.

  o Select any Icosahedron in your scene and enter a new size in the **Properties tray: General tab: Icosahedron panel**.
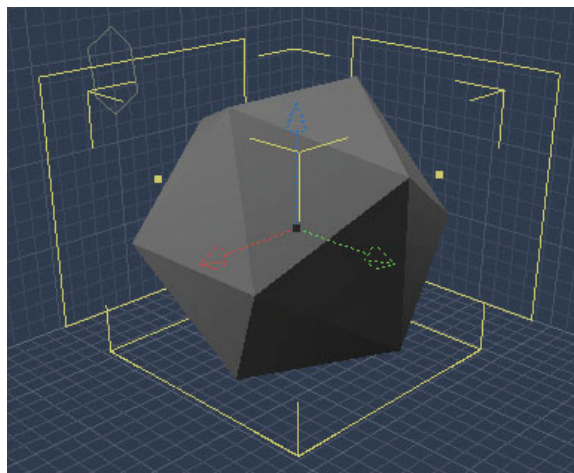
2D Primitive Tools

2D Primitive tools allow you to create various planar shapes, or two-dimensional shapes that lie across one plane. Carrara provides three types of 2D primitives, which you will learn more about in this section.

Planes

To insert a 2D plane primitive:

In the Assemble or Storyboard room, do one of the following:

- To create a plane of default size, choose **Insert > Plane**, or click the **Plane** tool and then click in the document window at the desired location.

- To create a plane of a custom size, do one of the following:

    o Click the **Plane** tool. Click in the document window to set the location and then drag upward or outward to size the plane.

    o Select any Plane primitive in your scene and enter a new size in the **Properties tray: General tab: Plane panel**.

Creating Infinite Planes

An Infinite plane is a flat geometric primitive whose sides extend out to infinity in all directions. This type of primitive can be used to create a surface for your scene such as a sky, ocean, or ground.

Carrara inserts Infinite planes parallel to the ground plane. You can move or rotate Infinite planes as you would any other object in the scene window.
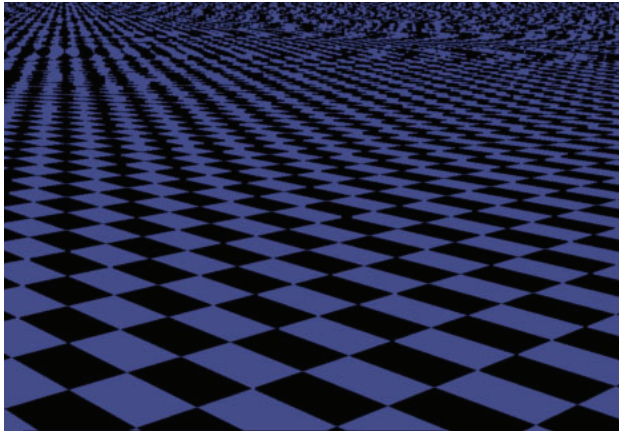
To insert an Infinite plane:

1. Choose **Insert > Infinite Plane**, or click the **Infinite Plane** tool.

2. Click in the document window. Although the plane appears finite in the scene window, it extends out to infinity when it is rendered.

Shading Infinite Planes

Applying shaders to an Infinite plane can be tricky, especially when you use texture maps or gradients. The plane's tiling controls can help you adjust how shaders are tiled on the plane.

To learn more about shaders, refer to "Shading Objects."

To adjust shader tiling:

1. Double-click the Infinite plane. You move to the Model room and the Infinite plane modeler appears in it.
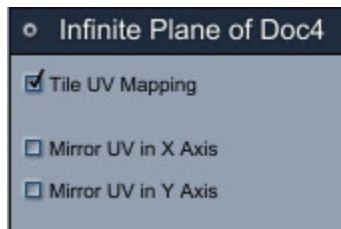
2. In the **Properties tray: General tab: Infinite Plane panel,** set the tiling options you want.

- **Tile UV Mapping: R**epeats the shader, aligning it across the plane. If Tiling is disabled, the shader is stretched to cover the entire surface of the plane.
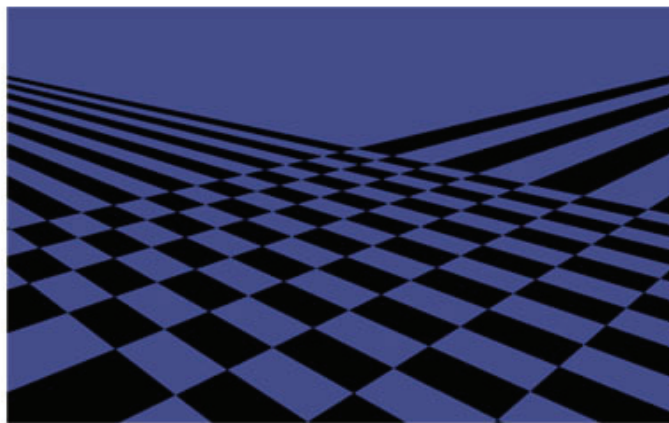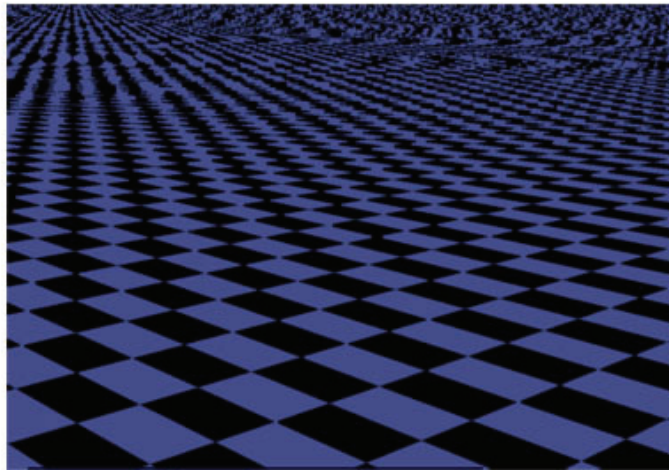
- **Mirror UV in X Axis**: TBD

- **Mirror UV in Y Axis**: TBD

3. When you are finished, close the Infinite Plane controls and go back to the Assemble or Storyboard room.

To adjust the size of the tiles:

- Adjust the size of the plane by selecting the Infinite plane and using the **Properties tray: Motion tab: Transform panel:** Size and Scaling settings. For more information, refer to "Viewing and Setting Object Properties."
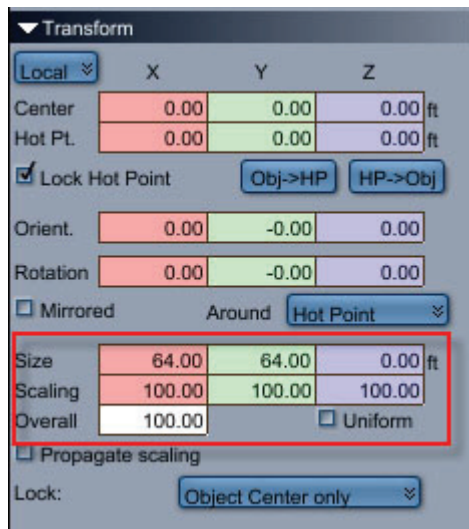
Splat Primitive

A Splat primitive is a billboard plane that always faces the camera, and is most often used to display textures that have alpha channels. For example, you can use a Splat primitive, and apply a rendered tree for a texture, using an alpha channel to remove the solid background color. Repeat the Splat object several times and you can have a forest without the geometry overhead.
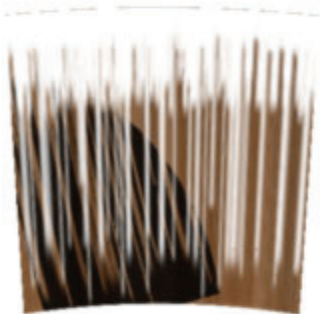
To insert a Splat primitive:

1. Do one of the following:

   - To create a Splat primitive of default size at the center of the universe, choose **Insert >Splat**.

   - To create a Splat primitive at any other location, drag the **Splat tool** to the desired position in the scene window.

2. In the **Properties tray: General tab: General panel**, click the **Edit** button to open the Shader Editor, where you can assign a shader to the Splat primitive. Alternatively, click the Preset button to assign a preset shader to the primitive.

3. After you assign the shader, use the Test Render tool to view the Splat primitive in your scene.

Creating Text Objects

Text objects that you create in the Assemble or Storyboard room can be edited for their bevel, scale, and depth, and for standard text features such as font, pitch, alignment, and spacing.

To create a Text object:

1. Open the Text modeler using one of the following methods:

2. To create a Text object of default size at the center of the universe, choose Insert **>Text**.

3. To create a Text object at any other location, drag the Text tool to the desired position in the scene window.

4. The Text modeler opens in the Model Room. In the blank section at the bottom of the window, type the text you want.

Nice and Easy

5. Select a font from the list that appears at the left side of the Text modeler. You can use any TrueType and Type 1 fonts in your system for creating Text objects. As you scroll through the fonts, the text that you entered changes to reflect the new font.

Font Style:		Alignment:		
Plain		Leading:	120.00	%
Font Size:	48.00	Word Spacing:	0.00	pts
Scaling:	100.00 %	Letter Spacing:	0.00	pts
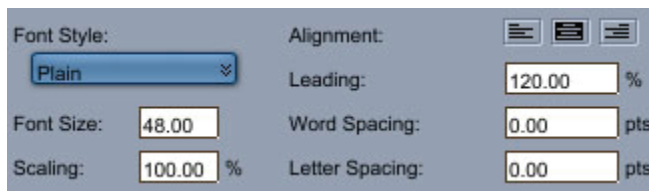
6. Use the Font controls (above the white text area) to adjust the font properties.

   - **Font Style**: Allows you to choose from the various styles associated with the text (such as Plain, Bold, Italic, or Bold Italic).

   - **Font Size**: Sets the size of the font, measured in points.

   - **Scaling**: Scales the length of the text without affecting the height (Font size) or Depth. For example, a setting of 200% doubles the width of the Text object.
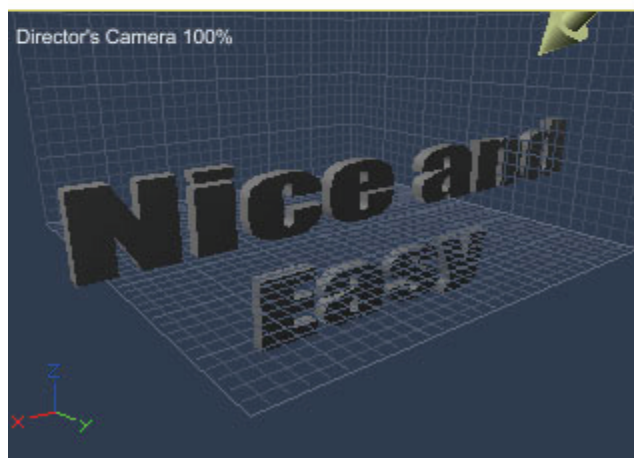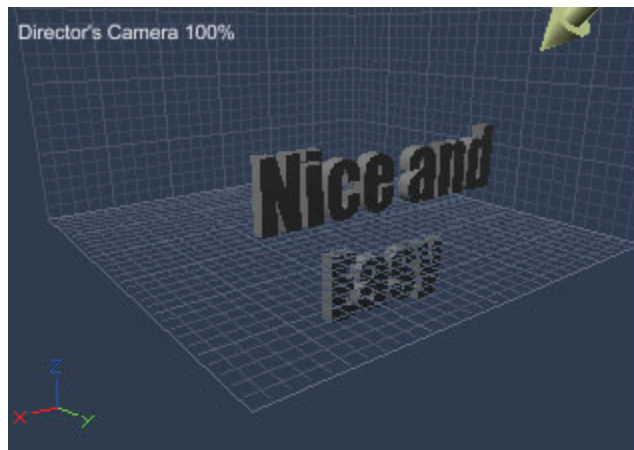
- **Alignment**: Aligns text on the left, right, or center.
- **Leading**: Sets the spacing between lines.
- **Word Spacing**: Sets the spacing between words, measured in points.
- **Letter Spacing**: Sets the spacing between letters, measured in points.

7. Use the controls in the **Bevel** section to bevel the text if desired:

- **Front Face** creates a bevel on the front face of the text.
- **Back Face** creates a bevel on the back face of the text.
- **Depth** and **Height** set the slope of the front and back bevels.
- **Type** determines the shape of the front and back bevels.
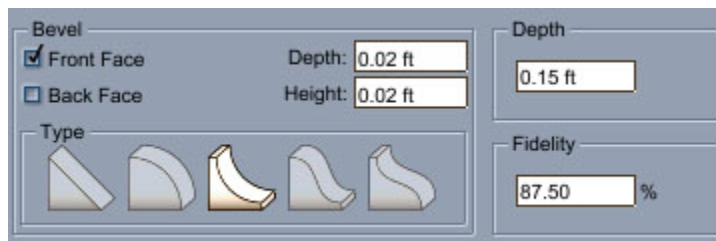
8. Use the Bevel controls to bevel your 3D text.

159

- Adjust the **Depth** setting to determine the extrusion depth of the text.

- Adjust the **Fidelity** setting to improve the appearance of the bevel if necessary.
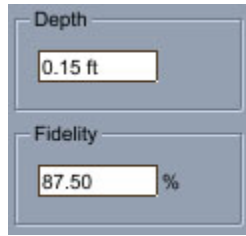
9. When you are finished, go back to the Assemble or Storyboard room, where you can view the final results. If you need to make changes, double-click the text object to return to the Text modeler.

Creating Particle Systems (Standard, Pro)

Particle systems can be used to create complex particle effects such as sparks, erupting volcanoes, and explosions.

The basic steps to creating a particle system are:

1. Insert the particle system in your scene.

2. Set basic particle system options.

3. Set options to determine the behavior of particles in the system.

4. Preview the particle system.

5. Shade the particle system.

To insert a particle system:

- Depending on the desired location of the particle system, do one of the following:

  o To create a particle system of default size at the center of the universe, choose Insert **>Particle Emitter**.

  o To create a particle system at any other location, drag the Particle Emitter tool to the desired position in the scene window.

Editing Particle Settings

Particle settings are what really determine the look and behavior of your particle system. These settings include such parameters as the lifetime, number, speed, and color of particles, whether they are emitted in a straight tail or fan out, whether they bounce on impact, and whether they "spawn" other particles.

After you create a particle system, you automatically enter the Particle modeler. The modeler consists of two tabs: Basic and Advanced.

You can also enter the Particle modeler at any time when you double-click on a particle system, or select a particle system and click the Model Room icon.

Basic Particle Settings

The Basic tab appears when you first create a particle emitter.
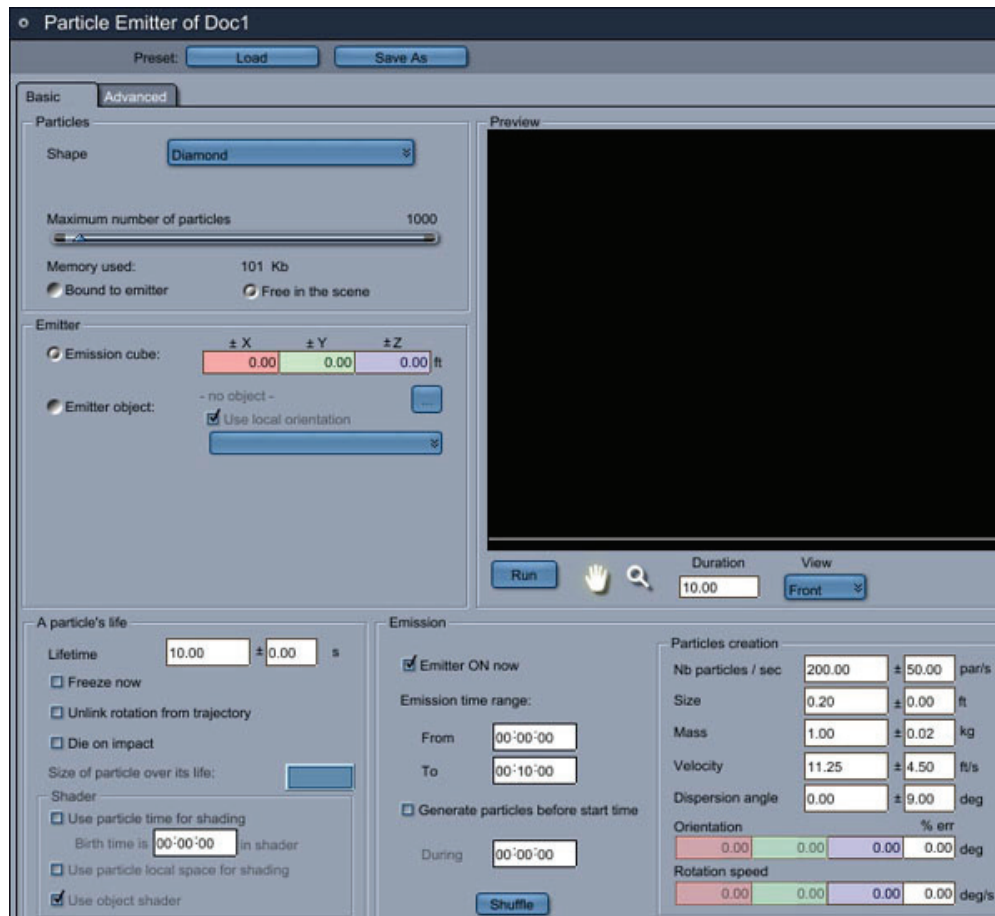
For further information on the settings in this dialog, see the following sections.

Particle Shape and Quantity

The **Particles** section determines the shape and number of particles:
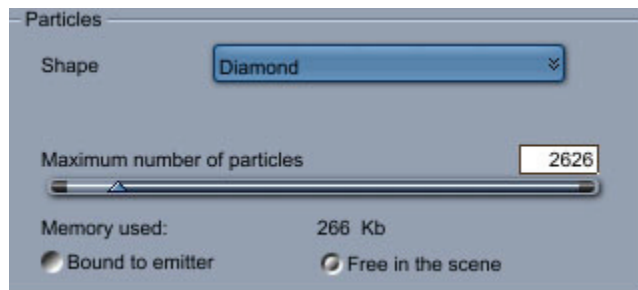
- **Shape:** Determines the geometry of the particles. Options are Triangle 1, Triangle 2, Rectangle, Diamond, Sphere, Facing camera, Metaball and Objects.

NOTE! If you choose Metaball you can set the influence of metaballs that specify the deformation. If you choose Objects you can add particle objects as sons of the particle emitter in the hierarchy.

- **Maximum number of particles**: Adjust the slider to determine the number of particles that the emitter generates. As you adjust the value, you can view the amount of memory that the particle emitter uses.

- **Bound to emitter**: Choose this option to create particles that move with the emitter. For example, with the exhaust from a rocket, the exhaust would stay behind the engine rather than flying off into space.

- **Free in the scene:** Choose this option to create particles that move independently from the emitter. For example, water spraying from a hose whipping back and forth flies out and falls where it is emitted.

Particle Placement

The **Emitter** section determines where the particles are created:
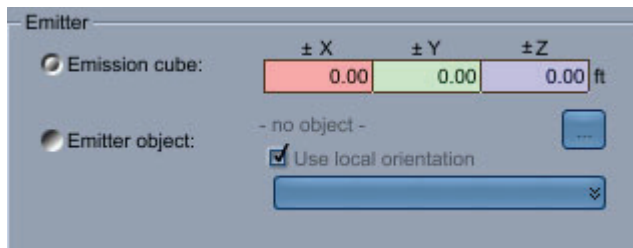
- **Emission cube**: Creates particles in a cube around the base of the emitter object. You can offset the particles from the emitter object by entering values in the +/- X, Y, and Z fields.

- **Emitter object**: Creates particles from the surface of another object. Click the file selector button to choose an object in your scene.

- **Use local orientation**: Check this option to align the particles with the surfaces of the emitter object. Uncheck this option to align the particles in the same orientation.

- If your emitter object has multiple shading domains, you can align particles with one part of the object's surface. Choose the desired shading domain from the popup menu.

Particle Life

The **A particle's life** section determines how many seconds the particles exist after they are emitted.
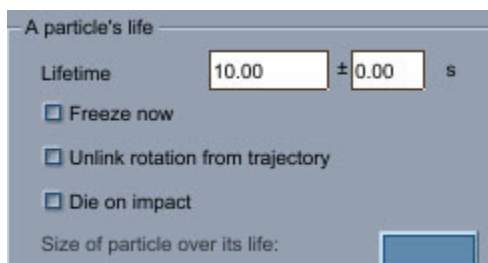
- **Lifetime:** Enter a +/- factor to make some particles live longer or shorter than others.

- **Freeze now**: Freezes a particle system at a particular point. For instance, you can freeze the system immediately to create a galaxy or frozen waterfall, let it remain frozen for a moment (changing the camera) and then unfreeze it.

- **Unlink rotation from trajectory**: By default, particles are oriented with their trajectory so that they always face their next position. Check this option for more realistic effects.

- **Die on impact**: Useful for creating a stream of particles that is limited to a certain area, since the particles will not bounce and spread after impact. Since particles are recycled, this option is useful for creating a fast-moving flow without increasing the maximum number of particles

- **Size of particle over its life**: A curve that enables your particles to get bigger over time to, for example, make air bubbles in a glass of soda

Particle Shading

Options in the **Shader** section are discussed in more detail in "Previewing the Particle System."

Particle Emission

The **Emission** section includes the following settings:
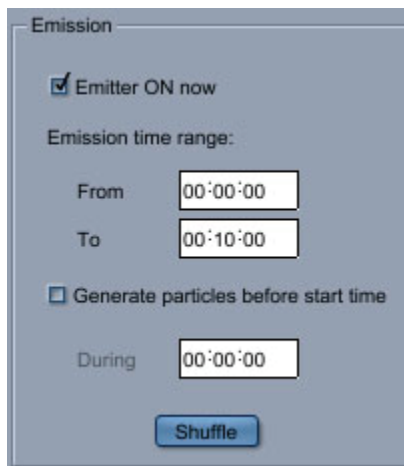
- **Emitter ON now:** Active particles are created using the parameters. When unchecked, no more particles are created, but the ones that are alive keep moving.

- **Emit time range:** A more global parameter than Emitter ON. This setting helps accelerate calculations by specifying the global period when you want to use the emitter. If you don't want to display the beginning of the creation of particles and you only want it when it is fully working, you can generate particles before start time during a certain amount of time.

- **Generate particles before start time**: TBD

- **Shuffle**: TBD

- **Particles creation** determines the different parameters of the particles at their birth. Most parameters can be altered on each particle using a +/- factor. These are initials parameters and some can change with time.

Advanced Particle Settings

The Advanced Particle Physics settings fall into two categories: Physics and Spawning.

Particle Physics
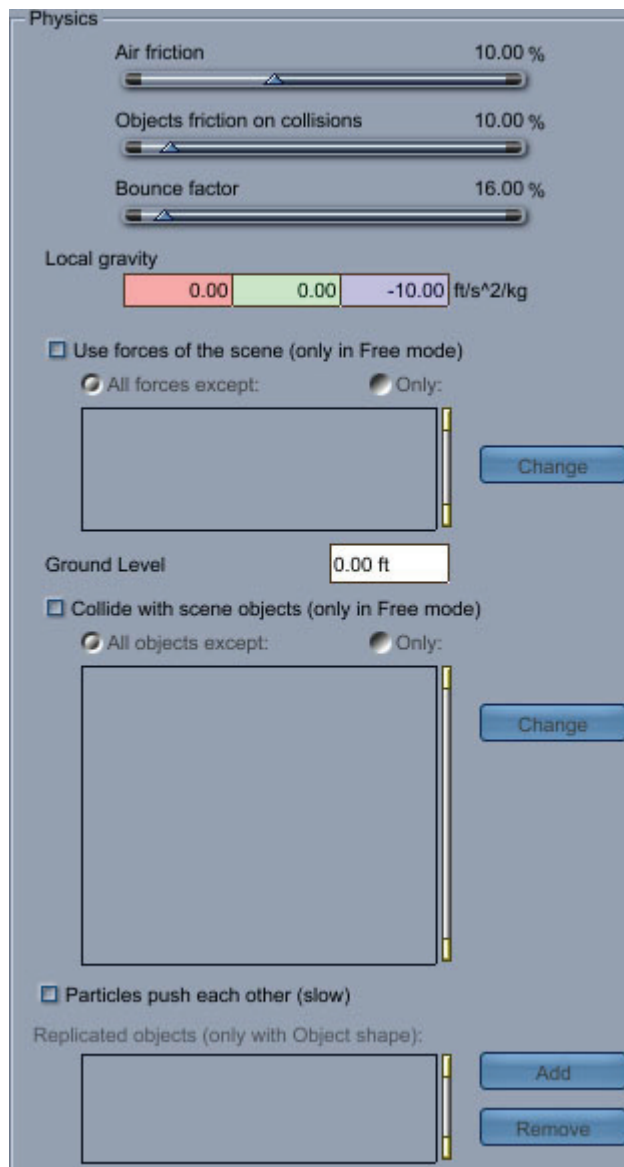
- **Air Friction:** Determines the action of air on the speed of particles. Positive numbers slow the system down, and negative numbers speed the particles.
- **Objects friction on collisions:** Determines the percentage of friction the particles encounter on collision. This setting does not affect the Bounce Factor.
- **Bounce Factor:** Determines how high the particles bounce as they hit other objects. The higher the number, the higher the particles bounce.
- **Local gravity:** Specifies the magnitude and direction of the pull of gravity on the particles.
- **Uses force of the scene (only in Free mode)**: Lets you use the forces you already use for other objects in your scene. You can choose external forces you don't want to apply to this particle system, or choose only the external forces you want to apply. When using forces, local gravity is disabled.
- **Ground Level:** Sets an imaginary plane in your scene that is considered ground level. The default is 0. Particles always collide at ground level. The ground level behaves differently

depending on whether you have Bound or Free particle systems set in the Particles section of the Basic tab:

- o **For Bound systems**, the ground level is measured along the local Z-axis of the emitter, and moves and rotates with the emitter.
- o **For Free systems**, the ground level is measured as a height along the Z-axis of the universe. If the emitter is placed or animated below ground the level, no particles are emitted.

- **Collide with scene objects (only in Free mode):** Lets you use the other objects of the scene for collisions.
  - o **All objects except**: Choose objects with which you don't want to collide, and the particles will collide with all the other
  - o **Only**: Choose only the objects with which you want the particles to collide. When using objects colliding, ground is disabled.
- **Particles push each other (slow):** Makes the particles that are too close to push each other and particles that are less close but not too far to attract each other. This option will likely slow down calculations.
- **Replicated objects (only with Object shape):** The list of the objects set as particle if shape is set to Objects. The emitter will produce particle of each type of object randomly. To add a new object use the Add button. To remove an object, delete it in the assemble room.

Particle Spawning

The Spawning settings determine whether and how impacted particles generate further particles.
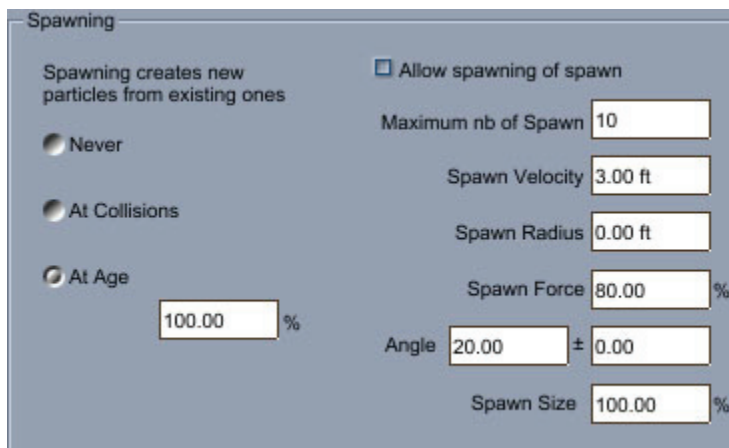
- You can choose to spawn new particles as follows:
  - o **Never**: New particles are not spawned from existing ones.
  - o **At Collisions:** Causes particles to spawn when they collide with the ground level or with collision objects.
  - o **At Age:** Causes particles to spawn when they reach a certain percentage of their lifetime. Enter a lower value in the percentage field to make them spawn sooner, or a higher value to make them spawn later.
- **Allow spawning of spawn:** Causes newly-spawned particles themselves to spawn. This option is available only when At Collisions is set. Enabling this option can quickly lead to a very large number of particles, so you may need to set the Maximum value in the Systems tab very high; this will cause performance to lag.
- **Maximum number of Spawn:** Sets the maximum for the number of new particles created when each particle spawns. The actual number will be a random value between 1 and this maximum.

- **Spawn Velocity:** Sets the minimum velocity that a particle must be traveling in order to spawn. This is useful in creating a waterfall in which the particles spawn to create splashes when they hit an object.
- **Spawn Radius:** The dispersion area for spawned particles, much like the Emit area.
- **Spawn Force:** Sets the speed at which the spawns travel, as a percentage of the original particle's speed. Set this value low to make the spawns be left behind by the original particle. Set this value higher to make the spawns shoot ahead of the original particle.
- **Angle:** Determines the arc in which the spawns are emitted. This setting works in the same manner as the Emit Dispersion setting for the entire system (on the Emitter tab), except that it applies only to spawned particles. The +/- field determines how much variation there is in the emit angle.
- **Spawn Size:** Determines the size of the newly created particles in relation to the original one.

Previewing the Particle System

You can see the effects of your edits to the particle settings in the Preview pane of the Particles modeler. The preview options you set do not affect how your particle system is rendered.
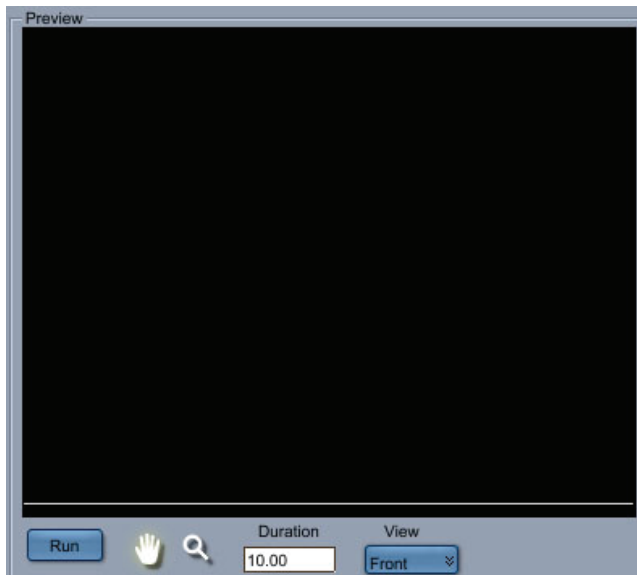
To preview the particle system:

1. In the Particles dialog, enter a **Duration** value to set how long the preview lasts in the Preview window.

2. Use the **View** selector to choose whether you want to view the particles from the **Front**, **Right**, or **Top**.

3. Click the **Run** button beneath the Preview window.

4. Use the **Pan** and **Zoom** buttons to pan and zoom your view.

Shading the Particle System

You can use any kind of shader to shade a particle system. The Shader section in the Basic tab of the Particle modeler contains settings that help you define how the particles are shaded.
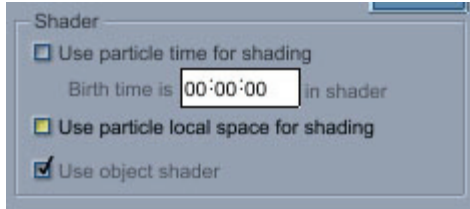
![NOTE icon] **NOTE!** To shade a particle system, you need to see some particles. Therefore, you need to advance to a frame other than 0. If you don't want to animate your shader, return to non-animated mode and clear the keyframes.

Using the Particle Shader

The particle shader is a way of shading all the particles at once but with differences depending on the particles properties. The value of the shader depends on a property of the particle that is shaded. The available settings are:

- **Use particle time for shading**: When a particle is created, the shader that it uses is the current shader at time 0. Check this setting to cause particles to be shaded at a different time than the current shader.

- **Birth time is (MM:SS:FF) in shader**: Enter the desired amount of time after which you want the shader to take effect. Time is entered in minutes, seconds, and frames. For example, if you want a 2 second delay, enter 00:02:00. After 2 seconds of its life, the particles will be shaded with current shader.

- **Use particle local space for shading**:

- **Use object shader**: Check this option to use the shader that is applied to the particle emitter.

Working with Particles in 3D View

Working with a particle system can be very time consuming, especially when there are lots of particles to calculate. In order to work faster you can use the 3D view particles settings that don't modify the final rendering but lighten the system in 3D view.

To view these settings, select the particle emitter in the Assemble room. You'll find the following settings in the **Properties tray: General tab: Particle Emitter panel**.
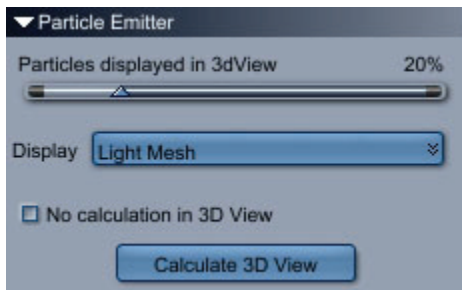
- **Particles displayed in 3D View:** Reduces the particles displayed, and reduces the particles calculated. Note that this also reduces the number of particles that appear in final rendering.

- **Display**: Displays the particles in the mode of your choice: None, Points, Crosses, Light Mesh, and Mesh.

- **No calculation in 3D View**: When you are done with your particle system and you want to work on another part of your scene, you can stop the calculation of particles in 3D view. When **no calculation in 3D View** is checked the system appears as it was calculated for the last time.

- **Calculate 3D View**: Click this button to recalculate from the start time.

Using the Blur Object Effect on Particles

The Blur Object effect lets you add a blur to particle emitters. Use the Blur Object effect to create smoke for a rocket, or a dusty trail behind a stagecoach.

To use Blur Object:

1. Drag a Particle System into the scene.

2. In the **Properties tray: Effects tab: Blur panel**, click the **Edit** button. The Blur Object dialog appears.

3. Check the **Enable Blur** option to enable the blur object feature.

4. Drag the **Radius** slider to the right to strengthen the intensity of the blur.

5. Choose a **Dropoff** function (Squared, Linear, Gaussian, None) to determine the how the blur fades out from the center of the particle.

6. Choose a **Quality** option (Low, Medium, or High) to control the appearance of the blur effect.

7. Use the Preview renderer to preview the changes you make. You can enable the Auto Update option to update the preview as you make changes.

8. Click **OK** to accept the settings and close the dialog.

Vertex Modeling Overview

The Vertex modeler lets you sculpt 3D objects by directly manipulating their surfaces but also create morph targets for performing advanced animation, fine tune bones weights or set UV coordinates for texture mapping.

Vertex modeling is the process of creating 3D objects by directly manipulating vertices, edges, and polygons. You can "sculpt" objects by changing the number and location of their vertices, edges, and polygons.

You often start with a predefined object, such as a sphere or cube, or you can use the Vertex modeling tools to generate complex shapes. Once you've created an object, you can reshape and refine it by editing the points that define its surface, edit morph targets, UVs and weights.

Enhancements in Carrara 6 allow you to edit imported content, and create new morph targets and full body morphs for DAZ or Poser content. You'll also notice improved performance with larger meshes when you move, scale, or rotate points.

The Vertex modeler lets you sculpt 3D objects by directly manipulating their surfaces. Vertex modeling is the process of creating 3D objects by directly manipulating vertices, edges, and polygons. You can "sculpt" objects by changing the number and location of their vertices, edges, and polygons.

You often start with a predefined object, such as a sphere or cube. Once you've created an object, you can reshape and refine it by editing the points that define its surface.

Vertex Modeling Concepts

A vertex object is any object created in the Vertex modeler, or created with another modeler and opened in the Vertex modeler.

The power of vertex modeling lies in the ability to change an object by directly positioning its vertices and edges.

Vertices, Edges, Polylines, and Polygons

Three-dimensional objects can be described as sets of vertices, edges, and polygons:

- A **vertex** defines a position in 3D space.
- In the Vertex modeler, all vertices are connected by straight lines. An **edge** is the line that connects two vertices. For example, eight vertices and twelve edges describe a cube. Each face of the cube is a polygon.
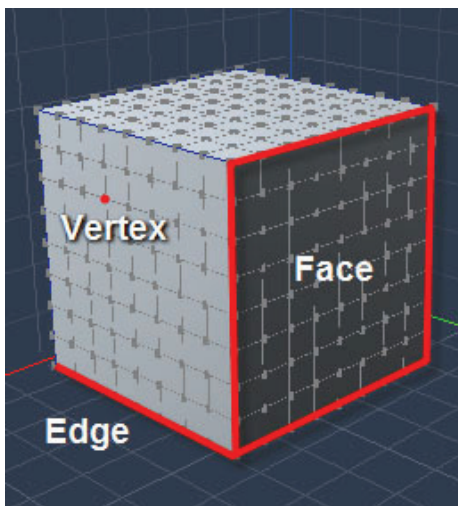
A vertex object's surface is defined by its vertices.

- A **polyline** is a set of connected edges.

- A polyline that forms a closed loop is a **closed polyline**. A closed polyline is not equivalent to a polygon:but if you fill a closed polyline, it becomes a polygon.

- A collection of vertices, edges, and the polygons that they form is called a **polymesh**. A polymesh can form a closed volume, such as a cube, or an open object such as an oval.

Although it is possible to sculpt a complex shape in the Vertex modeler from scratch by adding and editing polygons and vertices, the Vertex modeler provides you with a very rich set of advanced modeling tools such as extrusion, sweep, loft, lathe, chamfer, Boolean operations, thickness, offset surface, Coons Surface (Pro version only), Gordon Surface (Pro version only) or Double Sweep (Pro version only).

Although it is possible to sculpt a complex shape in the vertex modeler from scratch by adding and editing polygons and vertices, the vertex modeler provides you with a very rich set of advanced modeling tools such as extrusion, sweep, loft, lathe, chamfer, thickness, or offset surface.

When an object is modeled using a small number of polygons, its silhouette is often very rough and it is difficult to create smooth surfaces. Subdivision surface helps to create smooth objects without adding too many polygons. You will see in real time your the mesh as you edit a few control points.

The Vertex Modeling Window

The Vertex modeler is where you create vertex objects. You enter the Vertex Modeling Window when you choose the **Insert > Vertex Object** command from the Assemble or Storyboard Room, or when you double-click a vertex object that is already in your scene. Like the Assemble window, the Vertex modeling window provides up to four different views of your object.
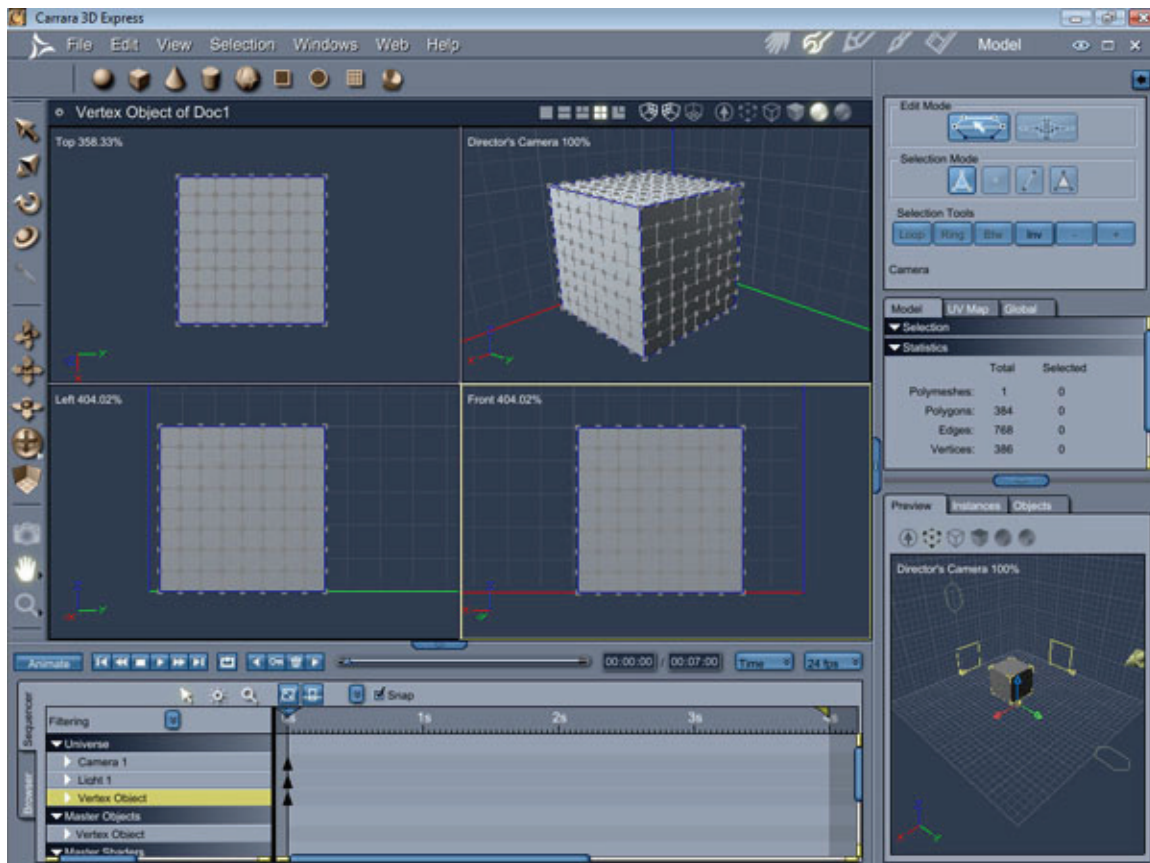
When you edit a vertex object, new menu commands are available, different tool sets appear, and the Properties tray describes your object. Your view switches to a close-up view of the object. The preview Scene window opens at the lower right of the Modeling window. This shows how your object will look in the Assemble room.

Opening the Vertex Modeling Window

You open the Vertex Modeling window by doing one of the following:

- Double-click the vertex object in the Hierarchy or the scene window. The view will switch to the Model room.

- Click on the **Model** button. The view will switch to the Model room.

- Select a vertex object, then click on the **Model** button.

- Select a vertex object while in the Assemble room and click on the **Edit Vertex Object** tool. The vertex modeling tools will activate (edit mode) without requiring you to enter the Model room.
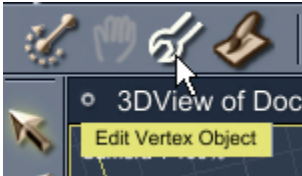
- For a new vertex object, in the Assemble room, click on the **Vertex Object** tool, then in the viewport. The view will switch to edit mode.
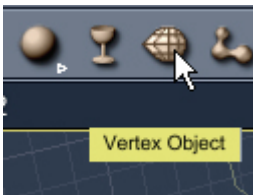
When you're finished modeling, you can save your changes and enter a different room by clicking the appropriate room's button.

NOTE! To avoid the memory-consuming accumulation of windows, always close the modeling window when you are finished modeling.

The Vertex Modeler 3D View

The Vertex modeler's 3D view shows a working box that contains the features for creating shapes and paths. Three colored lines indicate the intersections of the planes of the working box.
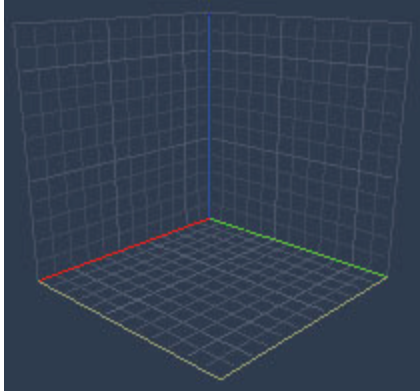
By default, the working box is centered on the origin of the coordinate system. The bottom of the working box sits on the ground plane of the universe, so the working box's vertical center is slightly elevated from the universe's ground plane. Each side of the box measures 1 foot, with a grid line at each inch. For information on changing these dimensions, refer to "Changing the Grid."

You can move and rotate the working box and its planes independently of the object you are modeling. Any adjustments you make to the orientation and location of the working box take place in relation to the universe. Thus, if you move the entire working box 10 inches up along the Z axis, you are effectively moving it up by 10 inches in your scene. The object itself, however, remains in place both in the 3D view and in the scene.

The Drawing Plane

The Drawing plane is the plane you're currently working on, whether you are drawing a shape, adjusting a sweep path, or viewing your object. By default, the Drawing plane lies on the X and Y axes of the working box. It is highlighted as a brighter border

NOTE! For information on changing and moving the Drawing plane, refer to "Changing Your Point of View."
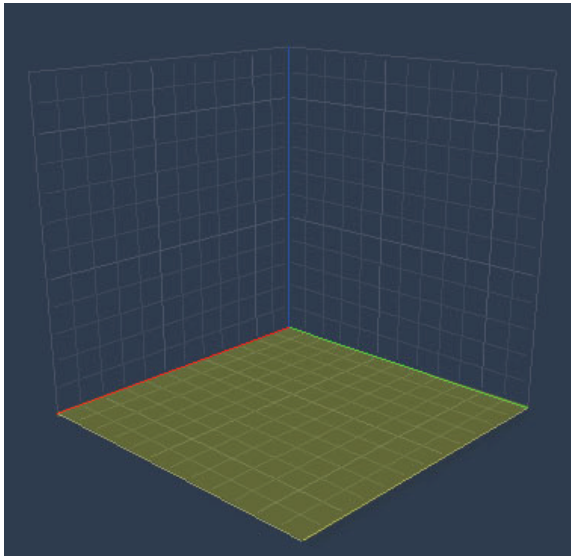
Setting Preferences for the Vertex Modeler

In the **Vertex Modeler Preferences** dialog, you can set preferences for the default welding tolerance, edge and rotation angles, the maximum number of selections you can save, and the size of vertices in the 3D view.
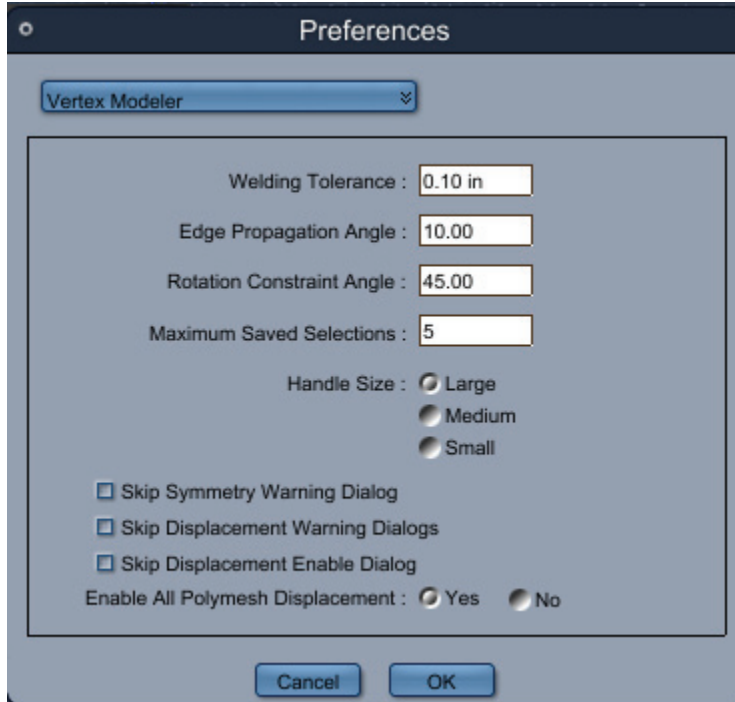
To set preferences for the Vertex Modeling window:

1. Choose **File > Preferences**.

2. Choose **Vertex modeler** from the popup menu.

3. Set the following options as desired:

   - **Welding Tolerance**: Vertices within this tolerance will be welded together.

   - **Edge Propagation Angle**: Use this option to specify the angle determining whether a second edge will be selected when you double-click on the first one edge.

   - **Rotation Constrain Angle**: Sets the angle increment used during a constrained rotation (to constrain a rotation, hold down the Shift key while dragging).

   - **Maximum Saved Selections**: Sets the limit of the number of selections than can be saved using the menu **Selection> Add To Saved Selections**. When the number of saved selections bypasses this limit, the oldest one is lost

   - **Handle Size**: Sets the size of the vertices in the 3D view.

   - **Pro Only**

     - **Skip Dialogs**: Select which warning dialogs to skip when working with displacement.

     - **Enable All Polymesh Displacement**: Sets whether to enable displacement for all polymeshes or not.

4. Click **OK** to apply the settings.

Working with the Modeling Window

Like the Assemble room, the Vertex modeler can simultaneously display up to four separate panes. Each pane gives you a different view of your object.

You can work in any view you want. For instance, you may find it convenient to work in a 2D view when drawing a sweep or extrusion path, then switch to the 3D view when you want to work in 3D. To work in a different view, simply click in the pane.

NOTE! You cannot view objects through user-created cameras in the modeling room.

To make your work flow easier, you can change different features of the Vertex modeler:

- Change the display and behavior of the grid. For more information, refer to "Setting Preferences for the Vertex Modeler."

- Change the active plane.

- Change the Drawing plane.

- Move the working box.

- Show and hide the working box's bounding planes.

Showing and Hiding Planes

By default, the planes that define the boundaries of the working box are visible. You can show them using the Grid Controls.

For information on using the Grid Controls, refer to "Changing the Grid."

Changing the Active Plane

The active plane is the reference plane that Carrara uses when creating points or objects in all tools.

There are three different ways to change the active plane depending on your preferred modeling technique:

- **User defined plane**: the active plane can be changed by clicking the plane icon in the Active Plane tool.

- **Automatic plane mode**: the active plane is automatically defined depending on the view orientation. The plane which appears to be the most perpendicular to the view direction is selected automatically.

- **All planes mode**: the active plane is the closest plane the cursor is in.

Changing the Drawing Plane

To make your work easier, you can change the Drawing plane to different axes and move it to a selection or a specific set of coordinates. You can also drag it to any position you like.

The easiest and quickest way to move the drawing plane and the working box to a specific position is to **Command/Ctrl**+click the vertex, edge or polygon where to move it.

To move the Drawing plane to a selection:

1. Select the vertex, polygon, or group of vertices with which you want to align the Drawing plane.

2. Be sure the Selection tool is chosen.

3. **Command/Ctrl+click** your mouse.

   - If the selection is a single vertex, Carrara centers the Drawing plane on that vertex without changing the plane's orientation.

   - If the selection is a group of vertices or a polygon, Carrara moves the Drawing plane to the plane shared by those vertices. If the vertices lie in different planes, the Drawing plane is moved so that all the selected points are as near the Drawing plane as possible.

   - To reset the working box position to its original position, **Command/Ctrl+click** an empty area of the 3D View or choose **View> Reset Working Box**.

To send the Drawing plane to specific coordinates:

1. Choose **View > Send Drawing Plane to**. The **Send Plane to Position dialog** appears

2. Specify the universe's X, Y, and Z coordinates to which you want to move the Drawing plane.

3. Enable a plane option:

   - **Drawing plane** Moves the Drawing plane on the axis perpendicular to itself by the amount you specify. For instance, you could move a Drawing plane lying on the X and Y axes by the specified amount on the Z axis **only**.

   - **Drawing plane centered** Moves the Drawing plane on the axis perpendicular to itself by the amount you specify, centering it on the remaining coordinates. For instance, you could move a Drawing plane lying on the X and Y axes by a specified amount on the Z axis, centering it on the X and Y axes.

   - **All planes** Moves the origin of the working box to the specified coordinates, centering all planes. Note that this option moves the working box itself.

4. Click **OK**.

Changing Your Point of View

Without changing your point of view to examine an object from all sides, it can be difficult to determine its position relative to other objects in the model.

When you perform a operation such as subtraction, this is particularly important:you must be sure that the objects are actually overlapping for it to work.

You can dolly, zoom, bank, and track the Director's Camera using the camera controls, and you can pan and zoom the view in 2D using the pan and zoom controls.

Use the camera controls to dolly, pan, bank and track the view through the Director's Camera. For information on using these controls, refer to "Camera Navigation."

Use the pan and zoom controls to perform 2D panning and zooming. For information on using these controls, refer to "2D View Tools."

You can also use the camera list in the upper left corner of the Vertex modeling window to switch among the Director's Camera (3D view) and isometric views of the working box's sides.
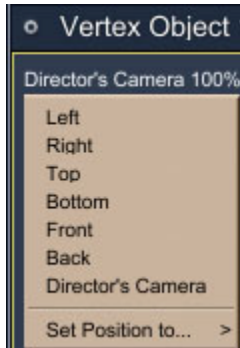
Use the camera list to switch between the Director's Camera and isometric views of the working box sides. For information on using these controls, refer to "Selecting a Camera."

After you have changed your view, the working box may appear off-center in its pane. By sending your view to the working box, you can restore the view to the origin of the coordinate system.

Previewing Objects

You can preview objects in the Vertex Modeling window at different levels of quality.

For more information on the Preview modes, refer to "Choosing a Preview Mode."

In a scene, a vertex model can have several instances, with each instance having a different shader. When the preview quality is set to Textured in the Vertex modeler, you can choose the instance to be used for the shader.

The list of instances of the object is displayed in the **Properties tray: Global tab: Shaders For Object panel**.
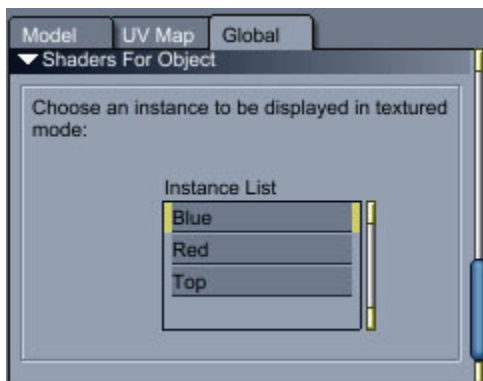
This shader is also used when a Test Render is done in the Vertex modeler.

Vertex Modeling Basics

Using the Vertex Primitives

The Vertex Primitive tools let you create several basic 2D or 3D polymesh shapes. When inserted from the tool bar, the dimensions of the object are set during the insertion by dragging, other attributes are then set with the + or - keys and validated by clicking **Enter** or **Return**.

NOTE! Vertex Primitive basic objects are inserted the same way primitives are inserted in the assemble room. For more information about how to insert an object, see "Using Primitive Tools."

To insert a vertex object from the toolbar:

1. Open or create a document in the Assemble room. (For more information, refer to "Creating an Empty Scene.")

2. Depending on the desired location of the vertex object, do one of the following:

   - To insert a vertex object of default size at the center of the universe, choose **Insert > Vertex Object** or drag the Vertex Object tool into the Hierarchy section of the Sequencer.

   - To insert a vertex object in any other location, drag the **Vertex Object** tool into the scene window to any location.

3. Choose a vertex primitive from the pop up.
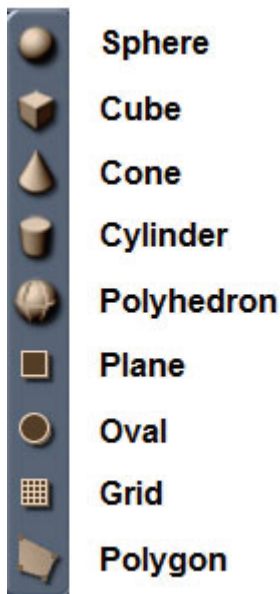
4. In the Modeling window, drag one corner of the bounding box to the opposite corner. Release the mouse button when the object is the desired size.

5. Use + or - to set the mesh density of the primitive.

6. Press **Enter** to validate the inserted primitive.

Inserting 3D Vertex Objects (Standard, Pro)

When inserted from the **Construct > Insert 3D** menu, each primitive has its own properties dialog. The dialog lets you define the primitives attributes before it is inserted in the modeler.

When inserted from the toolbar, each primitive still has its own properties dialog but you cannot define the primitive attributes before it is inserted in the modeler, just set the size by dragging in the viewport.

The 3D Vertex Primitives are covered in the following sections.

3D Spheres

The 3D sphere vertex object can be used for spheres and ellipsoids. While creating a sphere you enter diameter, height, and the number of sides and sections. You can also orient the sphere to align with the X, Y, or Z axis.

The following image shows a sphere with a diameter of 1 foot, height of 2 feet, 8 sides, and 8 sections. Object axis orients the object to the X plane.
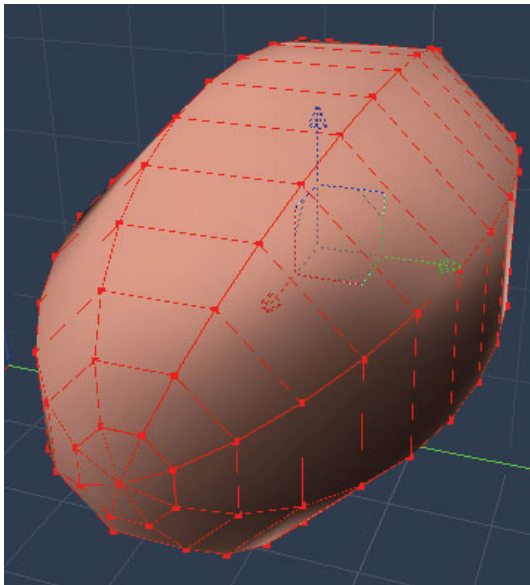
To insert a Sphere from the Construct menu:

1. Choose **Construct > Insert 3D > Sphere.**

2. The Insert Object dialog appears. Enter the following properties:

   - **Diameter**: The diameter of the sphere.

   - **Height**: The height of the sphere.

   - **Number of Sides**: The number of divisions along the diameter.

   - **Number of Sections**: This is half the number of divisions you want along the height of the sphere (or, the number of divisions in one hemisphere).

   - **Object Axis**: Click the X, Y, or Z arrow to choose the axis that you want the object to point to. The height of the object will be oriented along this axis.
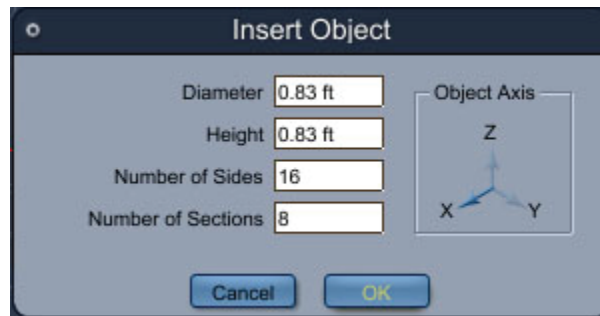
3. Click **OK**. Your sphere appears in the scene.

3D Cubes

The 3D cube vertex object can be used for cubes and other box shapes. While creating a cube or box you enter the X, Y, and Z dimensions and the number of segments that appear along each axis.

The image below shows a cube with X size of 1, Y size of 2, Z size of 3, and segmentation of 4.
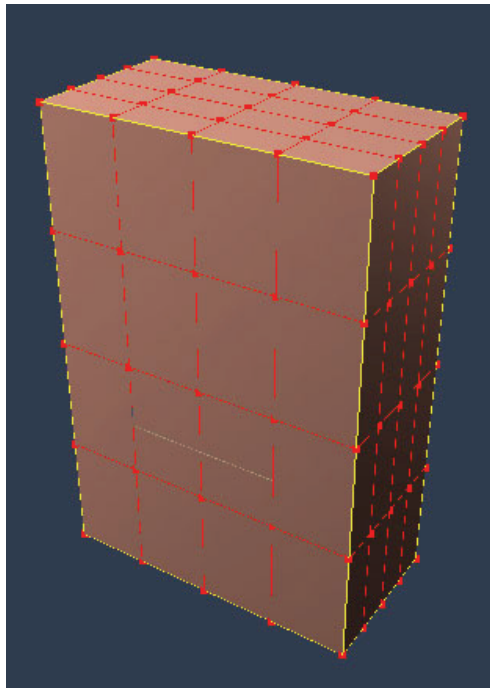
To insert a Cube from the Construct Menu:

1. Choose **Construct > Insert 3D > Cube.**

2. The Insert Cube dialog appears. Enter the following properties:

   - **X size**: Determines the depth of the cube.

   - **Y size**: Determines the width of the cube.

   - **Z size**: Determines the height of the cube.

- **Segmentation**: Determines the number of divisions along each axis of the cube.
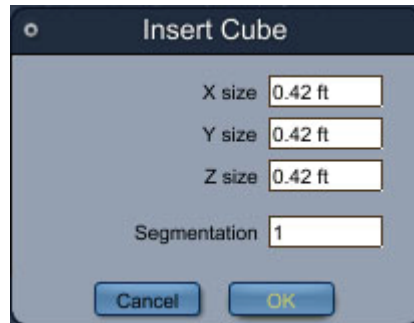
3. Click **OK**. Your cube appears in the scene.

3D Cones

The 3D cone vertex object can be used for cones, as well as 3 or 4-sided pyramids. While creating a cone you enter diameter, height, and the number of sides and sections. You can also orient the sphere to align with the X, Y, or Z axis.

The image below shows a cone with a diameter of .2, a height of .4, 8 sides, and 10 sections. Object axis orients the object to the X plane.
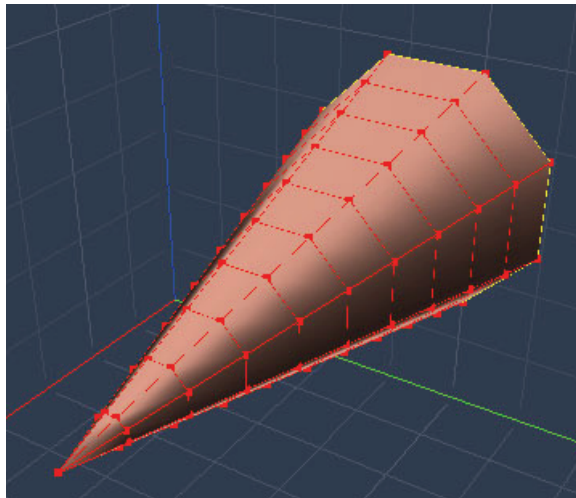
To insert a Cone from the Construct Menu:

1. Choose **Construct > Insert 3D > Cone.**

2. The Insert Object dialog appears. Enter the following properties:

   - **Diameter**: The diameter of the cone at its base.

   - **Height**: The height of the cube from base to top.

   - **Number of Sides**: The number of sides at the base of the cone. For pyramid shapes, set this value to 3 or 4.

   - **Number of Sections**: The number of sections from the base to the top of the cone.

   - **Object Axis**: Click the X, Y, or Z arrow to choose the axis that you want the object to point to. The height of the object will be oriented along this axis.
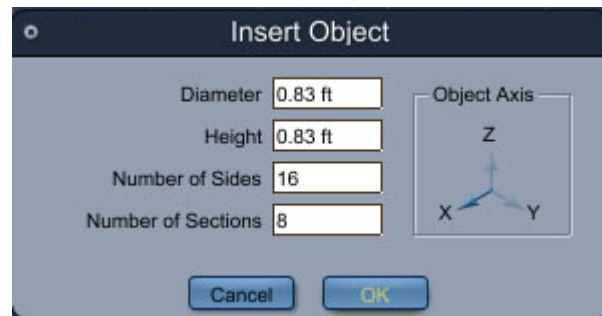
3. Click **OK**. Your cone appears in the scene.

3D Cylinders

The 3D cylinder vertex object can be used for cylindrical shapes. While creating a cylinder you enter diameter, height, and the number of sides and sections. You can also orient the sphere to align with the X, Y, or Z axis.

The image below shows a cylinder with diameter of .1, height of .2, 8 sides, and 16 sections. Object axis orients the object to the X plane.
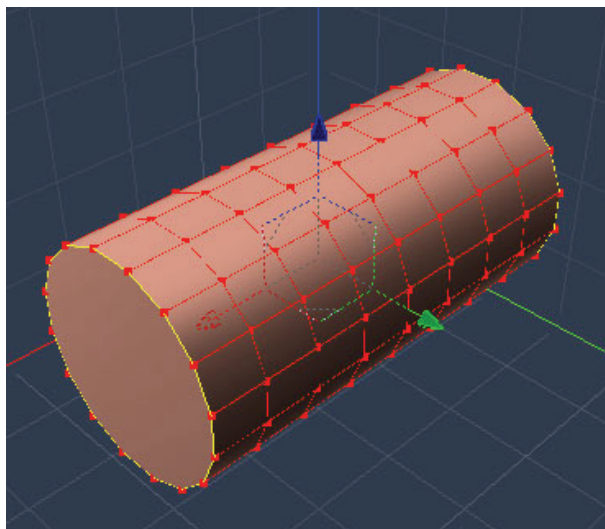
To insert a Cylinder from the Construct Menu:

1. Choose **Construct > Insert 3D > Cylinder.**

2. The Insert Object dialog appears. Enter the following properties:

    - **Diameter**: The diameter of the cylinder at its base.

    - **Height**: The height of the cylinder.

    - **Number of Sides**: The number of divisions along the base of the cylinder.

    - **Number of Sections**: The number of divisions along the height of the cylinder.

    - **Object Axis**: Click the X, Y, or Z arrow to choose the axis that you want the object to point to. The height of the object will be oriented along this axis.
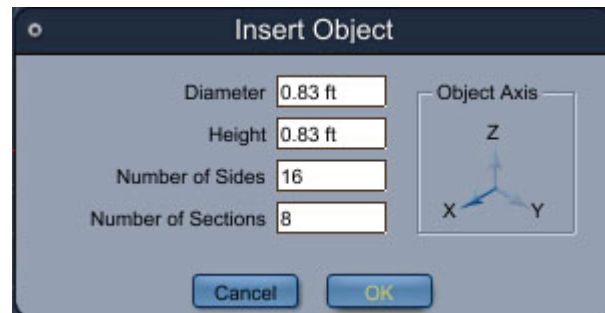
3. Click **OK**. Your cylinder appears in the scene.

Polyhedrons

Polyhedrons are 3D objects with a specified number of triangular sides. Based on the number of sides that you enter, Carrara uses that value to create a three-dimensional object. The minimum number of sides required to make a 3D object are four sides (which creates a 3-sided pyramid shape). The maximum acceptable number of sides is 999999.

If it is not possible to create a polyhedron with exactly the number of sides that you enter, Carrara adjusts the value as necessary to create a complete three-dimensional object. For example, if you enter the number 5, Carrara increases the number of sides to 6 to create a cube. The image below shows a polyhedron with 36 sides.
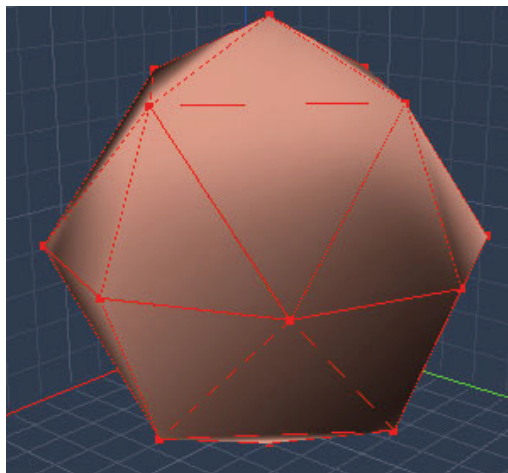
To insert a Polyhedron from the Construct Menu:

1. Choose **Construct > Insert 3D > Polyhedron.**

2. The Insert Polyhedra dialog appears. Enter the following properties:

    - **Number of Polygons**: Enter the desired number of polygons for the polyhedron. Acceptable numbers are 4 to 999999.

• **Diameter**: Enter the desired overall diameter of the polyhedron.
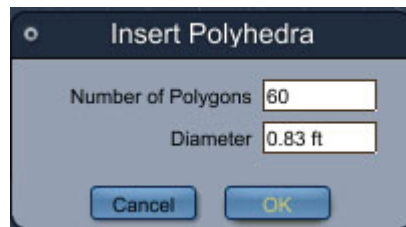
3. Click **OK**. Your 3D polyhedron appears in the scene.

3D Grid

The Grid primitive creates a flat plane along the axis that you specify.

The image below shows a grid with U Size of 6, V Size of 8, and 8 vertices for each side.
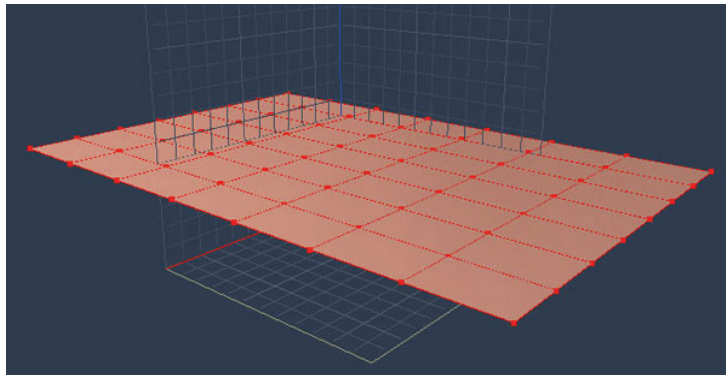
To insert a Grid from the Construct Menu:

1. Choose **Construct > Insert 3D > Grid.**

2. The Insert Grid dialog appears. Enter the following properties:

   • **U Size**: Determines the first dimension of the grid. If oriented to the Z axis, this dimension would pertain to the side of the grid that runs along the X axis.

   • **V Size**: Determines the second dimension of the grid. If oriented to the Z axis, this dimension would pertain to the side of the grid that runs along the Y axis.

   • **U Vertex Count**: Determines the number of divisions along the U Size measurement.

   • **V Vertex Count**: Determines the number of divisions along the V Size measurement.

   • **Perpendicular**: Click the X, Y, or Z arrow to choose the axis that you want the grid to be perpendicular to. Choose the X axis to create a floor, or the Y or Z axis to create a wall.
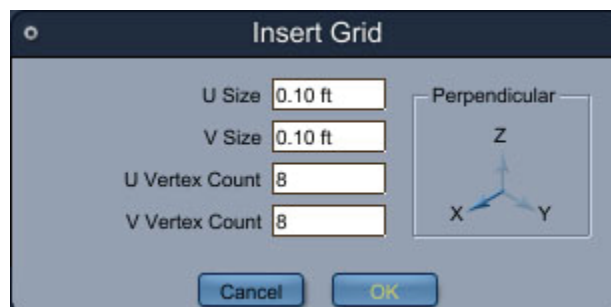
3. Click **OK**. Your grid appears in the scene.

3D Rectangle

While similar to the Grid, a Rectangle creates a flat plane with one division along each side. The rectangle lies perpendicular to the axis you choose.

The image below shows a rectangle with U Size of .2, V Size of .4, and Perpendicular to the Z axis.
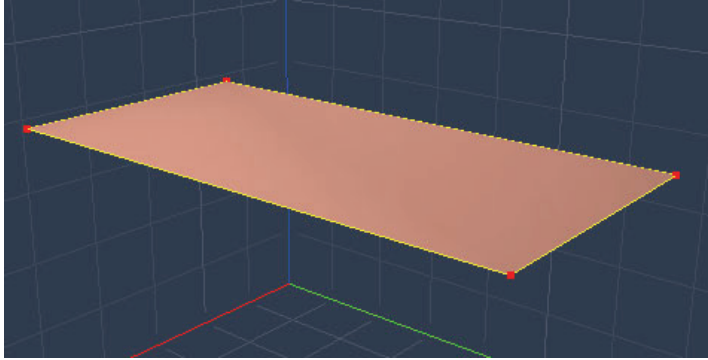
To insert a Rectangle from the Construct Menu:

1. Choose **Construct > Insert 3D > Rectangle.**

2. The Insert Rectangle dialog appears. Enter the following properties:

   - **U Size**: Determines the first dimension of the rectangle. If oriented to the Z axis, this dimension would pertain to the side that runs along the X axis.

   - **V Size**: Determines the second dimension of the grid. If oriented to the Z axis, this dimension would pertain to the side that runs along the Y axis.

   - **Perpendicular**: Click the X, Y, or Z arrow to choose the axis that you want the rectangle to lie perpendicular to. Choose the X axis to create a floor, or to the Y or Z axis to create a wall.
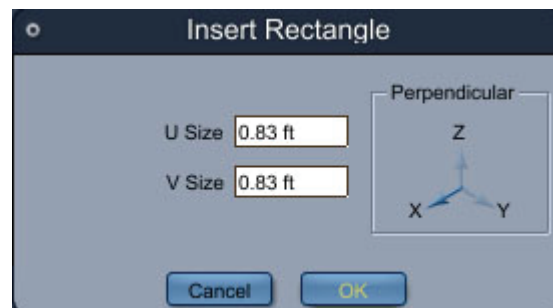
3. Click **OK**. Your rectangle appears in the scene.

3D Oval

The Oval primitive creates flat circles or ovals with a specified number of sides. The oval can lie perpendicular to the axis of your choice.

The image below shows an oval with a U Size of .2 and a V Size of .4, aligned to the Z axis.
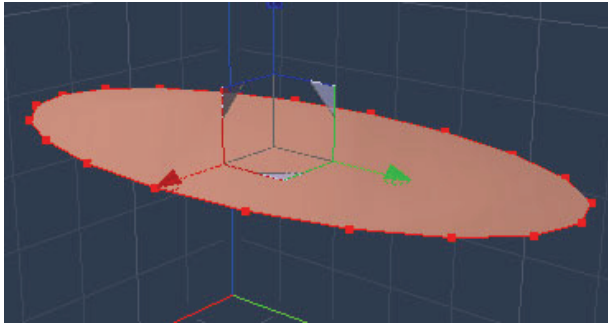
To insert an Oval from the Construct Menu:

1. Choose **Construct > Insert 3D > Oval.**

2. The Insert Object dialog appears. Enter the following properties:

    - **U Size**: Determines the first dimension of the oval. If oriented to the Z axis, this dimension would pertain to the side that runs along the X axis.

    - **V Size**: Determines the second dimension of the grid. If oriented to the Z axis, this dimension would pertain to the side that runs along the Y axis.

    - **Number of Sides**: The number of sides in the oval.

    - **Perpendicular**: Click the X, Y, or Z arrow to choose the axis that you want the oval to be perpendicular to. For example, choose the Z axis to lie the oval on the floor.
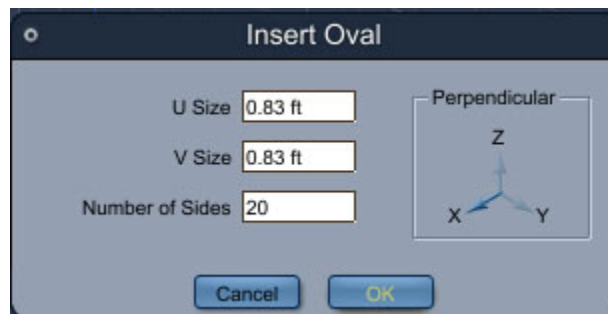
3. Click **OK**. Your oval appears in the scene.

3D Polygon

The Polygon primitive allows you to create a flat shape with any number of sides, and which lies parallel to the active plane.

A polygon is similar to a 2D polyline (see "Drawing Polylines") except that the shape is filled with a solid surface. You create polygons with the Polygon tool, which works similar to the Polyline tool.

You can also use the Polygon tool to add a new polygon to an existing polymesh, providing that the point you select for the new polygon belongs to the polymesh and is not common to more than three polygons

To insert a Polygon from the Construct Menu:

1. Choose **Construct > Insert 3D > Polygon.**

2. You enter **Edit Mode** in the Vertex modeler. Click to set each point that makes up the polygon.

3. After you set the last point in the polygon, press the **Enter** key to complete the process.
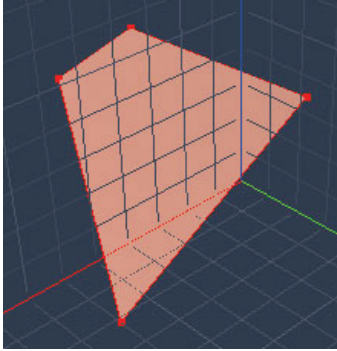
Inserting 2D Vertex Objects (Standard, Pro)

The Vertex modeler also features a number of different 2D primitives that you can use to create your models. The available 2D primitives are Polyline, Interpolated Curve, Curve, Rectangle, and Circle.
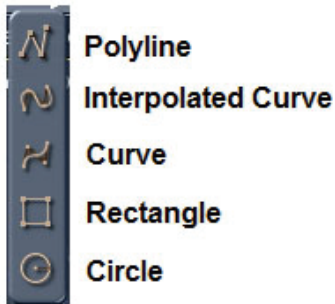
You can insert a 2D primitive using the tools, or by using the **Construct > Insert 2D** commands in the Vertex modeler.

Drawing Polylines

A Polyline is a set of vertices and edges that are joined but do not branch. An open polyline includes one or more edges that do not meet, while a closed polyline begins and ends at the same vertex. Examples are the outline of a square or a triangle. Use the Polyline tool to create polylines.

You can draw open and closed polylines, which can be used as cross-sections, paths, and lathe profiles.

To draw a polyline:

1. From the Assemble room, choose **Insert > Vertex Object** to enter the Model room. The Vertex modeler opens with the working box displayed.

2. Use the Working Box control to select the active plane. The polyline will be created parallel to the active plane.

3. From the Vertex modeler, use one of the following methods to select the tool:

- Click the **Polyline** tool.

- Choose **Construct > Insert 2D > Polyline**.

4. Set the desired options in the Properties tray:

   - **Weld with Snapped Points:** When you add a vertex on top of an already existing vertex, the two vertices are merged into one. Uncheck this option to prevent this from happening.

   - **Snap to Grid**: Check this option to automatically snap to the grid in the modeling window.

   - **Close:** Check this option to automatically create a closed polyline. Uncheck the option to create an open polyline.

5. Click in the modeling window to create each vertex in the curve:

   - **Vertices** are added to the current drawing plane except if you click on an existing point not lying in this plane. In this case, the subsequent points are created in a plane parallel to the active plane and passing through the existing point. This is very convenient for starting polylines attached to existing points of curves or polygons.

   - **Edges** are automatically drawn to connect the vertices.

6. If necessary, close the curve by clicking on the first point. You will not have to close the polyline manually if you have selected the **Close** option in the Properties tray.

7. Press **Enter** to complete the shape.

Converting Closed Polylines to Polygons

You can fill a closed polyline to turn it into a polygon.

To fill a polyline or polygon:

1. Select the polyline you want to fill.

2. Choose **Model > Fill Polygon**. The open polyline becomes a filled polygon.

Drawing Interpolated Curves

A Curve is a smooth representation of a polyline. It is created like a polyline but each point acts like a magnet for the curve. **Interpolated Curves** pass through control points.

To draw an interpolated curve:

1. From the Assemble room, choose **Insert > Vertex Object** to enter the Model room. The Vertex modeler opens with the working box displayed.

2. Use the Working Box control to select the active plane. The polyline will be created parallel to the active plane.

3. From the Vertex modeler, use one of the following methods to select the tool:

    • Click the **Interpolated Curve** tool.

    • Choose **Construct > Insert 2D > Interpolated Curves**.

4. Set the desired options in the Properties tray:

    • **Range:** Adjust the setting to the desired number of intermediate steps between each vertex that you add. Lower values use less resources, higher values create smoother curves.

    • **Close:** Check if you want to automatically create a closed curve. Leave unchecked to create an open curve, or one that you will close manually.

5. Click in the modeling window to create each vertex in the curve.

    • Vertices are added to the current drawing plane except if you click on an existing point not lying in this plane. In this case, the subsequent points are created in a plane parallel to the active plane and passing through the existing point. This is very convenient for starting curves attached to existing points of curves or polygons.

    • Additional vertices appear between the points that you click, equal to the Range value that you set in Step 4.

6. If necessary, close the curve by clicking on the first point (or by selecting the **Close** option in the Properties tray).

7. Press **Enter** to complete the shape.

Drawing Curves

The Curves tool creates smooth polyline curves that do not pass through control points.

To draw a curve:

1. From the Assemble room, choose **Insert > Vertex Object** to enter the Model room. The Vertex modeler opens with the working box displayed.

2. Use the Working Box control to select the active plane. The polyline will be created parallel to the active plane.

3. From the Vertex modeler, use one of the following methods to select the tool:

   - Click the **Curve** tool.

   - Choose **Construct > Insert 2D** > **Curve**.

4. Set the desired options in the Properties tray:

   - **Range:** Adjust the setting to the desired number of intermediate steps between each vertex that you add. Lower values use less resources, higher values create smoother curves.

   - **Close:** Check if you want to automatically create a closed curve. Leave unchecked to create an open curve, or one that you will close manually.

5. Click in the modeling window to create each vertex in the curve.

   - Vertices are added to the current drawing plane except if you click on an existing point not lying in this plane. In this case, the subsequent points are created in a plane parallel to the active plane and passing through the existing point. This is very convenient for starting curves attached to existing points of curves or polygons.

   - Additional vertices appear between the points that you click, equal to the Range value that you set in Step 4.

6. If necessary, close the curve by clicking on the first point (or by selecting the **Close** option in the Properties tray).

7. Press **Enter** to complete the shape.

Drawing Rectangles

The Rectangle tool allows you to create rectangular and square shapes.

1. From the Assemble room, choose **Insert > Vertex Object** to enter the Model room. The Vertex modeler opens with the working box displayed.

2. Use the Working Box control to select the active plane. The polyline will be created parallel to the active plane.

3. From the Vertex modeler, use one of the following methods to select the tool:

    - Click the **Rectangle** tool.

    - Choose **Construct > Insert 2D** > **Rectangle**.

4. Rectangle options appear in the Properties tray. Enter the width and height of the rectangle and press Enter. The rectangle will be created in the scene.
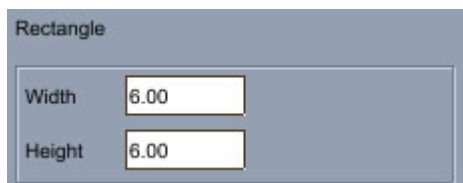
Drawing Circles

The Circle tool provides three different methods to create a circle. These methods are described below.

1. From the Assemble room, choose **Insert > Vertex Object** to enter the Model room. The Vertex modeler opens with the working box displayed.

2. Use the Working Box control to select the active plane. The polyline will be created parallel to the active plane.

3. From the Vertex modeler, use one of the following methods to select the tool:

    1. Click the **Circle** tool.

    2. Choose **Construct > Insert 2D > Circle**.

4. Circle options will show up in the Properties tray. Use the **Number points** slider or field to enter the desired number of sides.
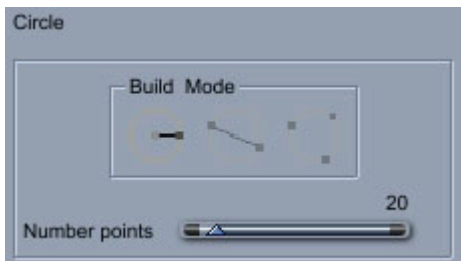
5. Choose one of the **Build Mode** options (**From center**, **From diameter**, or **From three points**), and continue to create your circle as described in the following manner.

To create a "From center" circle:

    1. After you choose the Circle tool as described previously, the Build Mode options appear in the Properties tray.

    2. Select the **From Center** build mode.
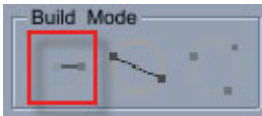
    3. Adjust the **Number points** value to set the number of vertices in the circle.

    4. Click in the Scene window to define the center point.

    5. Drag the mouse outward from the center to set the radius.

    6. Click to fix the radius. The circle is complete.

To create a "From diameter" circle:

    1. After you choose the Circle tool as described previously, the Build Mode options appear in the Properties tray.

    2. Select the **From Diameter** build mode from the Properties tray.
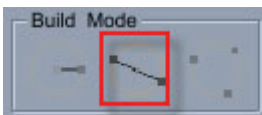
    3. Adjust the **Number points** value to set the number of vertices in the circle.

4.   Click in the Scene window to define the first point.

5.   Drag the mouse and click to set the second point. The circle is complete.

To create a "From three points" circle:

1.   After you choose the Circle tool as described previously, the Build Mode options appear in the Properties tray.

2.   Select the **From three points** build mode from the Properties tray.
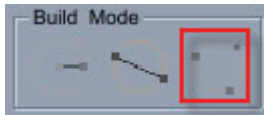
3.   Adjust the **Number points** value to set the number of vertices in the circle.

4.   Click in the Scene window to define the first point.

5.   Drag the mouse and click to set the second point.

6.   Drag the mouse to click and set the third point. The circle is complete.

Working with Selections

You can select vertices, edges, or polygons in an object and then modify them with the many different tools that Carrara provides. There are several different ways to make selections in Carrara, each of which will be discussed in the sections that follow.
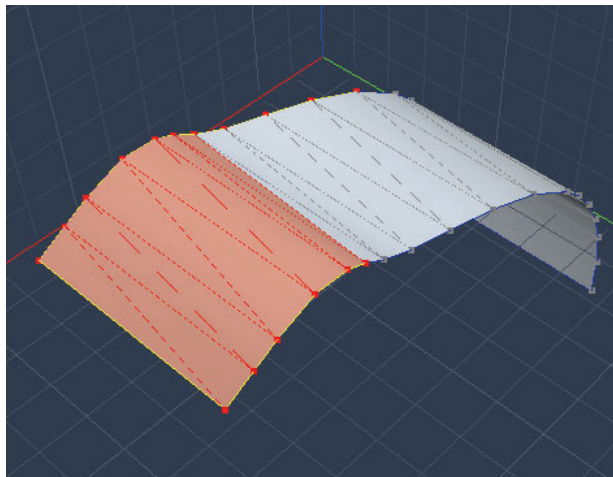
Selection Tools

The left toolbar in the Model room contains three selection tools, which can be used as follows:

Marquee Selection

Lasso Selection
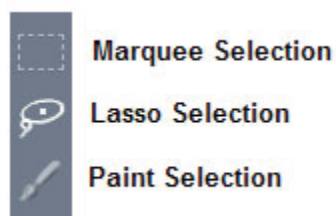
Paint Selection

- **Marquee Selection**: Drag a rectangular area to surround the objects, edges and vertices you want to select.

- **Lasso Selection**: Drag to define an irregularly-shaped area around the objects, edges and vertices you want to select.

- **Paint Selection**: Click and drag the paint cursor over the elements you want to select.

Vertex Modeler Selection Modes

Selection controls in the Vertex modeler allow you to select vertices, edges, or polygons in your model so that you can modify your geometry using Carrara's modeling tools. You can assign names to your selections so that you can easily retrieve them at a later point in your modeling. You can also assign selected areas to a shading domain or morph area.

Choosing a Selection Mode

There are four different modes of selection in the Vertex modeler: Regular, Vertex, Edge, and Polygon. Click their icons at the top of the Properties tray to switch the selection mode from one to another.

By default, selected vertices, edges and polygons are highlighted in red.

- **Regular Selector**: Vertices, edges and polygons can be selected. Furthermore, this mode automatically completes' the selection when possible. For example, if you select two vertices linked by an edge, the edge is also selected. If all the edges around a polygon are selected, the polygon is also selected.

- **Vertex Selector**: Selects only the vertices that you click.

- **Edge Selector**: Selects only the edges that you click.

- **Polygon Selector**: Selects only the polygons that you click.

Selecting Vertices

You can use the Regular selector or the Vertex selector to select vertices.

To select vertices:

1. Open the model that you want to modify in the Vertex modeler.

2. Click the Selection tool.

3. Select vertices as follows:

- Choose **Regular Selector mode** from the Properties tray, and click to select a vertex. **Shift+Click** additional vertices to add to the selection. Ctrl/Cmd+Click vertices to remove them from the selection.

- Choose **Vertex Selector mode** from the Properties tray, and click individual vertices or draw a selection rectangle around multiple vertices. Press the **Shift** key while selecting single or multiple vertices to add to the selection. Press the **Ctrl/Cmd** key while selecting

single or multiple vertices that you want to remove.
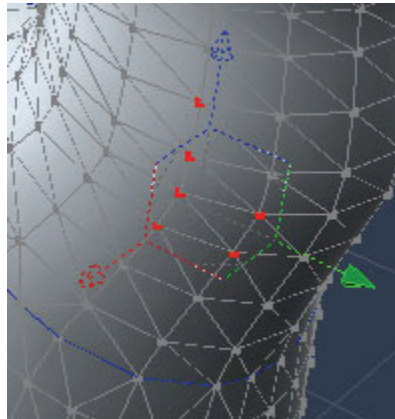
Selecting Edges

You can use the Regular selector or the Edge selector to select edges.

To select edges:

1. Open the model that you want to modify in the Vertex modeler.

2. Click the Selection tool.

3. Select edges as follows:

    - Choose **Regular Selector mode** from the Properties tray, and click to select an edge (the line between two vertices). **Shift+Click** additional edges to add to the selection. **Ctrl/Cmd+Click** edges to remove them from the selection.

    - Choose **Edge Selector mode** from the Properties tray, and click individual edges. You can also begin a rectangular selection from the outside of the object and drag inward to draw a selection rectangle around multiple edges. Press the **Shift** key while selecting single or multiple edges to add to the selection. Press the **Ctrl/Cmd** key while selecting single or multiple edges that you want to remove.
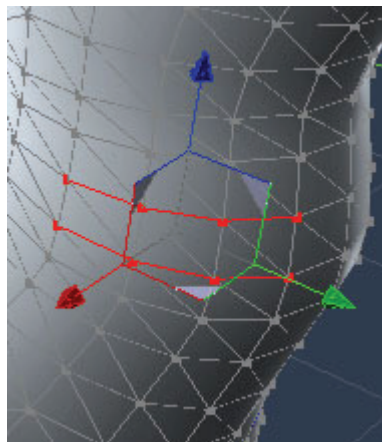
Selecting Adjoining Edges

An adjoining edge is considered to lie in the same direction if the angle formed between the selected edge and the adjoining edge is equal to or less than the value that you set in the **Properties tray: Tool Options: Edge Propagation** field.

If the adjoining edge lies in the same direction, it is selected and the angles formed between it and any adjoining edges are evaluated. In this way, the selection propagates until there are no more adjoining edges that meet the selection criteria.
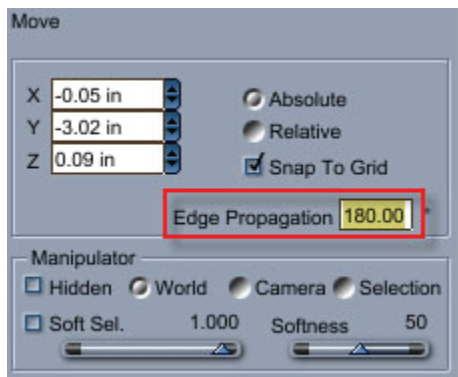
To select an edge and all connected edges in the same direction:

1. Open the model that you want to modify in the Vertex modeler.

2. Click the Selection tool.

3. Set the **Edge Propagation** setting in the Properties tray to the highest angle of edges that you want to select.

4. Double-click an edge in the object. All connecting edges below the set angle value are selected.
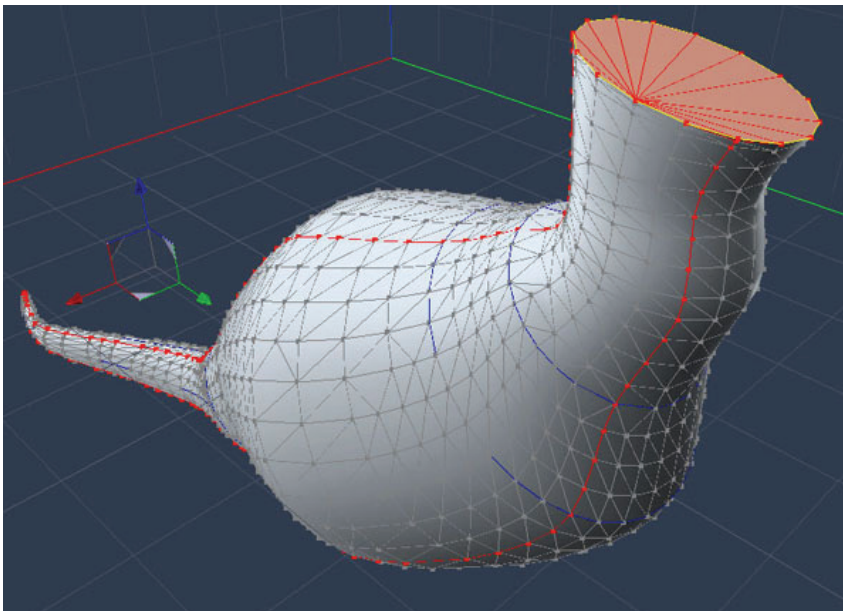
Selecting Polygons

You can use the Regular selector or the Polygon selector to select polygons.

To select polygons:

1. Open the model that you want to modify in the Vertex modeler.

2. Click the Selection tool.

3. Select polygons as follows:

   - Choose **Regular Selector mode** from the Properties tray, and click to select a polygon. **Shift+Click** additional polygons to add to the selection. Ctrl/Cmd+Click polygons to remove them from the selection.

   - Choose **Polygon Selector mode** from the Properties tray, and click individual polygons. You can also start from outside the object to draw a rectangular selection around multiple polygons. Press the **Shift** key while selecting single or multiple polygons to add to the selection. Press the **Ctrl/Cmd** key while selecting single or multiple polygons that you want to remove.
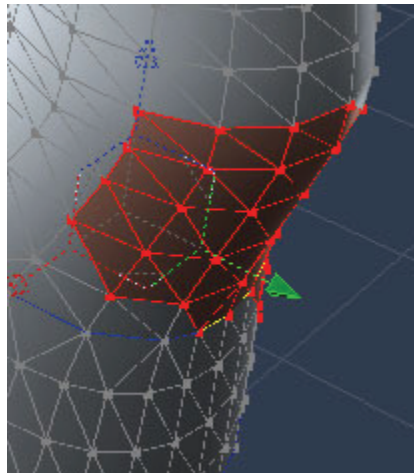
Selecting Objects

To select an entire object:

1. Click the Selection tool.

2. Double-click in the middle of any polygon. The entire object is selected.
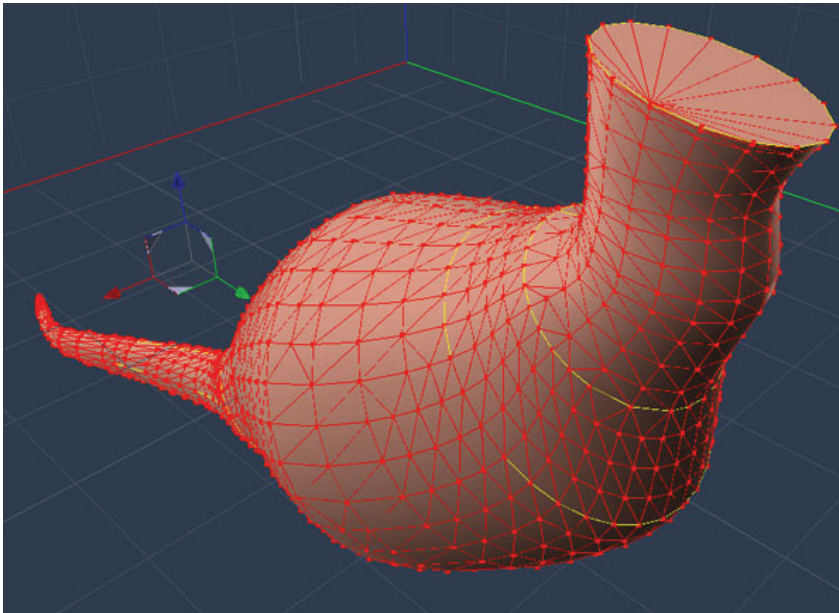
Soft Selections

Complementing the selection modes, there is a **Soft Selection** option that smoothly increases the influence around the active selection.

This feature is especially handy for when you want to create morphs for a human character. For example, select a polygon that corresponds to a part of the cheek on a face. If you move the selected polygon without Soft Selection, it will be the only element that moves.
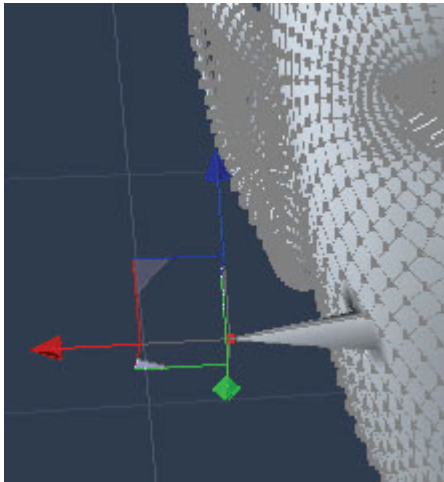
When you activate the Soft Selection feature, an influence area appears around the selected element. The area is represented by red polygons (maximum influence) whose color becomes grey (minimum influence). When you move the soft selection you will see that a part of the nose is modified as well while it smoothly follows your manipulation.
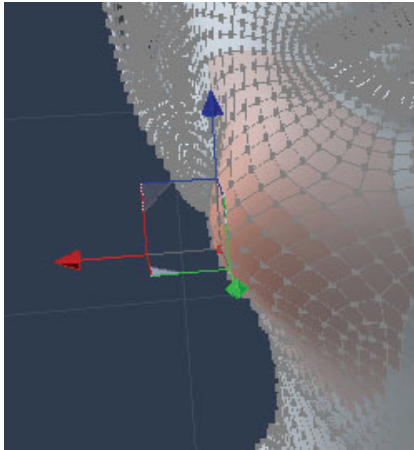
Two parameters, in the Properties panel, will allow you to fine-tune the Soft Selection:
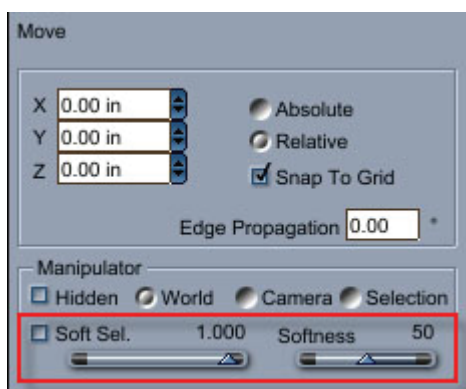
- **Radius**: defines the size of the influenced area around the selection.
- **Softness**: the influence ratio in the soft selection area. The higher the value, the less flexible the modified surface.

Viewing Selection Statistics

The **Properties tray: Model tab: Statistics panel** allows you to keep track of the number of polymeshes, polygons, edges, and vertices that are currently selected. This panel is located at the bottom of the Model tab.

Selection Menu Commands

The Selection menu in the Model room provides several different commands that allow you to work with selections in many different ways. You can assign names to selections and recall them at any time. You

can also select specific shading domains or morph areas in an object. The following sections describe how to use these commands.

Saving Selections

When you're working with complex objects, it's often convenient to save selections of objects so that you can easily reselect them.

Note You can have as many saved selection as you want. To set the maximum number of saved selection, use the Vertex modeler preference dialog (see "Setting Preferences for the Vertex Modeler").

To save the current selection:

- Choose **Selection > Add To Saved Selections**. Carrara adds the current selection to the others that can be recalled.

Restoring Selections

You can use the Restore Selection command to toggle through the saved selections until you find the one you want to work with.

To restore the saved selection:

- Choose **Selection > Restore Selection**. The previous saved selection is restored.

- To restore additional previous selections, choose **Selection > Restore** selection again.

Working with Named Selections

You can assign names to polymeshes, polygons, edges and vertices. Once named you can reselect the selection by name. Named selections are also used by some exporters.

Even though you can use the Select By Name feature to select large parts of your vertex object, the selection only directly applies to vertices. For example, when you select an edge, two vertices are selected.

In some cases, this means that you may actually select edges unintentionally. For example, when you select two edges of a triangular polygon, the third edge also becomes selected because all three vertices in the triangle are selected with the first two edges.

To name a selection:

1. Create a selection of polymeshes, polygons, edges, or vertices.

2. Choose **Selection > Name**, and then select the type of selection you have made: **Polymesh**, **Polygons**, **Edges**, or **Vertices**. A dialog prompts you to assign a name to the selection.

3. Enter a name for the selection and click **OK**.

To select named objects:

1. To select by name, choose **Selection > Select by > Name**. The Select by Name dialog appears.

2. Choose the type of selection (polymesh, polygon, edge, or vertex) that you want to reselect. If selections of that type have been assigned, their names appear in the Name List.

3. Select the name you want to select.

4. Click **OK**.

To deselect named objects:

1. To select by name, choose **Selection > Deselect by > Name**. The Deselect by Name dialog appears.

2. To deselect, choose **Selection > Deselect by Name**.

3. Choose the type of selection (polymesh, polygon, edge, or vertex) that you want to deselect. If selections of that type are selected, their names appear in the Name List.

4. Select the name you want to deselect.

5. Click **OK**.

Hiding Selections

When you're editing a complex object, you might want to hide parts of the object to make it easier to view and select the vertices and edges you want to work with.

To hide a selection:

1. Select the vertices, edges, or polygons that you want to hide.

2. Choose **View > Hide Selection**.
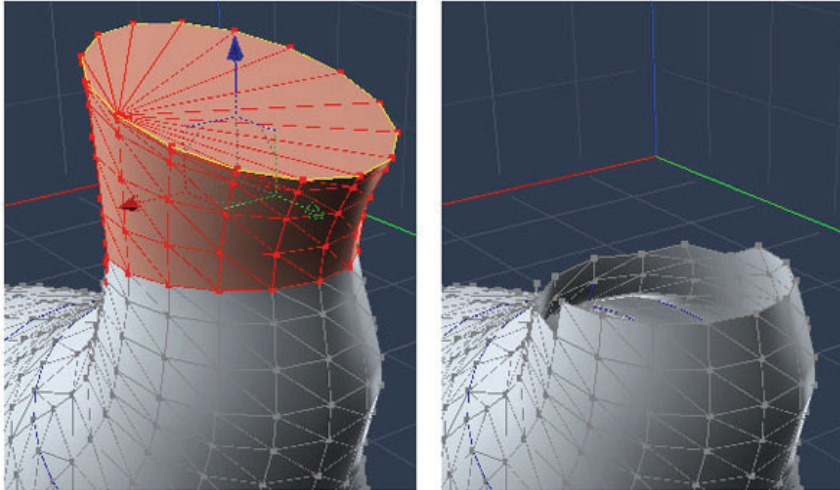
To reveal hidden vertices in an object:

1. Select the object.

2. Choose **View > Reveal Hidden Vertices**. All previously hidden vertices reappear.

Deselecting Items

To deselect all selections:

1. Click the Selection tool.

2. Click in an empty area of the modeling window.

Using the Properties Tray Selection Tools

A number of different selection tools appears in the Properties tray in the Model Room. From left to right, they are the **Loop**, **Ring**, **Between**, **Invert**, **Shrink**, and **Grow** Selection tools.
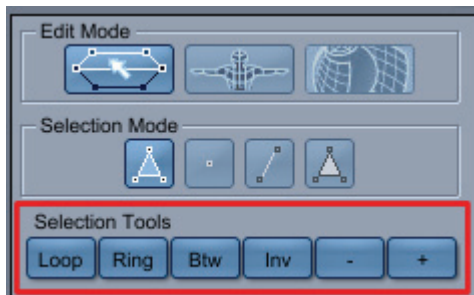
Looping from a Selection

The Loop tool lets you expand a selection along a surface by continuing it as far as possible in the direction you started. The Loop tool works with polygons and edges, but not with vertices. For polygons, you must have at least two polygons selected to indicate the direction to expand the selection.

To select a loop:

1. Select an edge, or two polygons to start.

2. Do one of the following:

   - Click on the **Loop** icon, in the property panel.

   - Choose **Selection > Loop**.
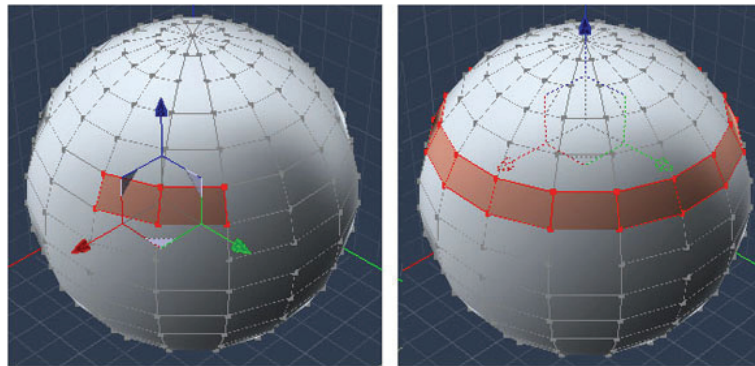
Ring from Selection

The Ring tool is similar to the Loop tool, but it expands a selection in parallel (like the rungs on a ladder). The selection expands along a surface by continuing it as far as possible in the direction you started. The Ring tool works only with edges.

To select a ring:

1. Select an edge.

2. Do one of the following:

   - Click on the **Ring** icon, in the property panel.

   - Choose **Selection > Ring**.
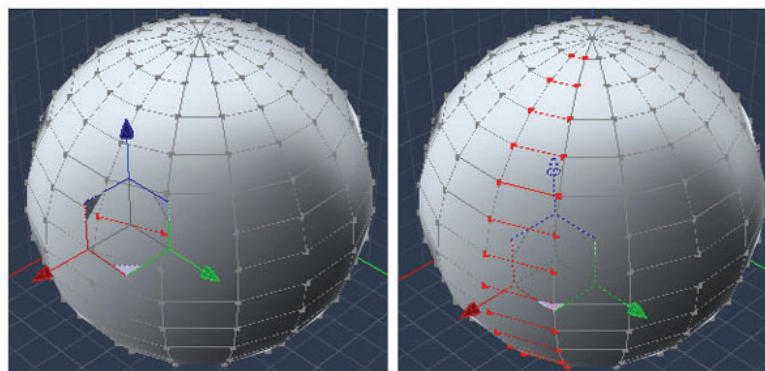
Selecting Between (or Select Connect)

The Between tool lets you connect a selection on a surface after defining its start and finish. The tool only works on simple cases, like a direct path. Since this tool is based on Loop and Ring, it will stop at the same types of problems which would stop those other tools.

To select between:

1. Select an edge or polygon as start element.

2. Select an edge or polygon as end element.

3. Do one of the following:

    - Click on the **Btw** icon, in the property panel.

    - Choose **Selection > Connect**.
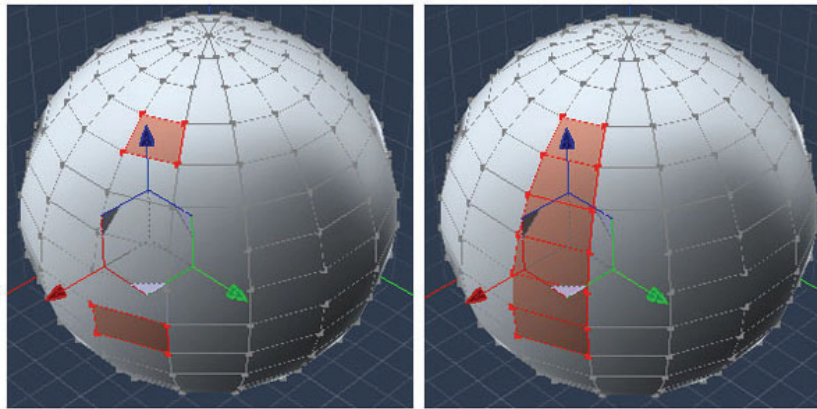
Inverting the Current Selection

Inverting the selection deselects the current selection and selects all other items.

To invert the current selection:

Do one of the following:

- Click the **Inv** icon in the Property panel.

- Choose **Selection > Invert Selection**.
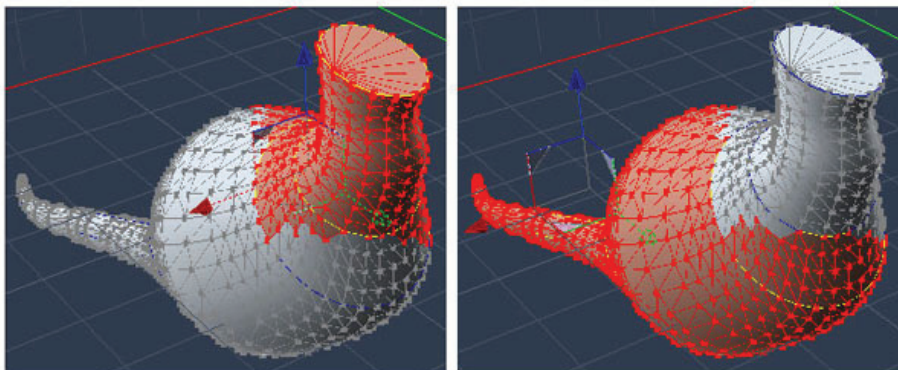
Shrinking Selections

The Shrink Selection tools allows you to reduce the selection by unselecting the items located on the outer limit of the currently selected items.

To shrink the selection:

1. Create a selection of vertices, edges, or polygons.

2. Do one of the following:

   - Click on the - icon, in the property panel.

   - Choose **Selection > Shrink**. Repeat as many times as necessary to shrink the selection as needed.
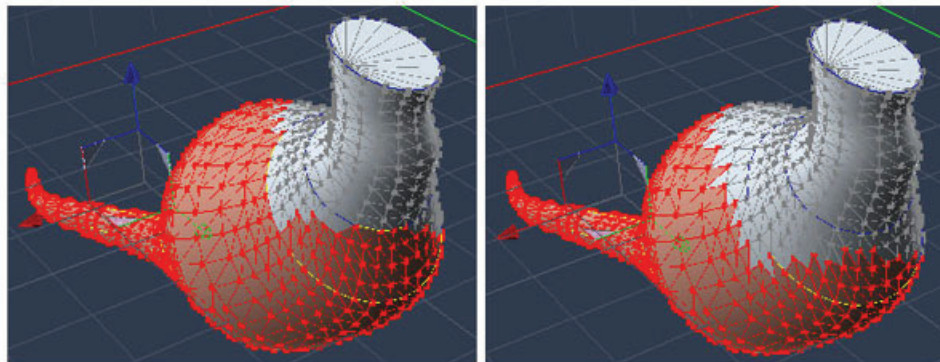
Growing Selections

The Grow Selection tools allows you to add to the selection faces, edges or points close to the currently selected items.

To grow the selection:

1. Create a selection of vertices, edges, or polygons.

2. Do one of the following:

   - Click on the + icon, in the Property panel.

   - Choose **Selection > Grow**.
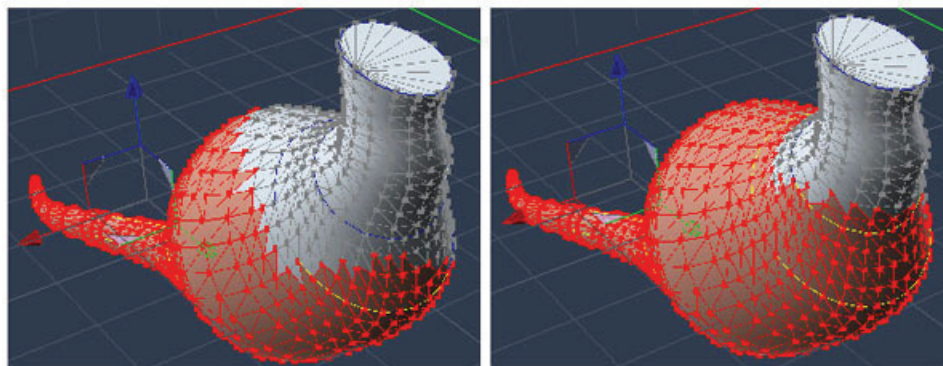
Working with Vertices and Edges

The power of Vertex modeling lies in the ability to directly position vertices and edges. By joining separate objects and moving vertices, edges, and polygons, you can model just about anything you can imagine.

Using the Magnet to Attract Vertices (Standard, Pro)

You can selectively move vertices in an object with the Magnet tool. This tool behaves like a magnet, attracting the vertices within its sphere of influence. The effects of the Magnet tool are most noticeable when you use it on a highly defined object.
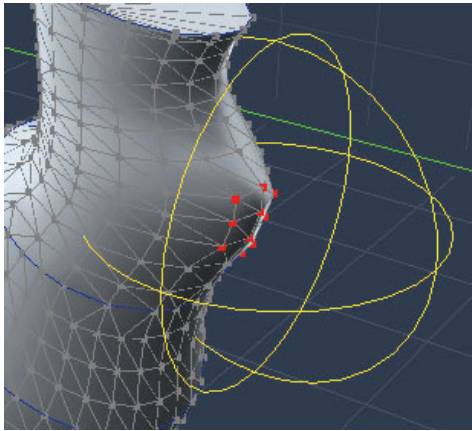
- The radius of the sphere of attraction.

- Whether nearby edges and vertices are attracted or only the selected vertices and edges are attracted.

To attract a group of vertices:

1. Select the vertices you want to move or attract.

2. Click the Magnet tool.

3. Adjust the settings in the Properties tray as needed:
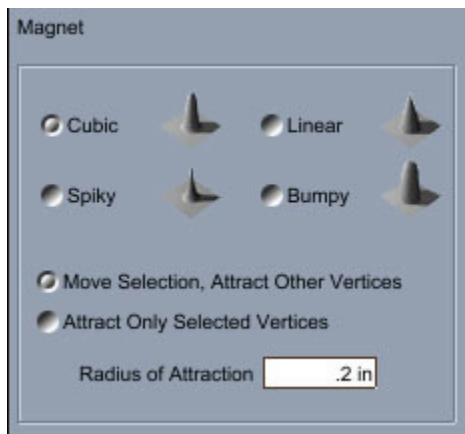
- Choose the shape of the attraction curve. Previews are shown in the Properties tray of the approximate shape that you will get from each shape. Choices are **Cubic**, **Linear**, **Spiky**, or **Bumpy**.

- Choose **Move Selection, Attract Other Vertices** to move the selected vertices as they would be with the Selection tool. Other vertices within the sphere of attraction are moved towards the selected vertices.

- Choose **Attract Only Selected Vertices** to move selected vertices only.

- Set the **Radius of Attraction** to determine the area around the tool that will be affected.

4. Drag the Magnet tool in the direction in which you want to move the vertices.

5. To constrain the move operation to the direction perpendicular to the Drawing plane, hold down the **Command/Ctrl** key.

Repositioning Vertices Numerically

In the Vertex modeler, you can precisely reposition vertices in two ways:

- Offset a selection from its current location with the Move tool.

- Change the coordinates of a vertex, edge, or polygon using the Properties tray. For information on using the Properties tray, refer to "Checking Object Properties."

To offset a selection by a specific amount:

1. Select the object or portion of the object you want to move.

2. In the Move tool options of the Properties tray, specify the amounts you want to move the selection on the X, Y and Z axis.
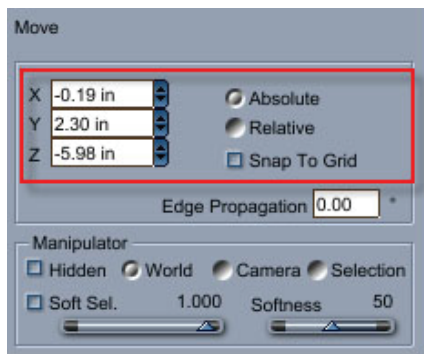
Deleting Vertices

Removing a vertex also removes the edges that connected the vertex to other vertices, unless the edges connected to this vertex are two. In that case, a new edge is created to replace the two previous ones.

To delete a vertex (all versions):

1. Select the vertex or vertices to delete.

2. Do one of the following:

- Choose **Edit > Dissolve** (creates new edges).

- Press the **Backspace** key (creates new edges).

- Press the **Delete** key (removes edges).

To delete a vertex (Standard, Pro):

1. Click the **Delete** tool.

2. Click the vertex you want to remove.

Adding Vertices (Standard, Pro)

You can edit existing polylines or edges by adding vertices on them. Adding vertices is useful when you need more control over the object than is provided with the existing vertices.

To add a vertex:

1. Click the Add Vertex tool.

The Add Vertex tool options show in the Properties Tray.

2. Choose one of the following options:

    *   **Constrain Position**: The new vertex is added at 1/4, 1/3, or 1/2 of the edge.
    *   **Free Position**: Adds a vertex at any position on the edge. You can also drag with the Add Vertex tool to add and immediately move a new vertex.

3. When two vertices are added consecutively to the same polygon, they are automatically linked with an edge. This behavior can be switch off by unchecking the **Auto-link** option.

4. Click an edge where you want to add the vertex. The new vertex is highlighted in red.

Welding Vertices (Standard, Pro)

There are three ways to weld vertices: with the Weld command, or with the Weld or Target Weld tools.

Using the Weld Command

You can join separate objects by welding selected vertices. Normally, if two vertices are within a specified distance from each other, they are welded when you use the Weld command. However, if welding the vertices would cause a single edge to be shared by more than two polygons, the vertices are not joined.
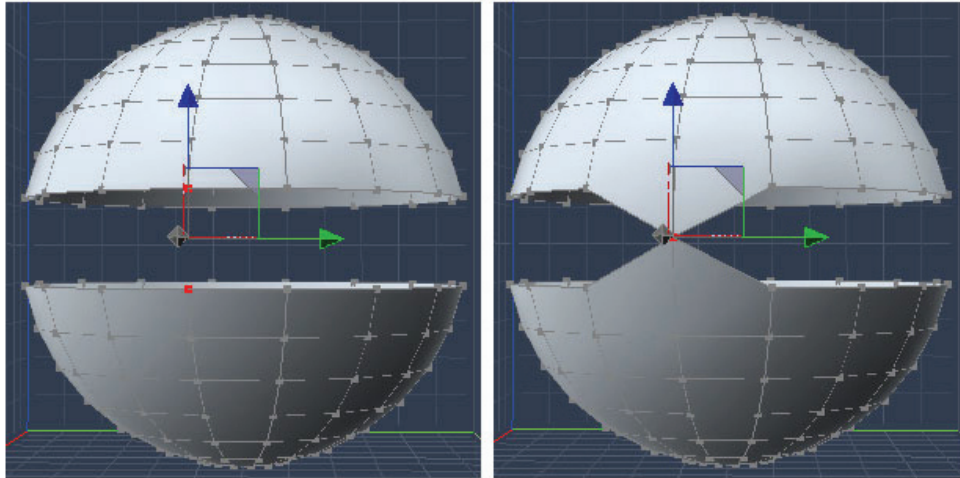
To weld vertices using the Model > Weld command:

1. Select the vertices to be welded.

2. Choose **Model > Weld**. The Weld Selected Vertices dialog appears.
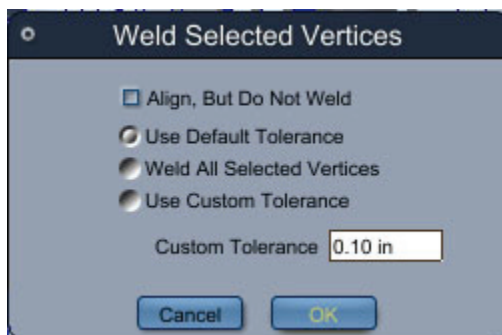
3. Set weld options:

   - **Align but do not weld** aligns the vertices but does not make two or more objects one, as the Weld command does. The vertices move to the average position of all the vertices that lie within the threshold.

   - **Use default tolerance** welds the vertices if the distance between them is within the default tolerance specified in the Vertex modeler preferences.

   - **Weld all selected vertices** welds the selected vertices no matter how far apart they are.

   - **Use custom tolerance** specifies the maximum distance vertices can be from each other and still be welded.

   - **Custom tolerance** determines how close the vertices must be to be welded. Enter a value.

4. Click **OK**. The welded vertices are joined into a single vertex.

Using the Weld Tool

Two vertices can also be welded together using the Weld tool or the Target Weld tool. The Weld tool welds points and moves the result at mid distance of the two selected points, whereas the Target Weld tool welds vertices and moves the first selected vertex to the position of the second vertex.

To Weld vertices using the Weld tool:

1.   Click the Weld tool.

2.   Click on the two vertices to be welded together.

Using the Target Weld Tool

The Target Weld tool allows you to weld two vertices so that the first selected vertex is moved to the second selected one and merged with it.

The Target Weld tool can also be used to weld edges.

To weld a vertex or edge to another vertex or edge using the Target Weld:

1.   Click the Target Weld tool.

2.   Click on the vertex to weld to another vertex.

3.   Click on the target vertex. The first vertex is moved to the second vertex position and the two vertices are welded.
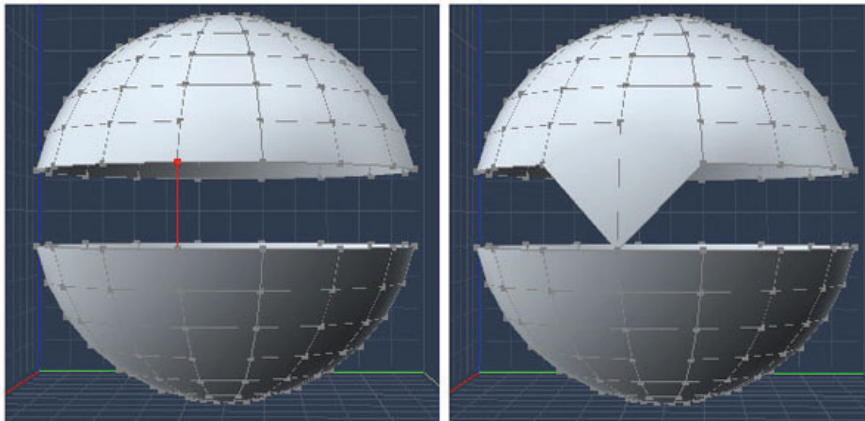
Linking and Unlinking Vertices (Standard, Pro)

Linking vertices creates an edge between them. Unlinking vertices removes the edge that connects them, but does not affect the vertices themselves.

Linking can be used for breaking a polygon in two between two of its vertices, or for closing or extending a polyline.

It is not possible to create an edge on a polymesh if the two vertices do not belong to the same polygon. For instance, linking two vertices on the border of a hole is not possible.

To link vertices:

1. Click the **Link** tool.

2. Select the first vertex. This vertex will turn yellow.

3. Select the second vertex. An edge is created to connect the vertices. If the two vertices are part of the same polygon, the polygon is divided into two smaller polygons.
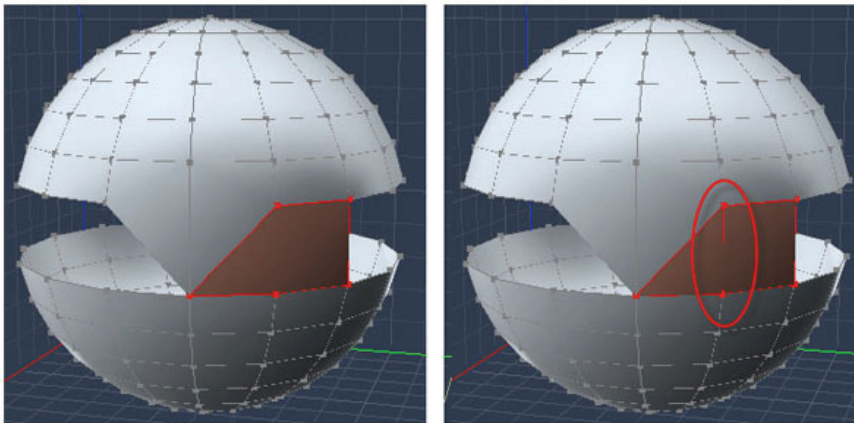
To unlink vertices:

1. Select the vertices you want to unlink.

2. Choose **Model > Unlink**. The edge connecting the two vertices is removed.

TIP! You can also unlink two vertices by clicking the edge with the Delete tool or by selecting the edge and using the Dissolve tool or hitting the Backspace key.

Deleting Edges

To delete an edge (all versions):

1. Select the edge or edges to delete.

2. Do one of the following:

   - Choose **Edit > Dissolve** (creates new edges).
   - Press the **Backspace** key (creates new edges).
   - Press the **Delete** key (removes edges).

To delete an edge (Standard, Pro):

1. Click the **Delete** tool.

2. Click the edge you want to remove.

Removing an edge does not remove the vertices that it connected.

Deleting Polygons

To delete a polygon (all versions):

1. Select the polygon or polygons to delete.
2. Press the **Delete** key.

To delete a polygon (Standard, Pro):

1. Click the **Delete** tool.

2. Click the polygon you want to remove.

- Removing a polygon does not remove the edges around it.
- Deleting a polygon is the same as emptying it (with the **Model > Empty Polygon** command).

Detaching Polygons (Standard, Pro)

You can detach polygons from objects and move them to another location. The main object will have a hole where you detached the polygon.

To detach a polygon:

1. Select the polygon you want to detach from the main object.
2. Choose **Model > Detach Polygons**.
3. Drag the polygon to any position you like.

Advanced Edge Tools (Pro only)

The Vertex modeler provides several advanced tools to manipulate edges quickly. These tools can be used to improve the topology of polygonal objects.

Quick Fillet

The Quick Fillet tool lets you extract a selected edge along the edges next to it. The result is similar to using the Fillet tool set to range 0.

To use the Quick Fillet tool:

1. Select one or several edges on a surface or a volume. This command also applies to edges in selected vertices or polygons.
2. Do one of the following:

- Select the Quick Fillet tool.

- Choose **Model > Fast Fillet Edge**.

3. Click the selection and drag in the direction that you want the new edges to move, until they reach their new position. The new edges are colored green until you release the mouse button.
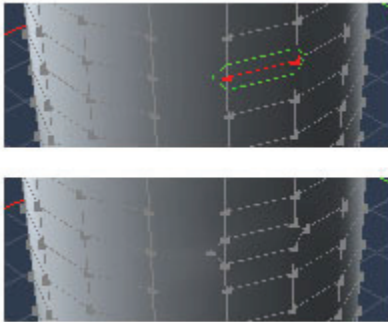
Extract Around

The Extract Around tool lets you create two edges parallel and around the original selected edge, or edges. Unlike the Quick Fillet tool, the original edges remain. This command is handy for creating a raised button or a pushed-in area.

To extract edges around:

1. Select one or several edges of a surface or volume. This command also applies to edges in selected vertices or polygons.
2. Do one of the following:
   - Select the Extract Around tool.

   - Choose **Model > Extract Around Edge**.
3. Click and drag on the selected edge or edges, and move it to define the offset of the newly created edges.
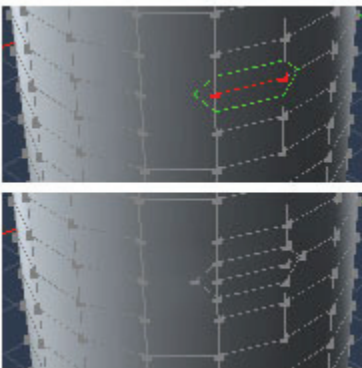
Extract Along

The Extract Along tool lets you extract a selected edge along the edges next to it. This tool is useful for fine-tuning a model, for example adding a loop of edges around an existing edge.

To extract edges along:

1. Select one or several edges on a surface or volume.

2. Do one of the following:

    - Select the Extract Along tool.

    - Choose **Model > Extract Along**.

3. Click and drag on the selected edge and move it to define the placement of the newly created edges.
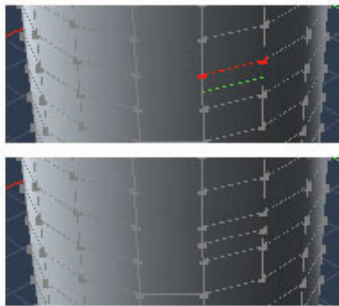
Move Along

The Move Along tool lets you move an edge along its adjacent face. It is a very useful tool for fine-tuning, for example, moving an "edge loop" on an existing surface, like around an eye.

To move edges along:

1. Select one or several edges on a surface or a volume.

2. Do one of the following:

    - Select the Move Along tool.

    - Choose **Model > Move Edge Along Edge**.

3. Click-drag on the selection of edges, and move them to their new position.
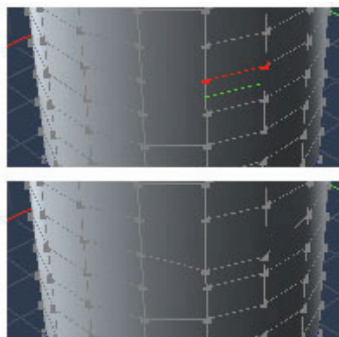
Working with Objects

The Vertex modeler provides a number of ways to manipulate an object without directly editing individual vertices and edges. This section describes how you can perform the following:

Transforming Objects and Selections

The Vertex modeler provides a number of ways to move, scale, and rotate vertex objects or selections. The most convenient way consists of using one of the 3D manipulators. But it can be necessary to use one of the other methods, such as entering values numerically to define a scale factor, for instance.

The 3D Manipulator

The 3D manipulator is displayed in the graphic view at the center of the selection. Depending on the selected mode (move, rotate, scale or universal) it shows several colored handles that can be used to transform the selection.

You can choose which manipulator to display by selecting its icon in the left tool palette or by using keyboard shortcuts.
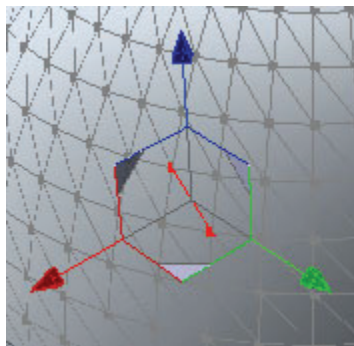
The Move Manipulator

Shortcut: **T**

This manipulator moves (translates) the selected elements. The manipulator is symbolized by arrow points (2D and 3D).

If you click on an axis, the movement of the selected item is restricted to that axis.

If you click on a triangle on the same plane as the two axis, the movement is restricted to that plane.

If you click drag on the selected element, the object is moved in the active plane.

The Scale Manipulator

Shortcut: **S**

This manipulator scales the selected elements. It is symbolized by cubes. If you click on a cube on the end of an axis, the scaling of the selected element is limited to that axis.

If you click on one of the grey cubes on the same plane as the two axes, the scaling is restricted to that plane.

This manipulator also shows a global interactive box which surrounds the object. Click on a face of this box to scale the object as if you were pulling on the face. Click on an edge of this box to pull on both faces that touch that edge. Click on a corner of the box to scale the box proportionally away from the opposite corner.

The Rotate Manipulator

Shortcut: **R**

This manipulator rotates the selected elements. It is symbolized by circles. If you click on a circle corresponding to an axis, the rotation of the selected element is limited to that axis.

The Universal Manipulator

Shortcut: **U**.

This manipulator is a combination of the three fundamental manipulators. It lets you perform moves, rotations and scalings without changing the tool.

Using this manipulator is practical to move the objects globally. However, for precise manipulations of 3D entities like vertices or edges, the number of clickable modifiers on each axis may become a problem.

3D Manipulator Properties

When one of the manipulators is selected, the following options are available from the property panel:
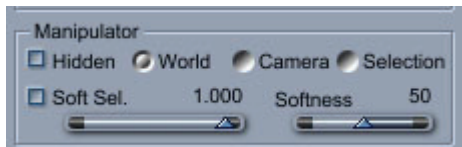
- **Hidden:** It is sometimes necessary to hide the manipulator for being able to select or view small details that can be hidden by the numerous controls of the 3D manipulator. Check the **Hidden** option to hide the manipulator.

  NOTE! You can still move, rotate,or scale selected items even if the manipulator is not visible but only in the active plan.

- Three options determine how the manipulator transforms the selected objects:
  - **World (default mode):** the 3D manipulator transforms the selection relatively to the world axes.

- o **Camera:** the manipulator transform takes place in the camera plane. Changing the camera orientation changes the transform plane. This mode is useful for tweaking selections.
- o **Selection:** the manipulator is oriented in the main plane of the selection. For instance, if a polygon is selected, the XY plane of the manipulator will be the plane of the polygon. This mode is convenient for modeling precisely while keeping the orientation of the selection.

Moving Objects to the Drawing Plane (Standard, Pro)

To move a selection to the Drawing plane:

1. Select the vertices, edges, polygons, or objects you want to move to the Drawing plane.

2. Choose **Model > Move to Drawing Plane**.
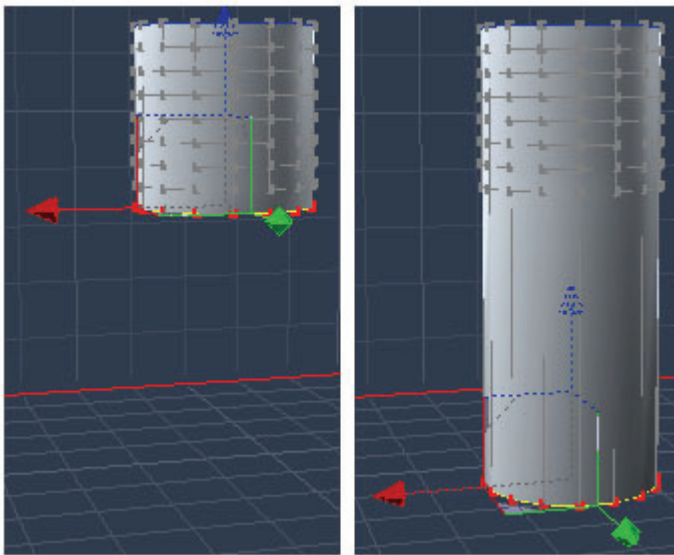
Converting Spline and Metaball Objects

You can create objects in the Spline modeler or Metaball modeler and then convert them to vertex objects that can be modified further in the Vertex modeler.

NOTE! If you edit a spline or metaball object in the Vertex modeler, its conversion is permanent; you cannot reopen it in the native modeler.

To convert a spline or metaball object:

1. Open the Spline modeler or the Metaball modeler and select an object.

2.   Choose **Edit > Convert to Other Modeler**. The Convert to Another Modeler dialog appears.
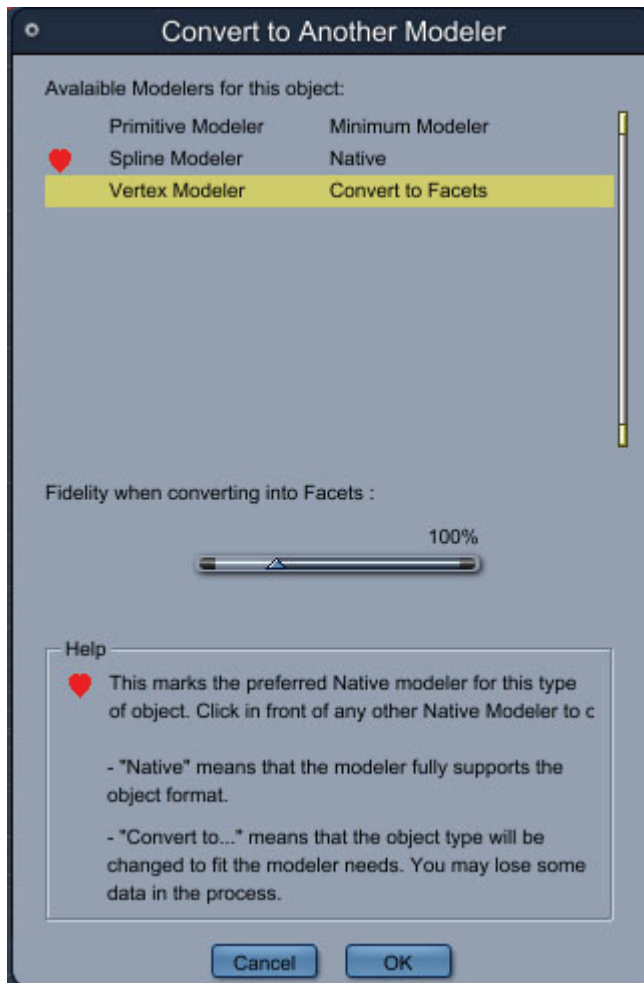
3.   Choose the **Vertex modeler** from the **Available Modelers** list.

4.   Specify the fidelity for the conversion using the **Fidelity** slider. The default, 100%, divides the object's surface into the optimum number of polygons. A higher percentage generates more polygons, providing more points of control.

NOTE! Regardless of the fidelity of the conversion, some data may be lost and the model altered.

5.   Click **OK**. The model opens in the Vertex modeler.

Duplicating Objects

Duplication creates a new object with the position, orientation, and scale attributes of the original object. Duplication is an efficient way of working and has a number of practical applications.

When you choose **Edit > Duplicate**, a second copy of the object is created at the same location as the original.

To duplicate an object:

1. Select one object.

2. Choose **Edit > Duplicate**. The duplicate occupies the same space as the original. Pull the duplicate away by dragging it or moving it numerically.

3. Perform any number of position, orientation, and scale operations without deselecting the object.

4. Choose **Duplicate** again. Each time you duplicate, the new copy receives the positioning, orientation, and scaling of the last duplicated object.

Using Multiple Duplications

You could use the Duplicate feature to build a DNA strand as follows:

1. Create the first base and place it between two helixes.
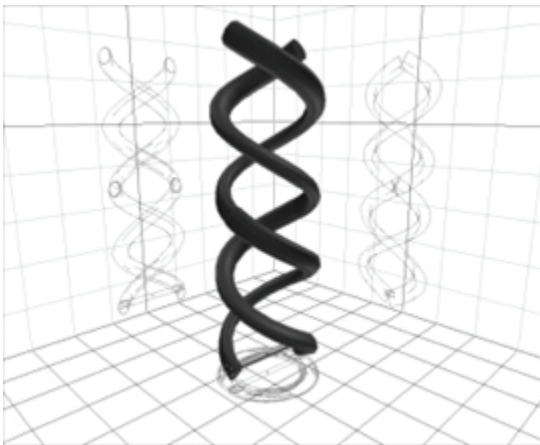
2. Then duplicate the base, raise the duplicate to the level where the next base would be located, and rotate it an appropriate amount.
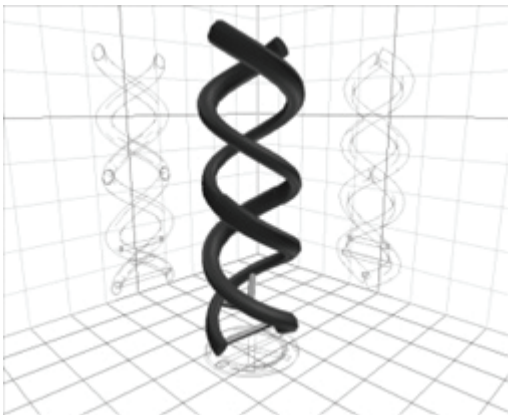
3. Choose duplicate again and again until you have built a DNA strand.
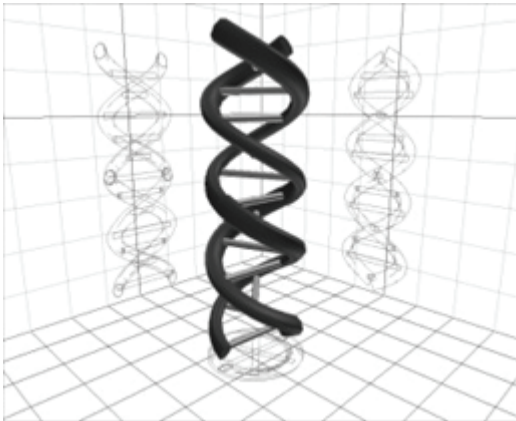
Duplicating Objects with Symmetry

When you choose **Duplicate with Symmetry**, the copy mirrors the original object. The Drawing plane behaves differently from usual in that neither the object nor its duplicate is placed **on** the Drawing plane, but on either other side of it.

To duplicate an object with symmetry:

1. Move the Drawing plane so that it will reflect the original object in the desired location. For further instructions see "Changing the Drawing Plane."

2. Select the object you want to mirror.

3. Do one of the following:

    - Choose **Edit > Duplicate with Symmetry.**

    - Press **Cmd/Ctrl+Shift+D**.
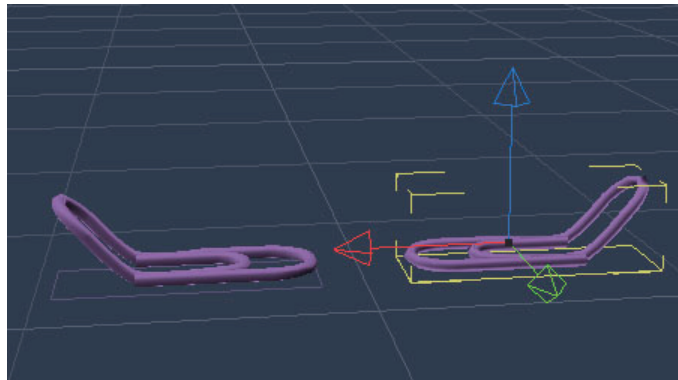
Replicating Objects or Selections

When an object is made of a repeated pattern, a model of the pattern can be created and then be replicated as many times as needed.

You can replicate complete objects in the Assemble room. You can also replicate portions of an object (selected polygons, for example) in the Model room.

To replicate an object:

1. Select the object (or part of the object) to be replicated.

2. Choose the **Edit > Replicate**. The Replicate dialog appears.

3. Choose a replication option:

- **Linear**: Arranges the replicated copies in a straight line from the original selection.

- **Array**: Arranges the replicated copies in a grid, with the original object in one of the corners of the grid.

- **Circle**: Arranges the replicated copies in a circle, with the original selection in its center.
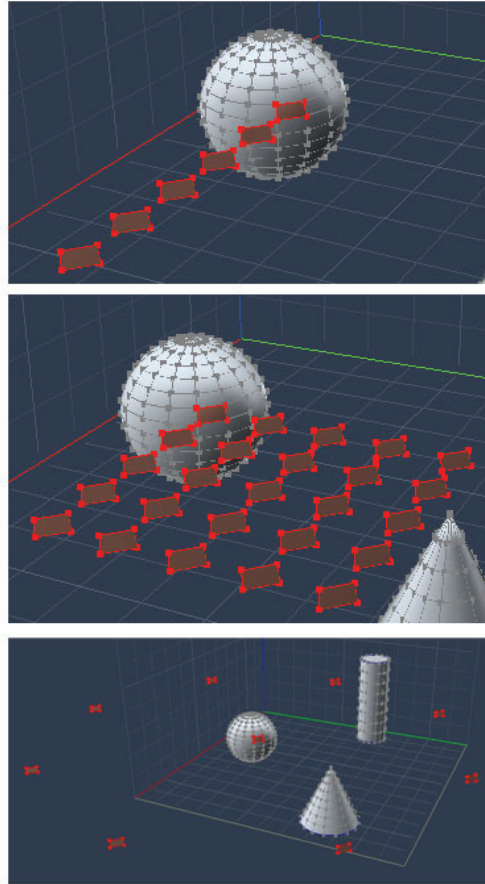
4. For **Linear** replications, set the following options:

- **Number of Copies**: Enter the desired number of copies (in addition to the original) that you want to create.

- **X Offset**: Enter the desired distance between objects along the X axis. Enter 0 if you do not want to replicate along this axis.

- **Y Offset**: Enter the desired distance between objects along the Y axis. Enter 0 if you do not want to replicate along this axis.

- **Z Offset**: Enter the desired distance between objects along the Z axis. Enter 0 if you do not want to replicate along this axis.

5. For **Array** replications, set the following options:

- **U Number**: Enter the number of rows that you want to appear in the array.

- **U Offsets (X, Y and Z)**: Enter the desired distance between each copy in the row.

- **V Number**: Enter the number of columns that you want to appear in the array.
- **V Offsets (X, Y and Z)**: Enter the desired distance between each copy in the column.

6. For **Circle** replications, set the following options:

- **Number of Copies**: Enter the number of items that you want to include in the circle or arc.
- **Arc Angle**: Enter the number of degrees for the circle or arc. For example, enter 360 to create a full circle, or 180 to create a half-circle.
- **Radius**: Enter the dimension of the radius.
- **Axis**: Select the axis along which you want the copies to be created.
- **Rotate Copies**: Check this option to rotate the copies along the path of the circle or arc.

7. Click **OK**. The items are replicated as you specified.

Aligning Objects

You can adjust the alignment of two objects by attaching them at selected vertices, edges, or polygons.

To align one object with another:

1. Select the vertices, edges, or polygons that you want to align on the first object.
2. Shift-select the vertices, edges, or polygons that you want to align on the second object. You must make the same type of selection (vertices, edges, or polygons) on both objects.
3. **Ctrl+Alt+Click** one of the selected vertices on the object you want to be the reference for the alignment. The vertex will turn yellow.

4. Choose **Edit > Align**. Your objects are aligned or distributed in the manner you specified.
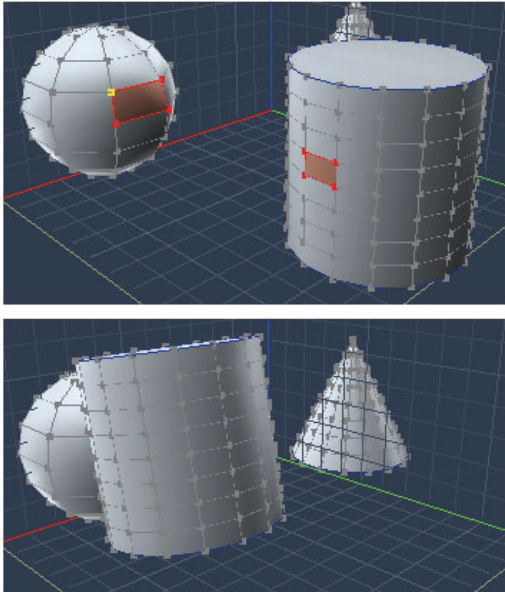
Flipping Objects

You can flip entire objects in the Assemble room, and can also flip selected portions of an object in the Model room. An object can easily be flipped using **Edit > Flip**. The selection is then flipped using the active drawing plane as a reference.

To flip an object:

1. Use the Working Box control to select the active plane.

2. Do one of the following:

   - In the Assemble room, select the object that you want to flip.

   - In the Model room, use the Selection tool to select all or part of an object to flip.

3. Choose **Edit > Flip**. The selected object or portion is flipped along the plane that you specify.

Checking Object Properties

You can use the Model tab to check statistics on the contents of the working box, to name a selected polymesh, and to move selected vertices and objects to particular X, Y, and Z coordinates. The **Properties tray: Model tab** displays the properties of the selected vertex, edge, or polygon.

The information shown in the Modeling tab depends on the current selection. To use the Modeling tab to reposition vertices, you must select either a vertex, an edge, or a polygon of no more than four sides.

   - When a vertex is selected, the Properties tray displays the coordinates of the vertex.

   - When an edge is selected, the Properties tray displays the coordinates of the edge's vertices.

- When a polygon of no more than four sides is selected, the Properties tray displays coordinates of each vertex.
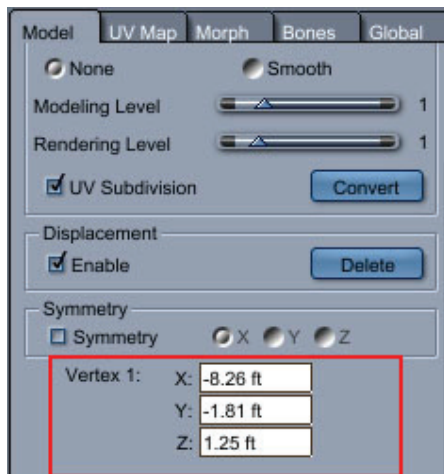
The Properties tray statistics show:

- The total number of polymeshes, polygons, edges, and vertices in the working box.

- The total number of each that are currently selected.
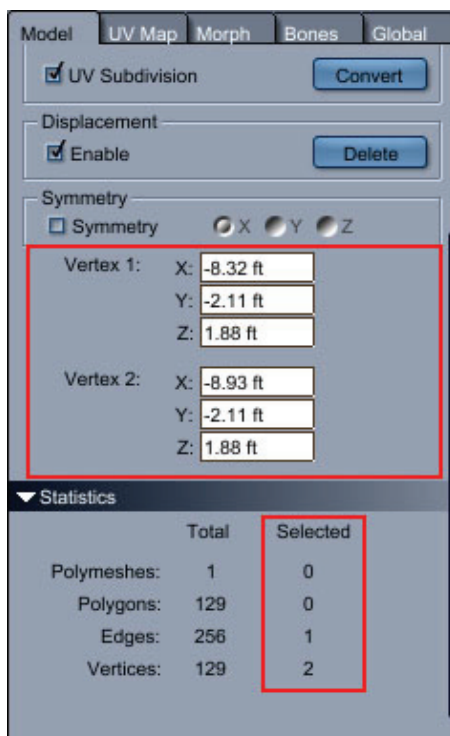
NOTE! You can also use the Modeling tab to specify an object's shading domain. For more information, refer to "Setting a Predefined Mapping."

Editing Selection Properties

The Properties tray also allows you to edit settings and parameters for the selected object. For example, when you select an object and then choose the Move tool, the options for the tool allow you to enter specific X, Y, and Z values for the new position.

To specify object properties in the Properties tray:

1. Select the object whose properties you want to change. You can change the name or the coordinates of the current selection.

2. Directly edit the displayed values in the X, Y, and Z fields.

Using an Image as a Model

It is often useful to place as reference image in the backdrop when you draw polylines. Any image stored as an external file can be placed in the backdrop of the Vertex modeler.

The image is not displayed in the Director's Camera view, but is visible in all other views. For example, you can configure your document window to display the Top, Left, and Front views and display drawings or photographs of the top, left, and front of an object in the respective viewport. This image is just a tool to help you in the construction of a vertex object and will not be visible in the rendering.

Note that the image uses the working box dimensions to set the size on the screen. Therefore, you can use the Zoom and Pan tools to assist in viewing your object while you model.
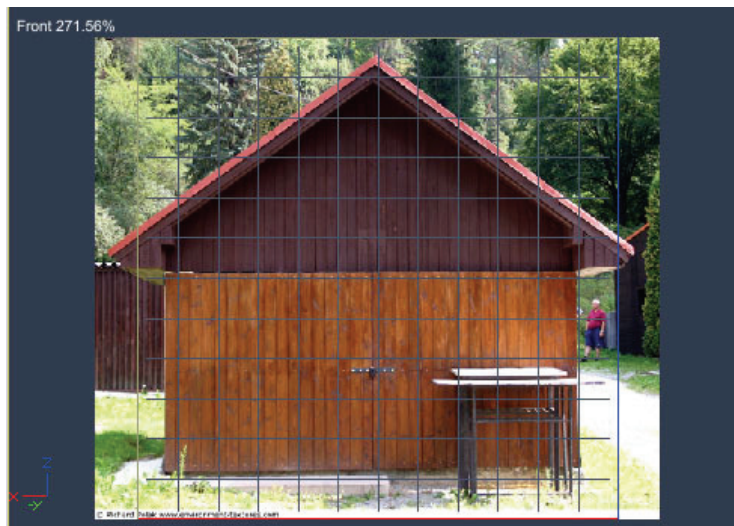
To place an image in the backdrop:

1. In the Model room, choose the **Properties tray: Global tab: Backdrop panel**.

225
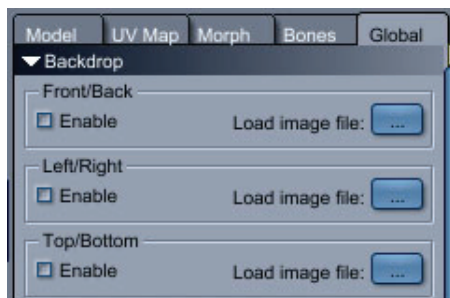
2. Click the **choose an image file** button. The Image File Chooser dialog appears.

3. Locate the file that you want to open and click **Open**.

4. Check the **Enable** option to display it in the appropriate viewport.

Basic Vertex Modeling Functions (Standard, Pro)

The following topics regarding vertex modeling procedures are discussed in this section.

Changing an Object's Definition

The Vertex modeler provides four commands that allow you to change the number of vertices and edges used to describe an object: Subdivide, Tesselate, Decimate, and Triangulate. You can apply these actions to an entire object, or to portions of an object.

You can learn more about these tools in the following sections.

Subdividing an Object

Subdividing an object adds vertices and edges to give you finer-grain control over an object's surface.

To subdivide selected polygons:

1. Select the polygons you want to subdivide.

2. Choose **Model > Subdivide**.
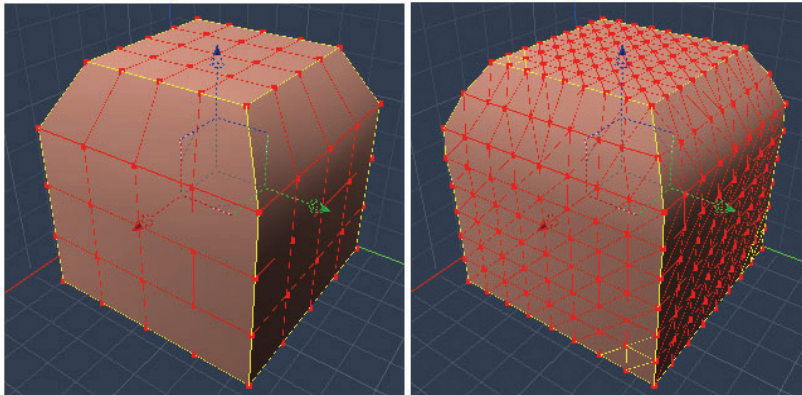
Tessellating Objects

The Tesselate tool and command allows you to change the number of vertices in the object, based on one of three different tessellation patterns. To switch from one mode to the other, click their respective icons in the tool options of the Properties tray.

- **Vertex To Center** Links the center of the polygon to its vertices.
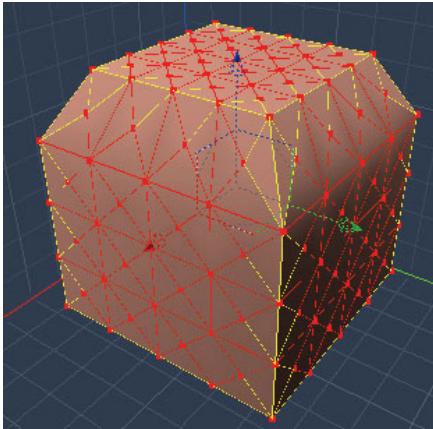
- **Mid-Edge To Center** Creates new vertices in the middle of the edges of the polygon and links them to the center of the polygon.
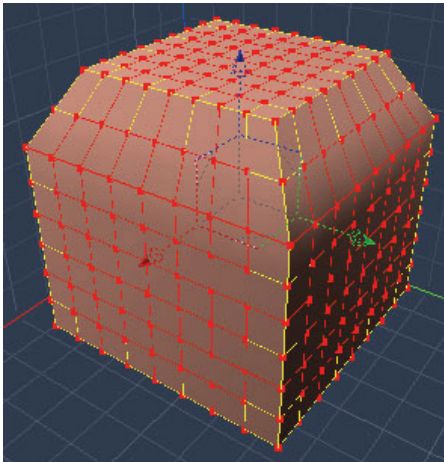
- **Mid-Edge To Mid-Edge** Creates vertices in the middle of the edges and links them consecutively one to the other.
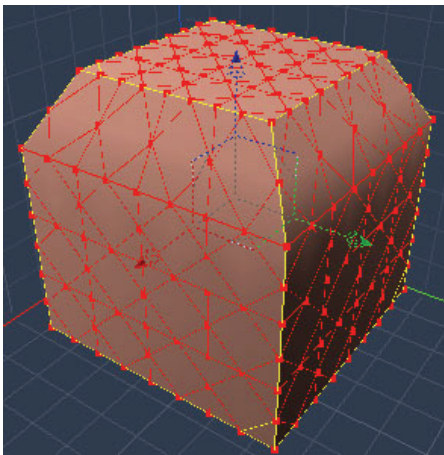
To tessellate a single polygon:

1. Do one of the following:

2. Click the **Tesselate tool** and select the type of tesselation desired from the Properties tray.

3. Choose **Model > Tesselate**.

4. The Tesselate options show up in the Properties Tray. Choose **Vertex to Center**, **Mid-Edge to Center**, or **Mid-Edge to Mid-Edge** from the submenu.

5. Click the polygons that you want to tessellate.
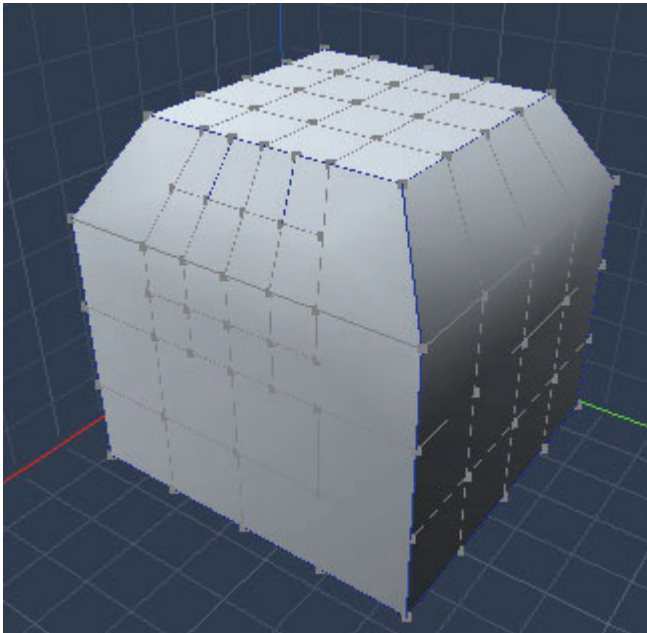
To tessellate multiple polygons:

1. Select the polygons that you want to tessellate.

2. Choose **Model > Tesselate**.

3. Choose **Vertex to Center**, **Mid-Edge to Center**, or **Mid-Edge to Mid-Edge** from the submenu.

Decimating an Object

Decimating an object removes information, such as a percentage or number of vertices. Decimating is an easy way to simplify imported models that contain more detail than you need.

TIP! To get the best results when you're decimating an object, alternate between decimating and triangulating the object.

To decimate an object:

1. Select the object or portion of an object that you want to decimate.

2. Choose **Model > Decimate**. The Decimate dialog appears.

3. Set the following options:

    - **Decimation**: Adjust the percentage to specify the percentage of vertices to be removed. The higher the number, the more vertices will be removed.

    - **Decimate by**: Choose **Vertex count** to remove the percentage of vertices specified in the Decimation field. Choose **Fidelity** to remove the required number of vertices to meet the **Criterion** setting, up to the threshold percentage.

    - **Criterion**: Choose **Distance** to remove vertices based on their distance from the planes shared by their neighbors. Choose **Angle** to remove vertices based on the angles formed by the polygons that connect the vertices to their neighbors.

4. Click **OK**.
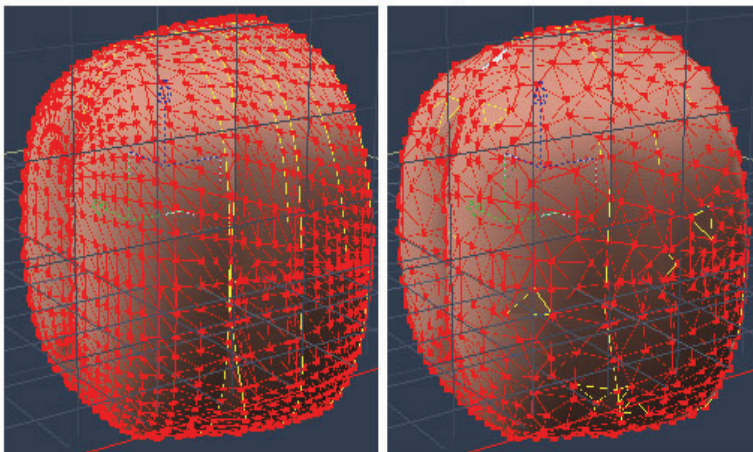
Triangulating and Untriangulating Polygons

The Triangulate command divides selected polygons with more than three sides into several triangular polygons, without affecting the number of vertices.

TIP! Triangulate can also be used in conjunction with the Decimate command to produce more satisfactory results.

NOTE! Triangulating a three-sided polygon has no effect, but you can subdivide a triangle.
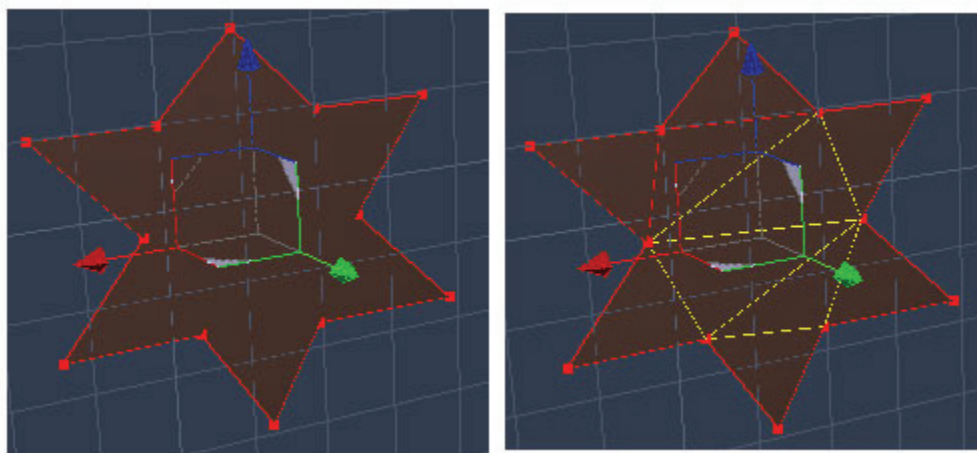
To triangulate selected polygons:

1. Select the polygons you want to triangulate.

2. Choose **Model > Triangulate Polygon**.

To untriangulate selected polygons:

1. Select the polygons you want to untriangulate.

2. Choose **Model > Untriangulate Polygon**.

Cleaning Topology

In some cases, after a long modeling process, complex objects may contain incorrect topologies that can create bad display when rendering. A set of tools can be used to fix these type of problems. These tools are located in the Model menu and work on the current selection.

Triangulating Non-Planar Faces

Non-planar faces are four-sided polygons that do not have all four vertices aligned on the same plane. They often present problems when rendered, because you can see the individual triangles in the polygon and half of the polygon can appear darkened. The Triangulate Non-Planar Faces command detects non-planar faces in a selection and automatically converts non-planar quad polygons into two triangular polygons.

To triangulate non planar faces:

1. Select some or all of the polygons in the model that you want to repair.

2. Choose **Model > Triangulate non planar faces**.
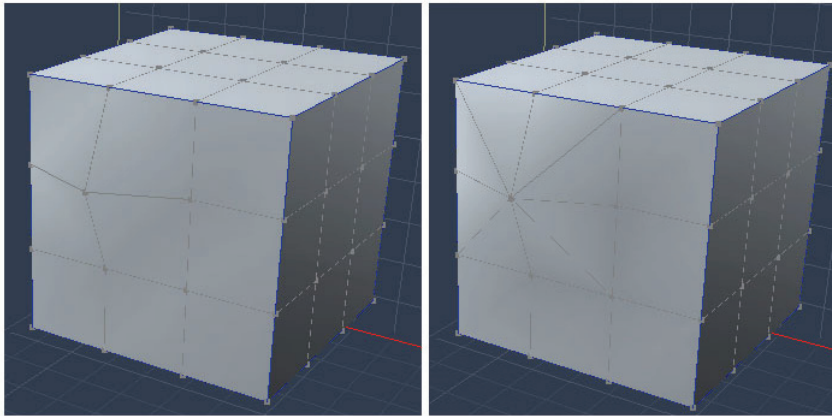
Triangulating N-Gons

N-gons generally refer to polygons that have more than four sides. Because they might not render properly, it is generally best to use 3 or 4-sided polygons in your models as a result.

The Triangulate N-Gons command detects polygons that have more than four sides, and converts them to multiple triangles.

To triangulate n-gons:

1. Select some or all of the polygons in the model that you want to repair.

2. Choose **Model > Triangulate n-gons (>4 points)**.
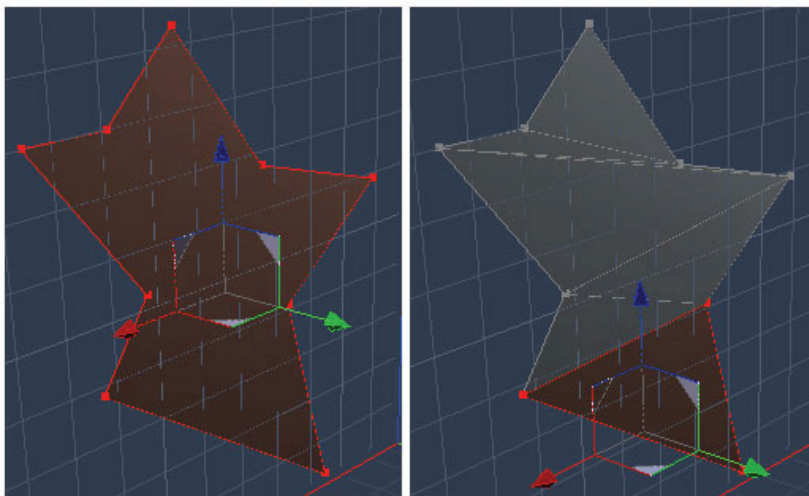
Removing Duplicate Points

Duplicate points are two or more vertices that occupy the exact same position. You can economize on the size of your model by removing them with the Remove Duplicate Points command.

To remove duplicate points:

1.   Select some or all of the polygons in the model that you want to repair.

2.   Choose **Model > Remove Duplicate Points**.

Merging Coplanar Faces

Coplanar faces are facets that share an edge and are in the same plane. You can merge coplanar faces with the Merge Coplanar Faces command.

To merge coplanar faces:

1.   Select some or all of the polygons in the model that you want to repair.

2.   Choose **Model > Merge Coplanar Faces**.

Creating and Filling Holes in an Object

Vertex objects do not have to be completely solid. You can cut holes in an object by emptying polygons on the object's surface, or you can detach polygons to create an effect such as a can with a half-open lid.

NOTE! You can also cut away parts of an object by using the operation. For more information refer to "Using Boolean Operations."
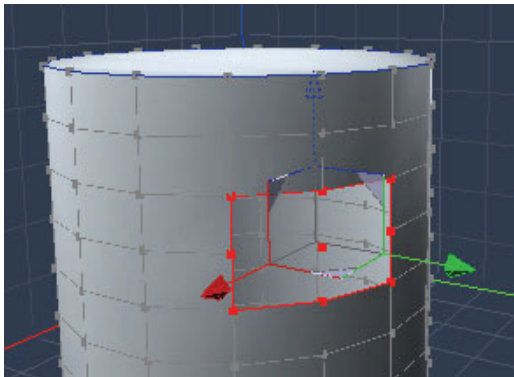
To create a hole in an object:

1.   Select the polygons you want to remove to create the hole.

2.   Choose **Model > Empty Polygon**.

3.   If the process leaves isolated vertices behind, select the stray vertices and delete them.

TIP! You can also empty polygons with the Delete tool.

NOTE! Filled and emptied polygons appear the same in the Wireframe views. To view the hole, switch to one of the shaded modes.

To fill a hole in an object:

1.   Select the closed polyline that you want to fill.

2.   Choose Model > **Fill Polygon**.

Closing a Curve

The Close Curve command located in the Model menu can be used to close an open curve in a single click.

To close curves:

1. Use Edge selection mode to select an open curve in the Vertex modeler.

2. Choose **Model > Close Curve**. The open ends of the open curve are joined into a closed curve.
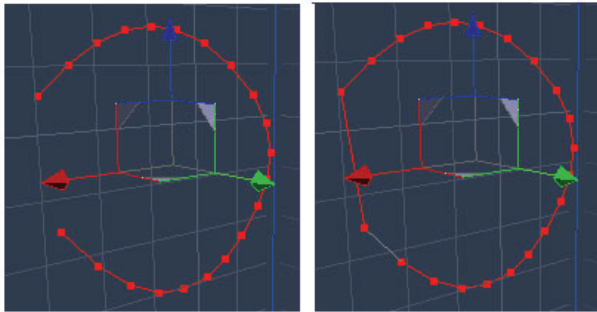
Extracting Edges from Curves

The Extract Curve tool located in the Model menu allows you to create a curve by transforming a selection of edge into a curve.

To extract curves:

1. Select one or more edges in an existing 2D or 3D vertex object.

2. Choose **Model > Extract Curve**. Carrara turns the selected edges into a new curve that is independent of the original object. The edges that you originally selected remain intact in the original curve.
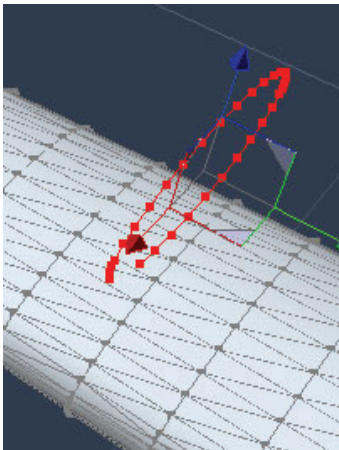
Modifying the Polygon Normals

Some situations require the polygons to be oriented in a certain way. This is the case with models that are exported to some other software that use this information for display purposes. It is also the case with

the Non-Photorealistic Renderer. The polygons oriented toward the camera are rendered whereas the others are not.

In the Vertex modeler, switching the orientation of one polygon will also switch all the adjacent polygons so that the mesh stays consistent.

To switch the normals orientation on a mesh:

1.   Select one or more polygons on the mesh.

2.   Use the menu **Model >Reverse Polygon Normals**.

Flattening Objects

The Flatten command allows you to flatten selected objects or vertices to the plane that lies closest to the selected vertices.

To flatten selections:

1.   Select the objects or vertices you want to flatten.

2.   Choose > **Model > Flatten**.
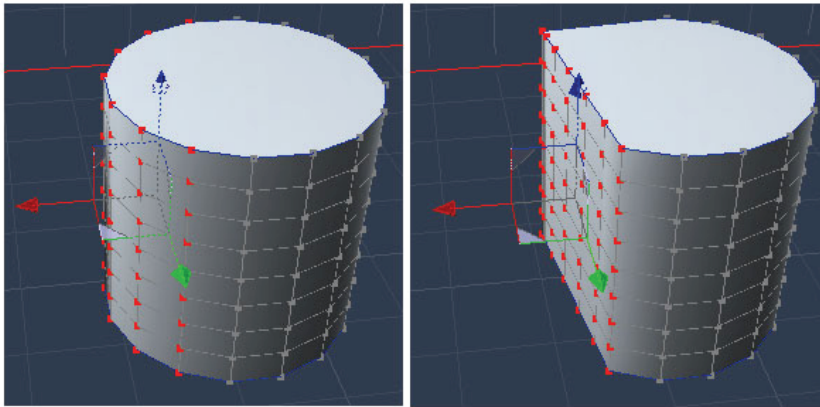
Adding Thickness to an Object

The Add Thickness command lets you extrude polygons at right angles to the plane of the surrounding polygons.
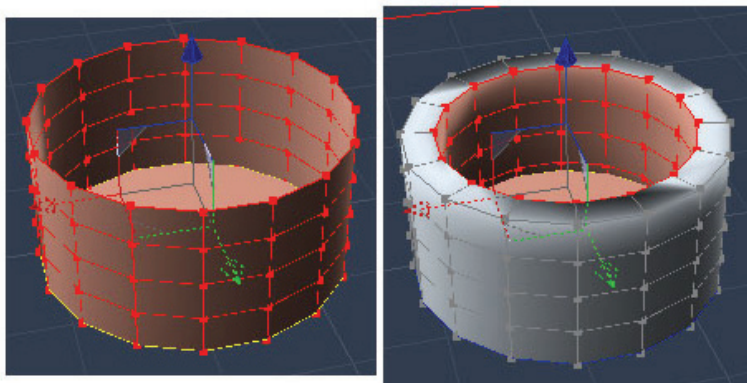
To add thickness to an object:

1. Select all or some of the polygons on the object you want to extrude. In this case the entire object was selected.

2. Choose **Model > Add Thickness**. The Add Thickness dialog appears.
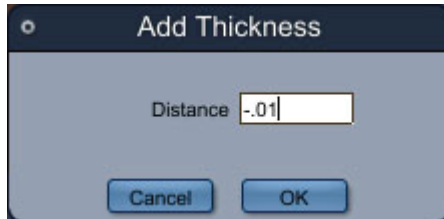
3. In the **Distance** field, specify how thick you want the surface to be:

   - Enter a positive value to add thickness toward the outside of the object.

   - Enter a negative value to add thickness toward the inside of the object.

4. Click **OK**. Thickness is added to the selected polygons.

Offsetting an Object's Surface

When you offset the surface of an object, the surface expands like a balloon. All selected vertices are moved outward at right angles to the average plane of the surrounding polygons
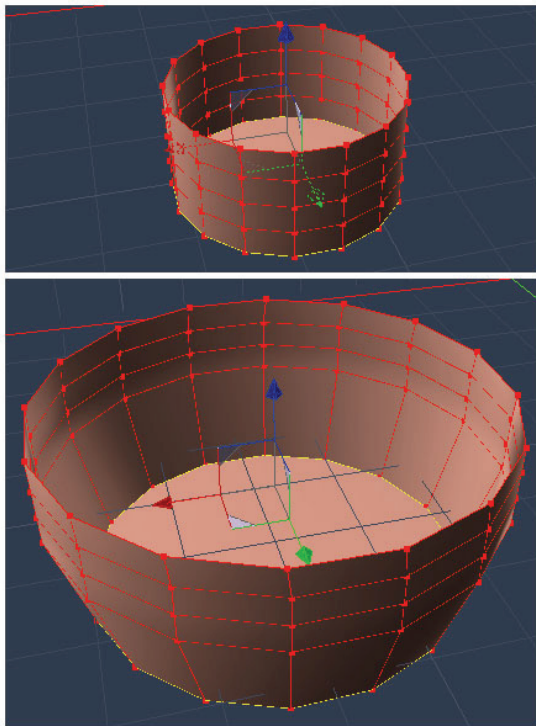
To Offset the surface of an object:

1. Select the polygons that you want to offset from the object.

2. Choose **Model > Offset Surface**. The Offset Surface dialog appears.
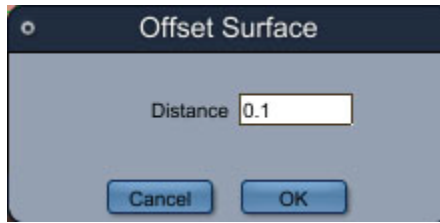
3. In the **Distance** field, specify the amount you want to move the selected polygons.

4. Click **OK**.

Filleting Vertices or Edges

Most real objects seldom have truly sharp edges. An example is right in front of you: look at your computer screen, whose shell probably has rounded edges, whether for safety or design or both. To get this effect, use the Fillet tool to create smooth angles and corners, by rounding chosen edges or vertices of a polymesh.

The Fillet tool has two properties to set: Radius and Range.

To fillet selected edges:

1. Select one or several edges on a surface or a volume.

2. Do one of the following:

    * Select the **Fillet tool**.

    * Choose **Model > Fillet**.

3. Set the following properties as desired:

    * **Radius:** Defines the size of the rounded section that makes the edge or vertex rounded.

    * **Range:** Defines the number of sections that will be created to form the rounded edge. Lower numbers result in fewer polygons, while higher numbers provide higher quality.

4. If using the Fillet tool, press the **Enter/Return** key to complete the process.
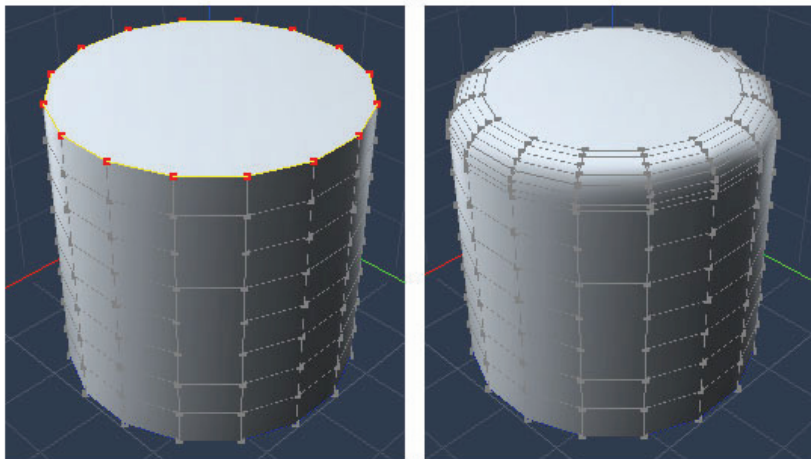
To fillet unselected edges:

1. Select the Fillet tool.

2. Do one of the following:

   - To fillet a single edge, click on the desired edge to complete the fillet.

   - To fillet multiple edges, Shift-click to select multiple adjacent edges. Then release the Shift key and click on the selection to complete the fillet

3. Set the following properties as desired.

   - **Radius:** Defines the size of the rounded section that makes the edge or vertex rounded.

   - **Range:** Defines the number of sections that will be created to form the rounded edge. Lower numbers result in fewer polygons, while higher numbers provide higher quality.

4. Press the Enter/Return key to complete the process.
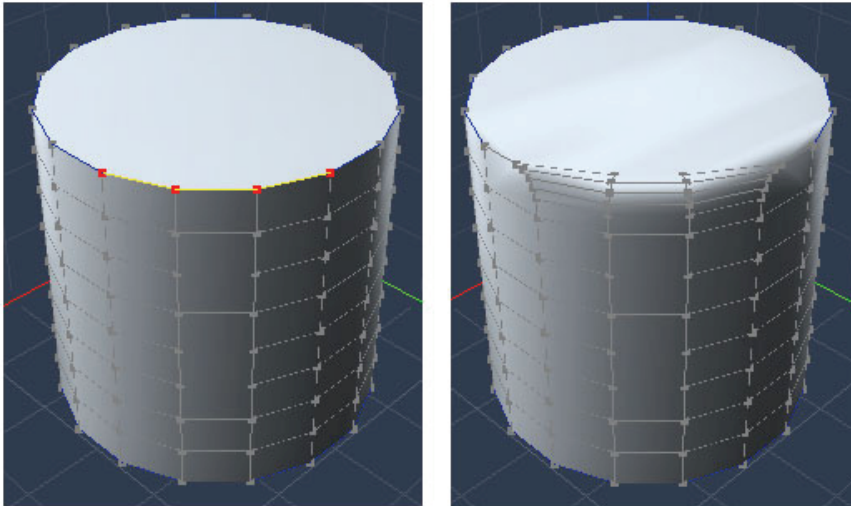
To fillet rounded corners from vertices:

1. Select the Fillet tool.

2. Do one of the following:

   - To fillet a single vertex, click on the desired vertex to complete the fillet.

   - To fillet multiple vertices, **Shift-click** to select multiple adjacent vertices. Then release the **Shift** key and click on the selection to complete the fillet

3. Set the following properties as desired.

   - **Radius:** Defines the size of the rounded section that makes the edge or vertex rounded.

   - **Range:** Defines the number of sections that will be created to form the rounded edge. Lower numbers result in fewer polygons, while higher numbers provide higher quality.

4. Press the **Enter/Return** key to complete the process.
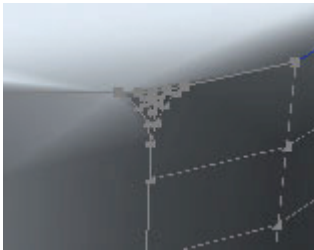
Using the Bridge Tool

The Bridge tool allows you to connect selected polygons between two objects in the Vertex modeler.

To use the tool:

1. Select the polygons, edges, or vertices that you want to connect.

2. Select the Bridge tool.

The following options appear in the tray:
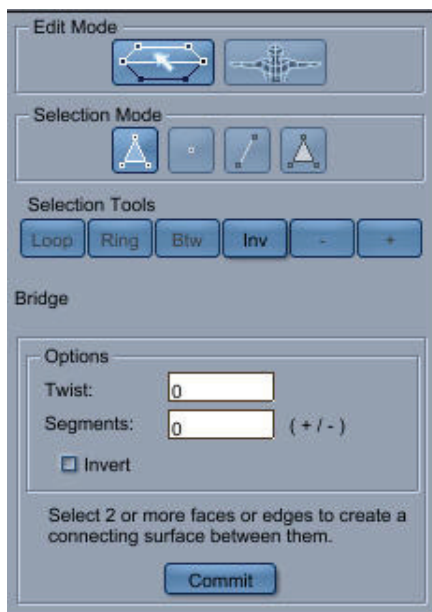
3. In the Options section, choose any of the following options:

    o **Twist** specifies how many times the bridge should twist before it connects to the other side.

    o **Segments** defines the number of segments in the bridge. The default setting is 0, which adds no additional segments to the bridge.

    o Check the **Invert** option to reverse the direction of the twist.

4. Click the **Commit** button to complete the bridging operation.

Working With Subdivision Surfaces

When an object is modeled using a small number of polygons, its silhouette is often very rough and it is difficult to create smooth surfaces. If you use more polygons, it becomes more difficult to make changes to your model, due to the higher number of vertices.
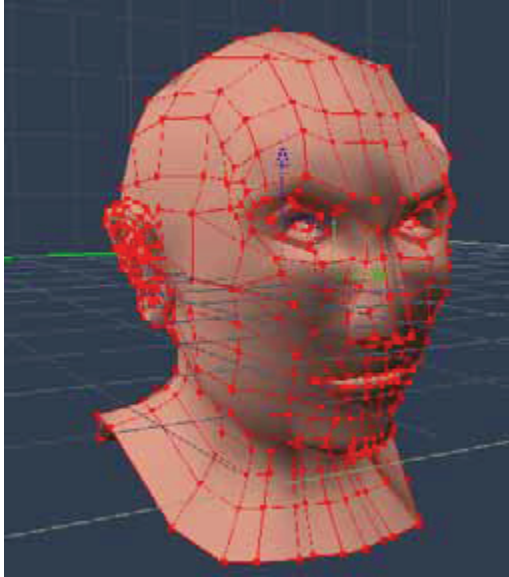
When you subdivide the surfaces of your model, you can continue to model a low-polygon model while viewing the result as if there are more polygons. Subdivision surfaces help you create smooth surfaces without increasing the number of control points of your model. With subdivision surfaces you can see, in real time, your final mesh as you edit its control points. The image below shows the same model as above but with subdivision surfaces set to 4.
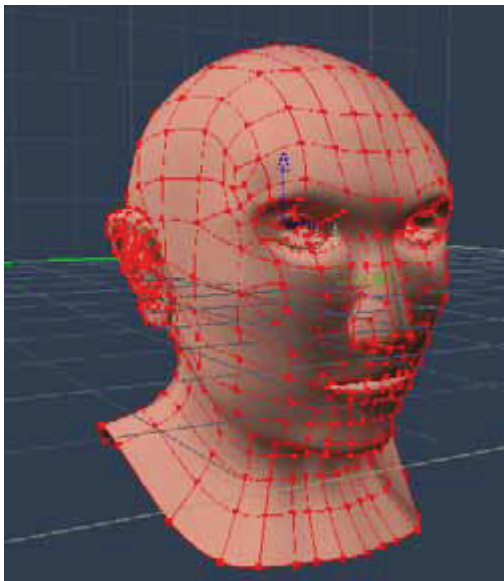
The Subdivision Surfaces can be applied to either a closed or an open polymesh. On an open polymesh, the resulting surface is smoothed along its edge and no distortions are created in respect to the rest of the surface.
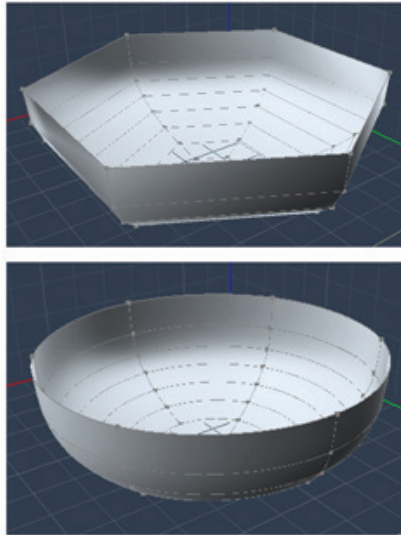
Subdividing a Polymesh

Subdivision is done in the model room after you select any portion of the model.

To use subdivision surfaces:

1. In the Assemble room, do one of the following to display the desired model in the Model room:

    - Double-click one of the facets on the polymesh you want to smooth.

    - Select the desired object in the Instances tab and switch to the Model room.

2. Create a selection on the object to display options in the **Model tab: Selection panel**.
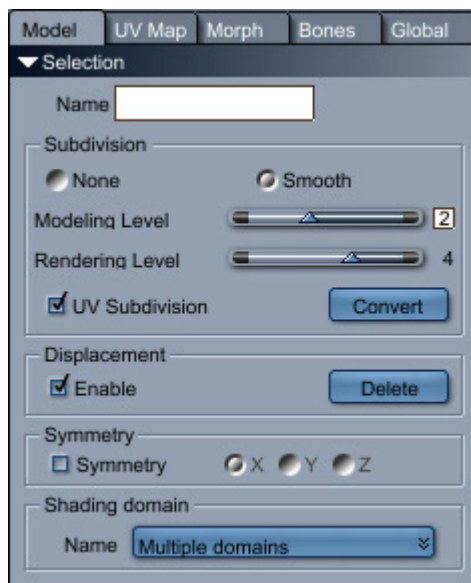
3. In the Subdivision section, adjust the following settings as necessary:

    - **None**: Choose this option to turn off subdivision smoothing.

    - **Smooth**: Choose this option to enable subdivision smoothing.

- **Modeling Level**: Increase the modeling level until you add the desired amount of smoothing to your low-resolution mesh.

NOTE! A high subdivision level can slow down modeling operations, so you should keep the level low while modeling and only turn it up once your object is finished.

- **Rendering Level**: Increase the rendering level to achieve the desired amount of smoothing during rendering.

NOTE! You can set the rendering level higher than the modeling level.

- **UV Subdivision**: TBD

- **Convert**: Click this button to convert the model to the number of polygons that are required to achieve the amount of detail in the current modeling level setting. Each incremental increase in the modeling level quadruples the number of polygons from the next lowest setting. For example, a model that contains 400 polygons would increase to 1600 polygons at Modeling Level 1, 6400 polygons at Level 2, and 25600 polygons at Level 3.

4. **Displacement**: TBD

5. **Symmetry**: TBD

6. **Shading Domain**: TBD

Edge Creasing

In the process of subdividing a polymesh, the smoothing of the edges has a primary importance. Creased edges will still display a sharp angle after the subdivision is applied.

When an object such as a sphere is rendered, the individual polygons are shaded to appear smooth. In some cases, you do not want smooth shading:you might want an edge to represent a crease in the object's surface, such as an edge on an auto body. A crease separates regions that are flat or appear smoothly curved.
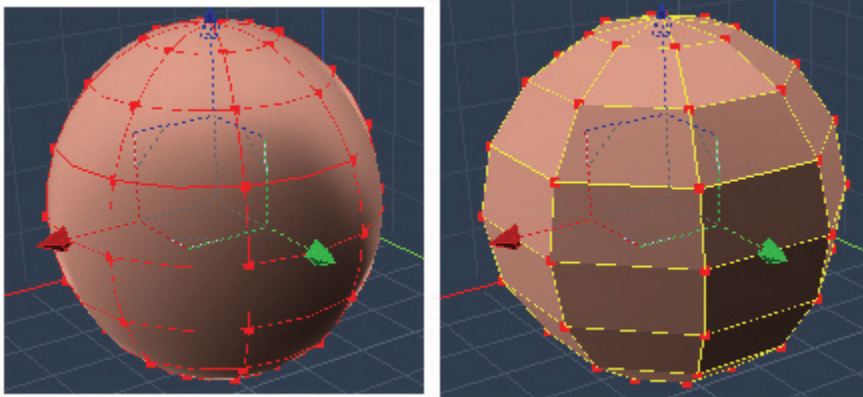
Following is an example of a Subdivision applied to a cube with none, some or all of its edges smoothed. The resulting meshes are radically different and underline the importance of this parameter in the Subdivision Surfaces.
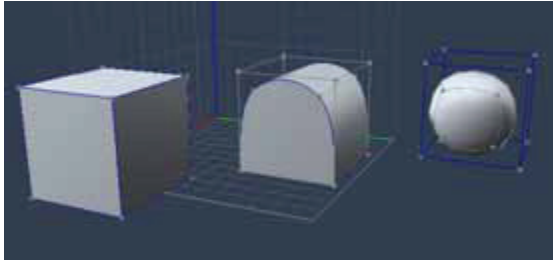
The Vertex modeler provides two different ways to smooth or crease edges on an object's surface:

- Use the **Model > Smooth Edges** or **Model > Crease Edges** command to smooth or crease selected parts of an object.

- Use the **Crease tool** to crease an edge so that it remains creased, regardless of how you redefine the object through subdivision.
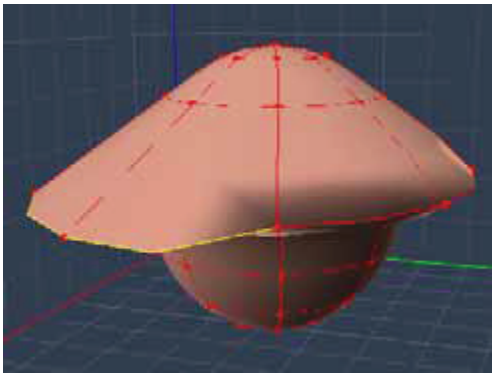
Creasing an edge will prevent it from being rendered as smooth or curved. In this example, the Crease tool was used to create the sharp ridges on the left side of the smoothed sphere.

To smooth or crease selected edges:

1. Select the edges you want to be rendered as smooth or creased. Creased edges are highlighted in yellow when selected.

2. Do one of the following:

   - To smooth edges, choose **Model > Smooth Edges.** The Smooth Selected Edges dialog appears.

   - To crease edges, choose **Model > Crease Edges.** The Crease Selected Edges dialog appears.

3. Enter a value in degrees for the crease angle.

   - For smoothing, any edges that meet at an angle greater than the crease angle will be rendered as smooth; any edges that meet at a lesser angle will be rendered as creased.

   - For creasing, any edges that meet at an angle less than the crease angle will be rendered as sharp; any edges that meet at a larger angle will be rendered as smooth.

4. Click **OK**.

To crease an edge:

1. Click the Crease tool.

2.   Select all edges you want to be rendered as straight. Creased edges will be highlighted in yellow when selected, in blue when deselected.

To uncrease an edge:

1.   Click the Crease tool.

2.   **Command/Ctrl** +click all creased edges that you want to uncrease.

Symmetrical Modeling

When working with polymeshes in the Vertex modeler, a new Symmetry section in the Model tab allows you to make symmetrical modeling changes to your meshes.
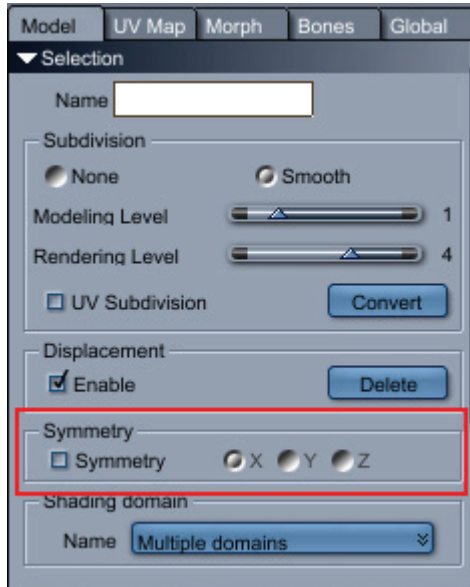
To use symmetrical modeling:

1.   Create a selection on the model that you want to model symmetrically.

2.   Check the **Symmetry** option to enable Symmetrical modeling.

3.   Select the axis of symmetry (X, Y, or Z) to display the correct symmetrical plane.: X (front to back), Y (side to side) or Z (top to bottom).

4.   Make the desired changes to your model. When Symmetrical modeling is enabled, any changes you make on one side of the axis will be repeated on the other side.

5.   When your symmetrical modeling is completed, uncheck the **Symmetry** option to resume normal modeling.

Displacement Painting (Pro only)

Carrara 6 includes a new displacement modeling feature in the Vertex modeler. This feature allows you to increase the detail in your model without increasing the geometry. The typical workflow is to take a model and paint your displacement using the Displacement Painting tool. After the displacement painting is done, you save the displacement map, which you later apply to the original model to restore the detail.

Showing, Hiding and Deleting Displacement

By default, Carrara enables the Displacement option so that you can see changes you make while you paint with the Displacement Painting tool. You can enable or disable this display while you work so that you can compare the base model to the displaced model.

To enable displacement:

1. Select a model and open it in the Vertex modeler.

2. Create a selection on the model to display the options in the Model tab: Selection panel.

3. Do one of the following:

    •   To enable displacement, check the **Enable** option in the Displacement section.

    •   To turn displacement display off, uncheck the **Enable** option in the Displacement section.

    •   To completely remove the displacement painting from the model, click the **Delete** button.

Using the Displacement Painting Tool

When you use displacement painting on a subdivided mesh, you can increase the number of subdivisions to increase the detail of the displacement painting. Checking the Symmetry option when available also applies the displacement symmetrically along the axis that you select.

The Displacement Painting tool allows you to control the size and shape of the brush as you paint. There are also several different alpha images that can be used to add texture to your brush strokes. For example, the preceding figure shows the results of the Crackles alpha image being used to paint the hair, and the Horns alpha image used for the beard and eyebrows.

To paint with the Displacement Painting tool:

1. Select the model that you want to change.

2. To enable the Selection options in the Model tab, switch to the Model room and make a selection using one of the following methods:

    •   Choose **Selection > Select By > Name** to select a polymesh, polygon, edge, or vertex selection to which you have previously assigned a name.

- Choose **Selection > Select By > Shading Domain** to select all polygons that appear in a shading domain. Repeat the command to add to the selection.

- Use the selection tools in the Properties tray to create a selection.
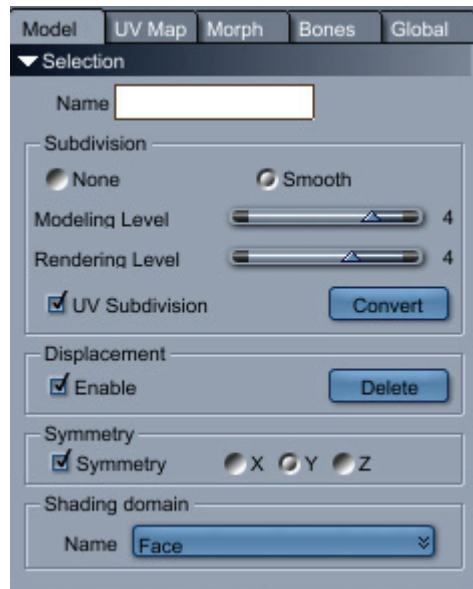
3. Adjust the following settings in the Selection panel:

   - Use the **Name** field to assign a name to the current selection, if desired.

   - In the Subdivision section, select the **Smooth** option to enable subdivision smoothing.

   - Set **Modeling Level** to a high enough value to achieve the desired amount of detail while you paint your model.

   - Set **Rendering Level** equal to or greater than the Modeling Level so that you can view a rendered version of your displacement map on occasion.

   - Check the **UV Subdivision** option if you want to smooth the UVs with the surface. When checked, the UVs will be much smoother, but in some cases they may appear to slide over the surface. When not checked, the UVs of the original vertices will remain unchanged. They will follow the deformation of the original surface but may cause some perturbation in the UV map.

   - Click the **Convert** button to convert the model from the original low-polygon mesh to a higher polygon mesh based on the subdivision settings. For example, if the Subdivision Modeling Level is set to 2 before conversion, the converted model will have 16 times the number of polygons after conversion. Note also that converting the model will also make the displacement painting a permanent part of the mesh. You will not be able to delete displacement painting after a model is converted.

   - In the Displacement section, check the **Enable** option to display existing or new displacement while you paint. While in this mode, every transformation of the mesh (including point translations) is considered as displacement, so the original mesh is not modified. However, if a transformation is applied to the whole mesh (such as moving), the transformation is applied to the original mesh.

   - Click the **Delete** button to remove existing displacement painting from the model.

   - If you want to paint symmetrically, enable the **Symmetry** option and choose X (front to back), Y (left to right) or Z (top to bottom) symmetry plane.

4. Click the Displacement Painting tool in the toolbar.
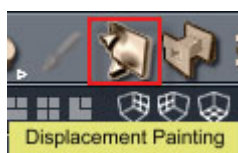
The options appear in the Properties tray.
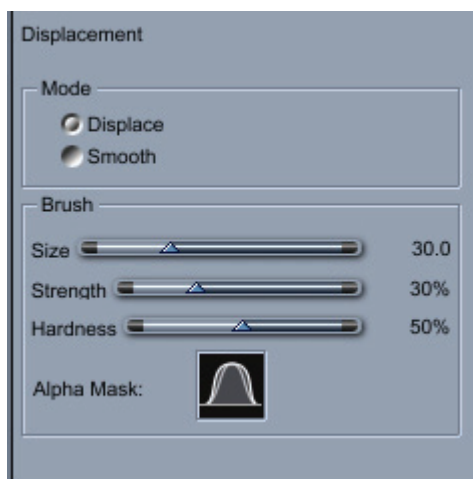
5. Choose a mode for painting:

    - Choose **Displace** to paint raised or lowered areas onto the mesh. Paint normally to raise areas, and press the Shift key while painting to lower areas.

    - Choose **Smooth** to soften the areas between the raised and lowered areas, to blend them together.

6. Set parameters for the brush as follows:

    - Increase or decrease the **Size** to displace wider or finer areas of the mesh.

    - Adjust the **Strength** of the brush to increase or decrease the amount of displacement you can achieve in a single stroke.

    - Increase or decrease the **Hardness** of the brush to create a brush that has less or more falloff around the edge of the brush.

    - Click the **Alpha Mask** button to select an alpha mask image that will be used to texture the brush.
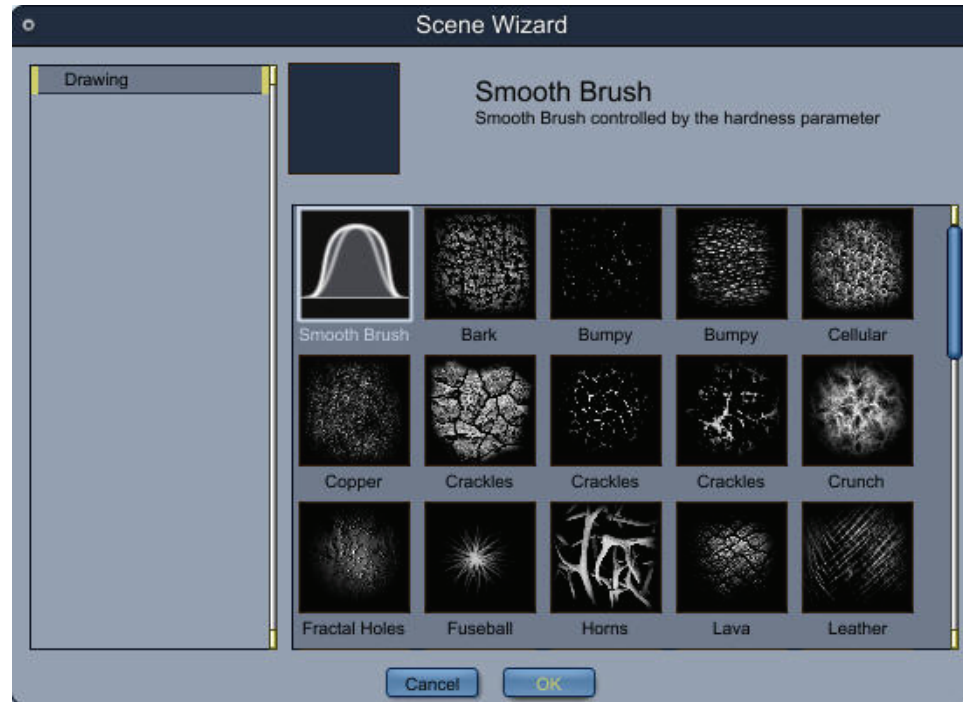
7. Paint on your model in the areas that you want to displace. Adjust brush size and strength while painting to achieve desired results. Press the **Shift** key while painting to displace the model inward instead of outward.

Exporting Your Displacement Map

After you have finished painting your displacement map, you can export the map for use on a similar model with the same UV maps and shading domains.

Carrara allows you to generate displacement maps for each shading domain in your model, each of which can be a different size. The displacement maps are calculated relative to the UV coordinates.

NOTE! You will need to create multiple maps if your object contains UVs that are overlapping.

The Export Displacement Map dialog allows you to select a subdivision level, texture map size, and shading domain, and file location.

To export displacement maps:

1. Displacement maps are calculated relative to the UV coordinates when they are created and exported. From the Model room, select the UV Map tab in the Properties tray. Verify that the Mapping option is set to Custom.. Any other UV mode will result in unexpected results.

2. Choose **Selection > Export Displacement Map**. The Export Displacement Maps dialog appears.
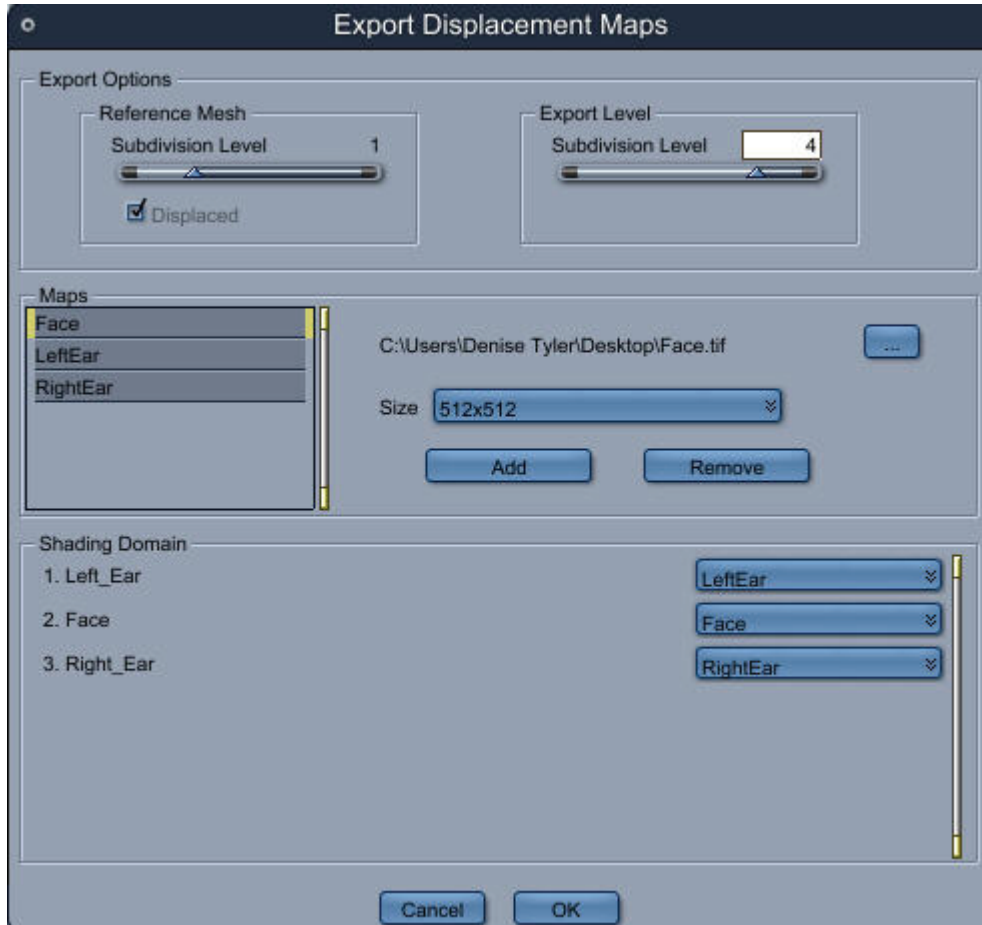
3. In the **Export Options** section, set the following options to determine the amount of displacement to be exported in the maps. Carrara outputs a grayscale image that increases the subdivision level based on the difference between the reference mesh and the export level. For example, if you have your Reference Mesh Subdivision Level at 0 and your Export Subdivision Level at 4, the resulting map will produce a 4-level change. If your Reference Mesh Subdivision Level is set to 1, and your Export Subdivision Level is set to 3, the resulting map will produce a 2-level change:

    -   **Reference Mesh Subdivision Level**: Set this value to the subdivision level that will be set for your geometry.

    -   **Displaced**: This option determines whether the displacement is calculated relative to the original mesh, instead of being relative to the selected reference level or relative to the displaced mesh. For example, if you want to export a displacement map for a DAZ or Poser figure, make sure that the option is unchecked to prevent changes to the number of polygons of the original mesh. If you are modeling your own object, it may be beneficial to check this option to obtain maximum precision on the final object when you export the base mesh with the displacement included.

- **Export Subdivision Level**: Set this value to the subdivision level that you want to arrive at by adding the displacement map. Normally, this will be equal to the level of subdivision that you used while painting the map.

4. In the **Maps** section, you can create a displacement map for one or more shading domains in your model. Sizes can range from 64-pixel to 2048-pixel square texture maps. Do the following for each displacement map that you want to create and assign:

   - Click the **Add** button to choose a folder and assign a name for the displacement map in the **Export Displacement Maps** dialog.

   - Choose a file type from the Save as Type list. You have the option to save in 16-bit TIF format as well as other 8-bit image formats.

   - Click **Save** to store the information.

   - Choose the desired texture map size from the **Size** popup menu.

5. In the **Shading Domain** section, click the **texture button** beside each shading domain that will use one of the displacement maps that you created in step 3. Choose the displacement map from the menu that appears.

6. Click **OK** to export your displacement map or maps to the locations that you specified in step 3.

7. To assign the displacement maps to an object, see "Displacement Mapping."

Using Morph Targets (Standard, Pro)

Morph targets are a powerful animation tool that allow you to define deformations to a Vertex Primitive and then animate the morph between the original object and the deformed version. For example, on a model of a face, you might define a morph target for the eyes blinking, another for the jaw opening, and another for the tongue moving. Then, when animating speech, it is easy to add all of these deformations in turn to provide realistic facial motion, without having to move the vertices by hand each time.
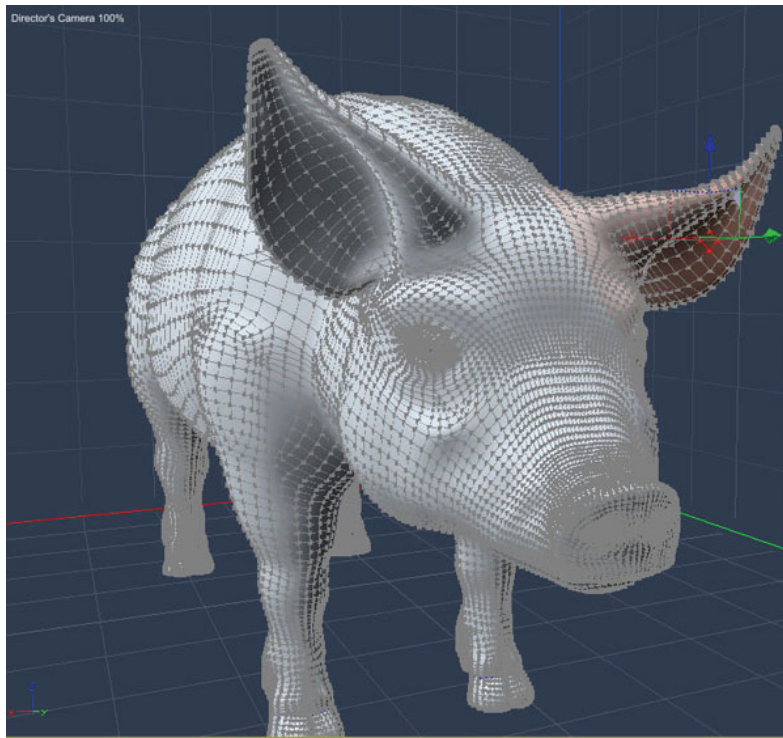
Defining a Morph Area

The set of vertices on which Morph targets are defined is known as the Morph Area. You can create any number of Morph Areas, each with its own collection of Morph targets.

To create a Morph Area:

1. Open the model in the Vertex modeler.

2. Click the **Properties tray: Morph tab**.
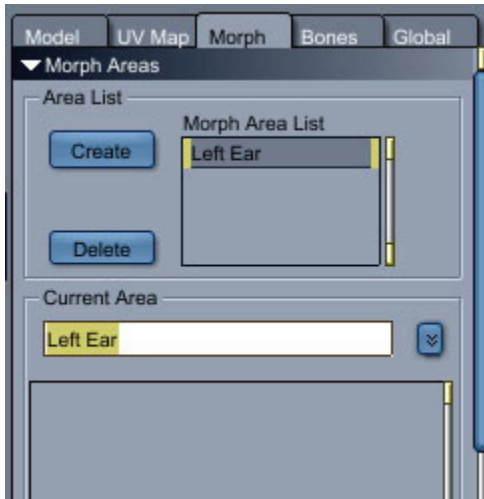
3. Select the vertices of the object to be included in the Morph Area.

4. Click the **Create** button.

5. Assign a name for the morph area in the **Current Area** field and press **Enter**.

To add or remove vertices from the Morph Area:

1. Select the vertices that you want to add to a morph area.

2. Choose **Model > Add to Morph Area**.

3. A dialog prompts you to select the morph area that you want to modify. Highlight the morph area and click **OK**.

To remove vertices from the Morph Area:

1. Select the vertices that you want to remove from a morph area.

2. Choose **Model > Remove from Morph Area**.

3. If you have selected to remove all vertices from a morph area, a dialog informs you that some morph areas are not used any more and asks if you want to remove them. Choose **Yes** to remove the unused morph areas. A dialog prompts you to select the morph area that you want to modify. Highlight the morph area and click **OK**.

Creating Morph Targets

You can create multiple Morph targets on the Morph Area. These are the steps to create and edit a Morph target.

To create a Morph target:

1. In the Model room, open the **Properties tray: Morph tab**. Select one of the Morph Areas that you created. The name of the selected morph area appears in the Current Area field.

2. Press the **Current Area** menu button and choose **Create Target** from the pop-up menu.
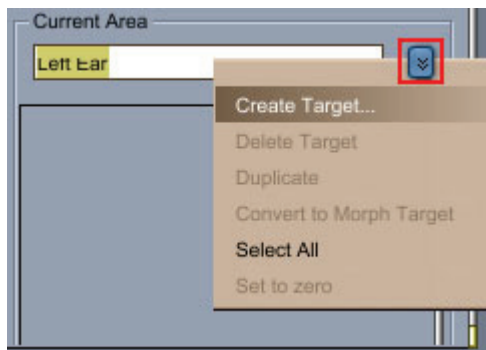
3. Enter a name for the morph in the **New Target Name** dialog and click **OK**. The new Morph target, with slider bar and Edit button, appears in the list.

Editing Morph Targets

After you create a morph area and add a morph target to it, you can modify the shape of the geometry to achieve the desired morphed shape.

To edit a Morph target:

1. Click the Edit button next to the Morph target's slider. The edges in the target being edited turn green while you are editing a Morph target.
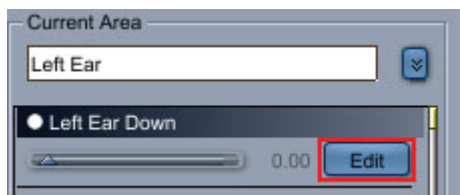
2. Move or rotate the vertices to their new positions. Note that you cannot edit vertices outside the current Morph Area while editing the Morph target.
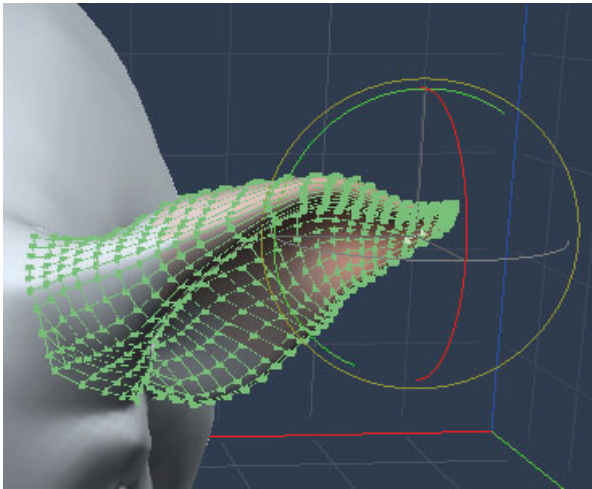
3. After the morph is complete, click the Valid button in the Properties tray: Morph tab: Morph Target panel. This defines the Morph target deformation.

Animating Morph Targets in the Model Room

Morph targets can be animated from either the Modeling room or the Assembly room. In both cases, the Morph target sliders appear in the Properties tray and can be moved from 0 to 1 to control the degree to which the Morph target is applied. A value of 0 results in no change from the original mesh, while a value of 1 results in the Morph target being fully applied to the mesh. Changing a slider value inserts a keyframe with the new value at the current time on the Timeline. See "Animation Techniques."

The Modeling room has two modes:
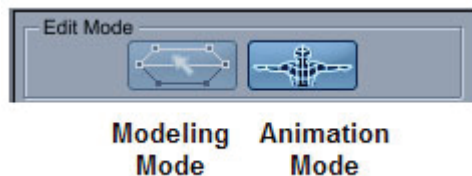
-   In **Modeling Mode**, Morph targets can be edited, but when you're not editing a Morph target, the mesh is displayed without any deformations applied. This mode lets you define multiple Morph targets, starting with a non-deformed mesh each time.

-   In **Animation Mode**, you can see the effects of Skinning and Morph targets on your mesh, allowing you to animate them conveniently.

To animate Morph targets in the Modeling room:

1. Switch to Animation Mode by clicking the Animation button at the top of the Properties tray.

2. Click on the Morph tab in the Properties tray.

3. Select the Morph Area whose target you want to adjust from the list appearing under the Morph tab.

4. Move the Morph target slider to the desired value.

Animating Morph Targets in the Assemble Room

It's generally most effective to animate Morph targets from the Assembly room, where you can see the changes to the Morph Area in context with the rest of your scene.

To animate Morph targets in the Assembly room:

1. Click on the Vertex object to be animated.

2. Select the Properties tray: General tab.

3. Select the Morph Area whose target you want to adjust from the list appearing in the Vertex object section of the Properties tray.

4. Move the desired Morph target slider or sliders to the desired value.
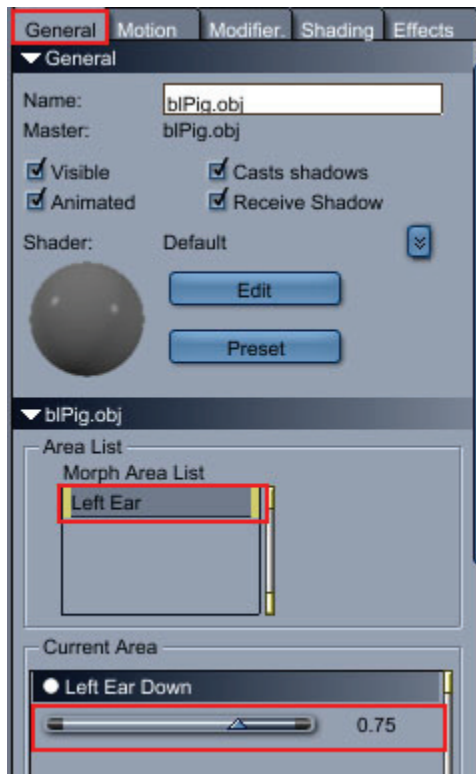
Using Morph Targets on DAZ Studio/Poser Figures

Many DAZ Studio and Poser figures are provided with a multitude of morphs that are used to modify the appearance and expression of the figures. You can create your own morph targets in addition to those that are provided. The process is very similar to those discussed in the previous sections, except that you can also export the morph target for use in applications such as DAZ Studio or Poser.

Creating Figure Morph Targets

To create a new morph target on a DAZ/Poser figure:

1. From the Content tab, double-click the icon of the selected DAZ or Poser figure to load it into your scene in its default position.

2. Select the figure and switch to the Model room. If you want to create symmetrical morphs, make a selection on the model. Check the **Symmetry** option in the **Properties tray: Model tab: Selection panel**, and select the X axis for symmetry.

3. Click the Morph tab to see a list of existing morphs in the Figure Morphs panel.
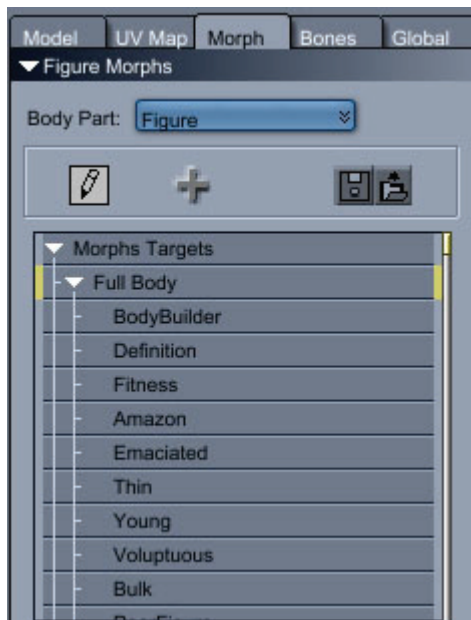
4. From the **Body Part** list select Figure if you want to create a full body morph. Otherwise, select the body part that you want to morph from the Body Part list.

5. If the selected body part has morphs that are arranged in multiple groups, select the group that you want the new morph to appear in.

6. Click the **Create .New Morph Target in Selected Group** button. The New Target Name dialog appears.

7. Enter a name for the new morph and click OK to continue.

8. The new morph name appears at the bottom of the list in the morph group that you selected. Select the new morph name.

9. Click the Edit Selected Morph Target button. The vertices associated with the selected body part turn green.
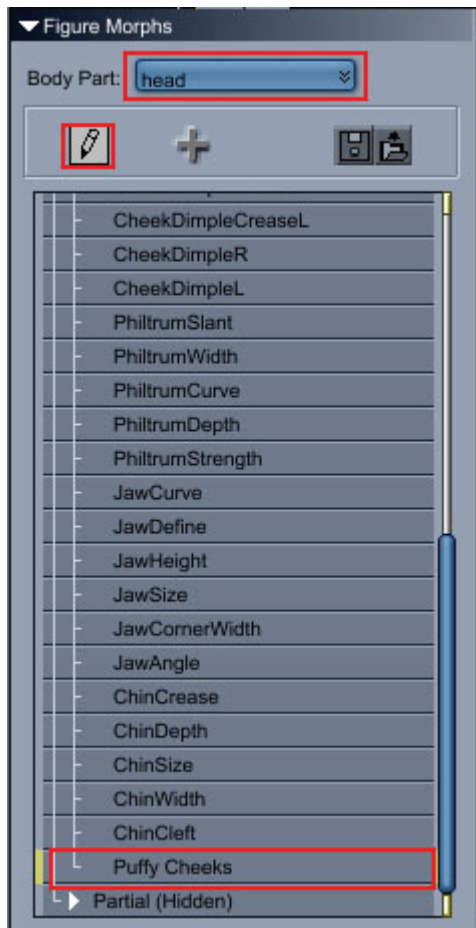
10. Select the vertices that you want to modify, and use the Move or Rotate tools to modify their positions.

11. When your changes are complete, click the Validate Changes to Morph Target button.
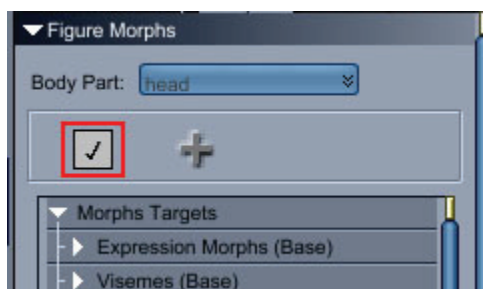
Editing Figure Morph Targets

To edit a morph target on a DAZ Studio/Poser figure:

1. From the Content tab, double-click the icon of the selected DAZ or Poser figure to load it into your scene in its default position.

2. Select the figure and switch to the Model room.

3. Click the Morph tab to see a list of existing morphs in the Figure Morphs panel.

4. From the **Body Part** list select Figure if you want to modify a full body morph. Otherwise, select the body part that uses the morph that you want to modify.

5. Expand the Morph Target list to select the morph target that you want to modify.

6. Click the **Edit Selected Morph Target** button. The morphed area turns green, and the object morphs to the target morph shape.

7. Select the vertices that you want to modify, and use the Move or Rotate tools to modify their positions.

NOTE! In order for morphs to be compatible with the original figures, you cannot make any changes to the geometry other than changing the positions of existing vertices. If you add or remove vertices to the original geometry, the morphs will not work because the number of vertices must be identical between the original model and the morphed version.

8. When your changes are complete, click the Validate Changes to Morph Target button. The morph is saved, and the geometry reverts to its unmorphed shape.
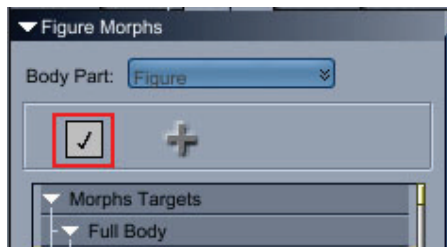
Exporting Figure Morphs

After you create or edit a morph target you can export an OBJ file so that you can apply the same morph to another figure. When Carrara exports the OBJ file it exports the full OBJ file, rather than the selected body part. Later you can reimport this morph and use it in another copy of the same figure.

To export a morph target from Carrara:

1. With the DAZ Studio or Poser figure open in the Model room, click the Morph tab to see a list of existing morphs in the Figure Morphs panel.

2. From the **Body Part** list select the body part for which you want to export the morph.
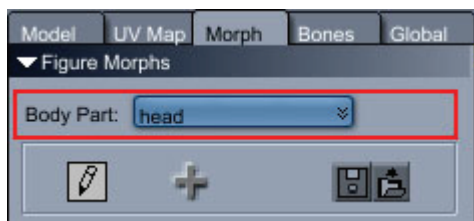
3. Expand the Morph Target list to select the morph target that you want to export.
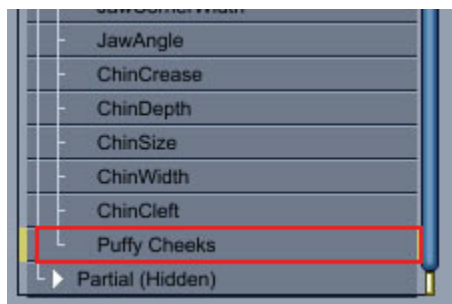
4. Click the **Export Morph Target** button. The Save As dialog appears.
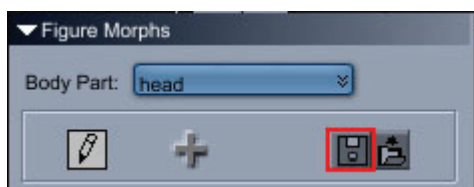
5. Locate the folder to which you want to save the OBJ file, and assign a name to the morph target.

6. Click **Save** to export the complete OBJ file.

Importing Figure Morphs

When you save morphed OBJ files you can later reimport the morphed file as a morph target when you use the same figure again. For example, if you create and save a morph for Victoria 4.1, you can import the morph target the next time you add Victoria 4.1 to a scene.

To import a morph target into another version of the same figure:

1. From the Content tab, double-click the icon of the selected DAZ or Poser figure to load it into your scene in its default position.

2. Open the figure in the Model room.

3. Click the Morph tab to see a list of existing morphs in the Figure Morphs panel.

4. From the **Body Part** list select the body part for which the morph target was originally created.
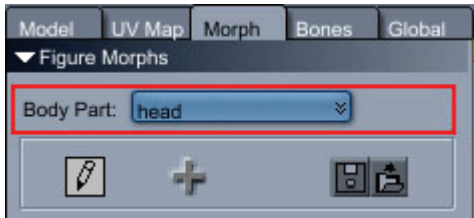
5. Expand the Morph Target list to select the category under which you want to import the morph target.

6. Click the **Import Morph Target** button. The Open dialog appears.

7. Locate the folder that contains the OBJ file for the morph target. Select the file and click **Open**.

8. The New Target Name dialog displays the filename in the Name field. Rename the morph target, if desired, and click **OK**. The morph target appears at the bottom of the morph list in the selected category.

Vertex Modeling Techniques

The Vertex Modeler provides several 2D and 3D objects from which you can begin the modeling process. After you add one or more of these objects, you can use standard modeling techniques to create objects that are more complex.

Carrara's Construction Tools (Standard, Pro)

Carrara provides a number of different construction tools that assist with building objects in the Vertex Modeler.

Extruding Cross-Sections

Extruding or sweeping a polygon creates an object similar to that created with the Spline Modeler. You can use an open or closed polyline to define the cross-section of an extrusion or sweep. Using an open polyline as a cross-section creates a shell with the shape of the polyline.

The cross-section can be a separate polyline object, or a set of selected edges in a larger object.

There are two ways of extruding a cross-section:

- Use "Dynamic Extrusions" to click and drag the cross section..
- Use a "Path Sweep" to sweep polylines along a path.

Dynamic Extrusions

The Dynamic Extrusion tool is a very easy and fast tool to create the rough shape of a model starting from a simple primitive or to add details to a particular part of a model after its creation.

When several polygons or polylines are selected, they are all independently extruded following the extrusion of the last selected one. Very complex models can so be created with a simple click and drag action.

The extrusion method can be chosen among the four available in the tool options of the Properties tray. There are several options that are common to the four extrusion modes.
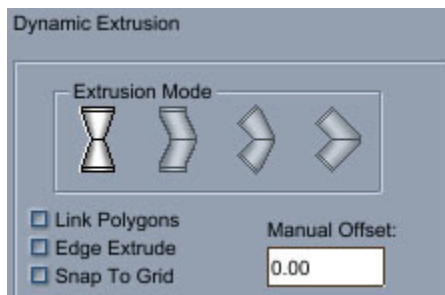
Dynamic Extrusion options in the Properties tray.

- **Link Polygons**: Check this option to keep the adjacent polygons linked together during the extrusion.

- **Edge Extrude**: Check this option to enable Edge Extrusion mode. This allows you to extrude edges into corners.

- **Snap To Grid**: Check this option to snap the extrusions to the grid. See "<u>Changing the Grid</u>" for further information on setting grid preferences.

- **Manual Offset**: If you want to manually enter an offset value for the extrusion, enter it in the Manual Offset field in the tool options of the Properties tray. All the selected polylines will then be offset at this distance.

The following sections provide further information on the four types of dynamic extrusions.

Normal Extrusions

Normal Extrusion creates an extrusion that is perpendicular to the original cross-section. The size of the extrusion can be different than the original cross-section.
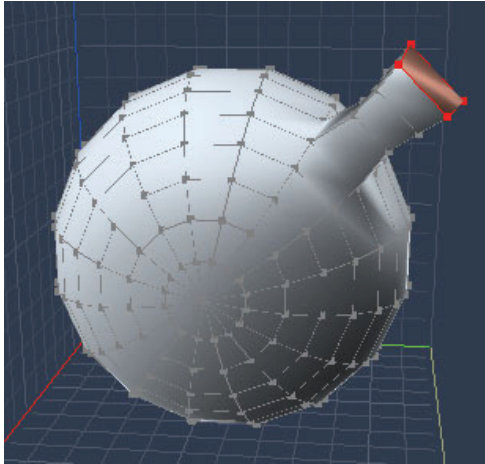
To create a Normal Extrusion:

1. Select the polygon(s) that you want to extrude.

2. Do one of the following:

    - Choose **Construct > Extrude.**

    - Click the Dynamic Extrusion tool.

3. In the Extrusion Mode section of the Properties tray, choose the **Normal Extrusion** mode.

4. Click and drag the polygons or polylines to be extruded.

    - Hold the **Shift** key down during the extrusion to constrain the extrusion to the plane of the original cross-section.

    - Hold the **Command/Ctrl** key down during the extrusion to constrain the size of the extrusion to the size of the original cross-section.

Parallel Extrusions

Parallel Extrusion creates an extruded cross-section parallel to the original one, but with a rotation of its center. By default the size of the extruded cross-section is the same as the original one, and the rotation of the center is done in the active drawing plane..
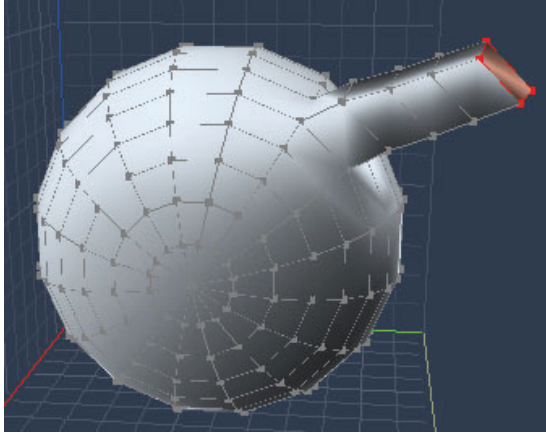
To create a Parallel Extrusion:

1. Select the polygon(s) that you want to extrude.

2. Do one of the following:

   - Choose **Construct > Extrude.**

   - Click the Dynamic Extrusion tool.

3. In the Extrusion Mode section of the Properties tray, choose the **Parallel Extrusion** mode.
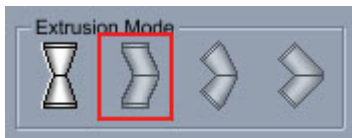
4. Click and drag the polygons to be extruded.

   - Hold the **Shift** key down during the extrusion to adjust the size of the extruded section.

   - Hold the **Command/Ctrl** key down during the extrusion to constrain the rotation in a plane that is perpendicular to the drawing plane.

Irregular Extrusions

Irregular Extrusion rotates the new cross-section from the original one. By default, the extruded cross-section and the original cross-section have the same size. The rotation of the cross-section is done in the active drawing plane. Depending on the angle between the two cross-sections, the extrusion may appear to shrink.
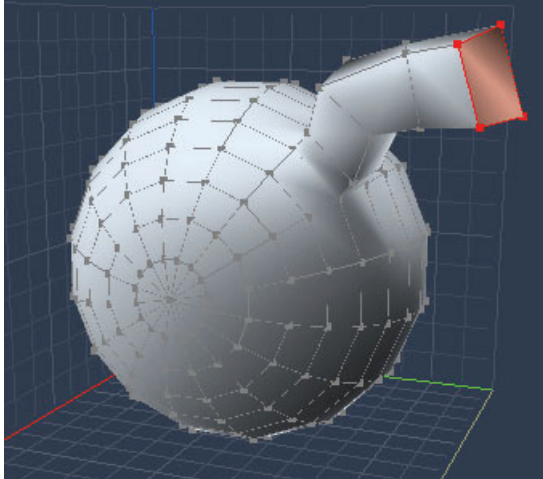
To create an Irregular Extrusion:

1. Select the polygon(s) that you want to extrude.

2. Do one of the following:

    - Choose **Construct > Extrude.**

    - Click the Dynamic Extrusion tool.

3. In the Extrusion Mode section of the Properties tray, choose the **Irregular Extrusion** mode.
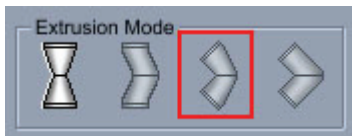
4. Click and drag the polygons to be extruded.

    - Hold the **Shift** key while extruding to set the size of the new cross-section.

    - Hold the **Command/Ctrl** key down during the extrusion to constrain the rotation in a plane perpendicular to the drawing plane.

Regular Extrusions

Regular Extrusion is similar to Irregular Extrusion with the difference here that the new cross-section is scaled so that the extrusion will appear to have a constant section, whatever the angle between the two cross-sections is.
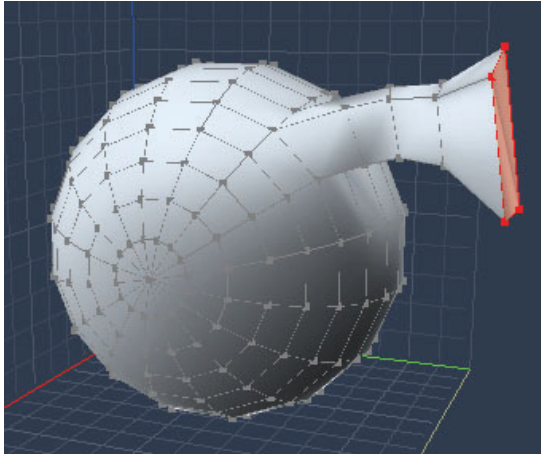
To create a Regular Extrusion:

1. Select the polygon(s) that you want to extrude.

2. Do one of the following:

    - Choose **Construct > Extrude.**

    - Click the Dynamic Extrusion tool.

3. In the Extrusion Mode section of the Properties tray, choose the **Regular Extrusion** mode.

4. Click and drag the polygons to be extruded.

    - Hold the **Shift** key while extruding to fix the rotation of the extrusion.

    - Hold the **Command/Ctrl** key down during the extrusion to constrain the angle of rotation to preset limits.

Path Sweep

The Path Sweep tool uses the same principles as Dynamic Extrusion. However, the extrusions are made along a predefined path rather than by manually clicking and dragging the polylines and polygons to extrude.

Like cross-sections, the sweep path can be defined by a separate polyline object or a set of selected edges of a larger object. The vertices of a sweep path do not have to lie on the same plane, allowing you to easily define complex 3D paths.

Path Extrusion

Sweeping mode

☐ link polygons

Choose a method and select the path to sweep.
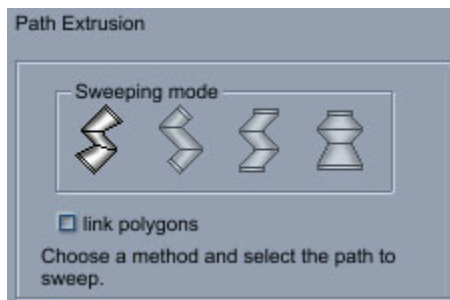
To sweep a cross-section:

1. Create one or more cross-sections. For this, you can use the Polyline tool, select an existing polygon or closed polyline, or create a 2D primitive.

2. Draw the sweep path with the Polyline tool, or select an existing polyline to use as the path.

TIP! You may find it convenient to switch to a 2D view to create an extrusion path perpendicular to the cross-section.

3. Select the cross-sections. You can also optionally select the path at this time.

4. Do one of the following:

   - Choose **Construct > Sweep**.

   - Click the Path Sweep tool.

5. Choose the sweep mode in the tool options of the Properties tray. There are 4 ways of sweeping a cross-section along a path:

   - **Irregular Sweep** The cross-section remains perpendicular to the sweep path and its size remains constant.
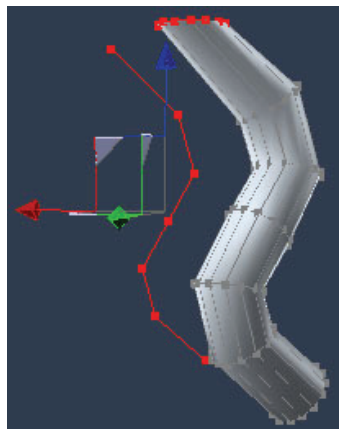
- **Regular Sweep** The cross-section remains perpendicular to the sweep path and its size is modified so that the section of the created volume is constant.
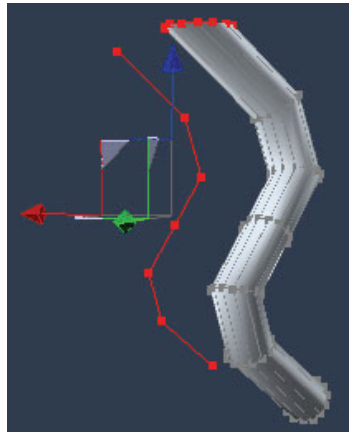
- **Parallel Sweep** The cross-section remains parallel to the original one.
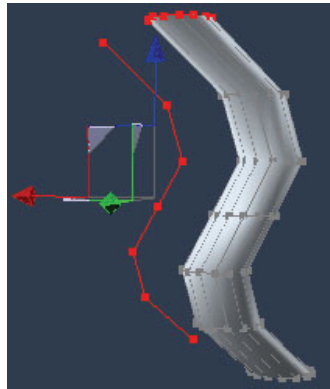
- **Envelope** The cross-section remains parallel to the original one and is scaled to follow the path. In this mode, the position of the path relative to the original is very important since the distance between the path vertices and the cross-section determines the scaling value.
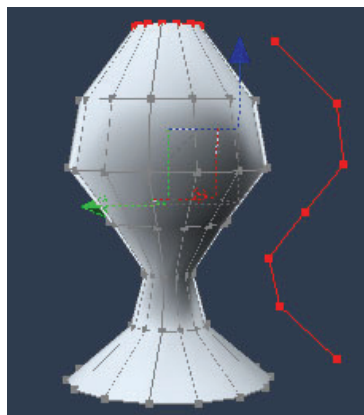
6. If you want to use a group of polygons as a single cross-section, check the **Link Polygons** option in the tool options of the Properties tray.
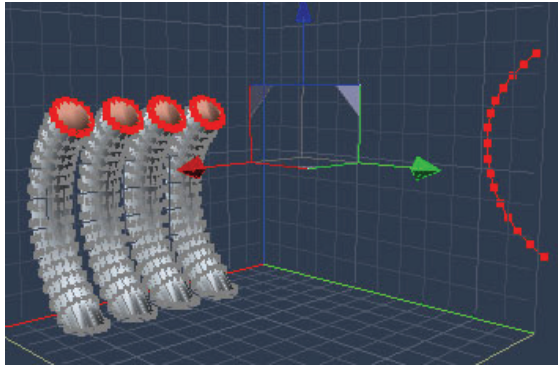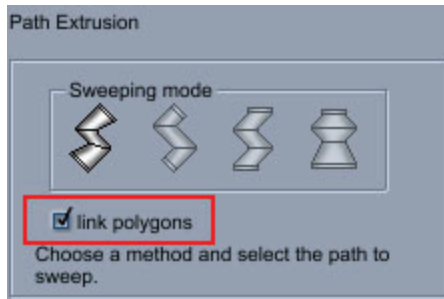
Lathing with Polygon and Polyline Profiles

When you lathe objects in the Vertex modeler, you control both the lathe profile and the lathe axis. The lathed object is created by revolving the profile around the specified axis. The lathe profile is always rotated a full 360 degrees around the axis.

Both polylines and polygons can be used as lathe profiles. The lathe axis can by defined by any edge.

To lathe an object:

1. Create the lathe profile with the Polyline tool, or select a set of connected edges to use as the lathe profile.

2. Select the profile that you want to lathe.

3. Choose **Construct > Lathe**, or click the **Lathe** tool. If the profiles are open polylines, new edges will appear in red. These new edges can also be used as an axis for the lathing.

4.   Select an edge to use as the axis.
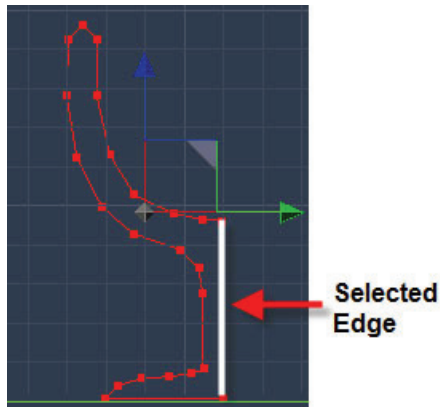
5.   The lathed object appears in the workspace. You can increase or decrease the number of sections with the + or - keys, or by manually entering the number in the **Number of Sections** field in the Properties tray.
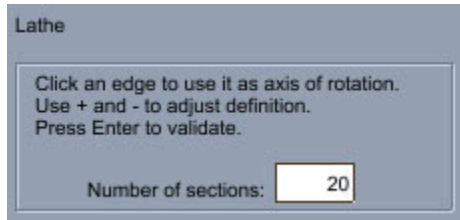
6.   Validate with **Enter** or **Return**.
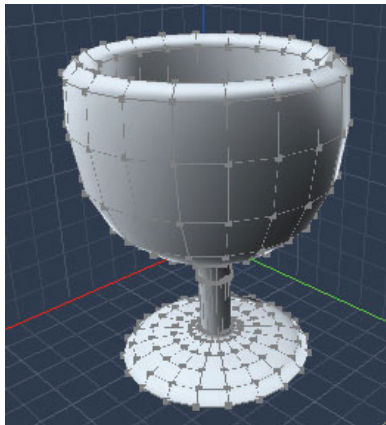
Lofting with Polyline Cross-Sections

Lofting enables you to construct a surface over a series of cross-sections. In the Vertex modeler, the cross sections can have the following characteristics:

- Cross-sections can be defined by open or closed polylines.

- They can be separate polyline objects, or selected edges in a larger object.

- You can define as many cross-sections for a single lofted object as you need, but you cannot mix open polylines and closed polylines. They must all be the same type.

- Cross-sections must lie on different planes.

- In certain situations, not all of the cross-sections you define are used. This might happen if two of the cross-sections lie very close to the same plane; one of them is used and the other ignored.
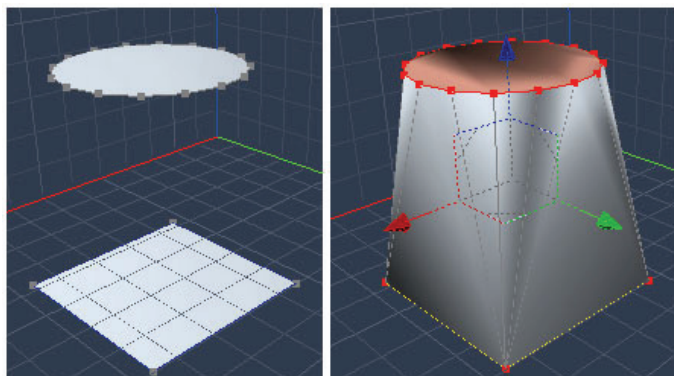
NOTE! Lofting can also be done using the Ruled Surface tool.

To loft an object:

1. Create the cross-sections with the Polyline tool or choose existing polylines to use as cross-sections.
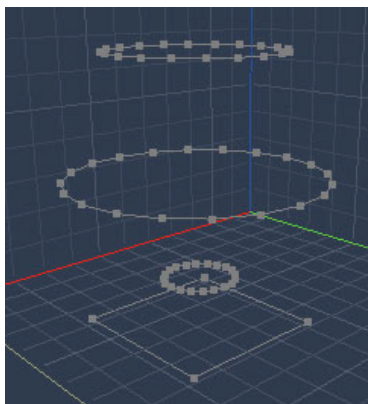
2. Select the cross-sections.

3. Choose **Construct > Loft**.
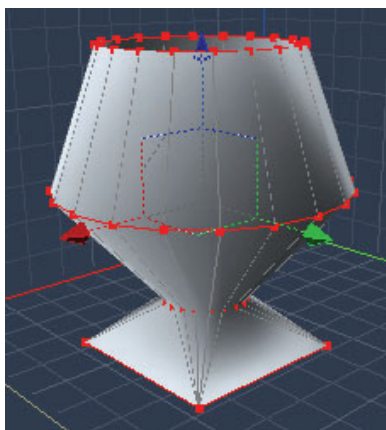
Creating Organic Models

Complex models can be created very simply by using flat polygons and applying Organic functions to them.

To create organic models:

1. Create a filled polygon using one of the following tools:

    - 3D Rectangle or Oval.

    - 2D Polyline, with the Close option checked in the Properties tray. After the polyline is created, use the **Model > Fill Polygon** command to fill it.

2. Select the filled polygon.

3. Choose **Construct > Organic**. The polyline becomes a three-dimensional object.
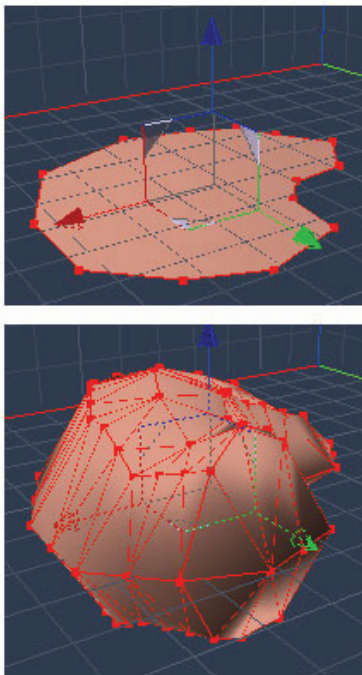
Ruled Surface

The Ruled Surface tool is another way to create Lofted surface. It's ergonomics allows you to select each section curve so that you can choose which curve is connected to the current curve by hand.

Sections curves must be all closed or all open. Closed curves must be empty (not filled). Polygon edges cannot be selected as section curves without being extracted first.

To create a Ruled surface:

1. Create the curves that you want to loft.

2. Do one of the following:

    - Choose **Construct > Ruled Surface**.

- Select the **Ruled Surface** tool.

3. Click on the first curve. It turns blue.

4. Click each successive curve that you want to connect. The surface appears between each selected curve as you click.

5. After you click the last curve, press **Enter** or **Return** to stop.
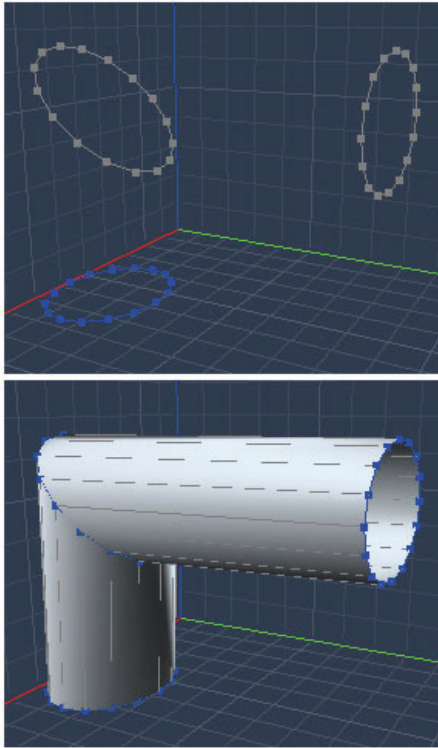
Coons Surface

The Coons Surface tool lets you create a surface between a closed curve, or several open curves connected together.

This tool is very useful for the creation of complex shells, such as for a car hood when two profiles and two sections are necessary. Here we can see that it could be difficult to create such a complex form quickly.

Using the Coons Surface tool, you can define a curve for the lateral part, one for the center, and one for the front, and finally another for the back.

To create a Coons Surface:

1. Create the curves to be used by the Coons Surface tool. Make sure each curve is connected on both ends so that the curves define a closed loop.
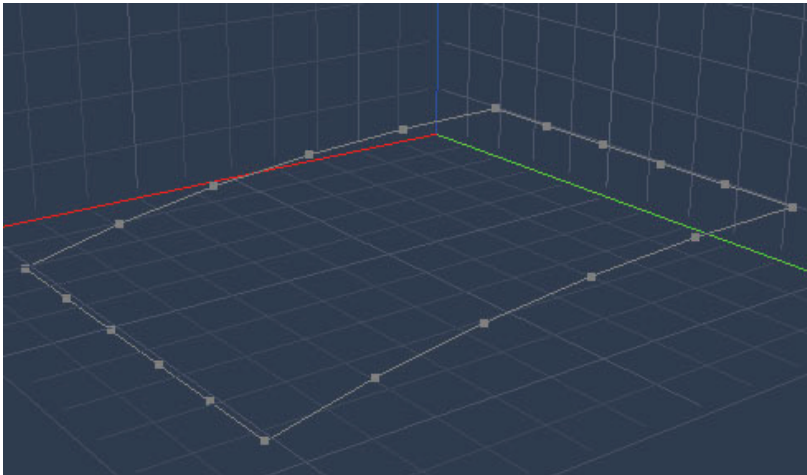
2. Do one of the following:

    - Choose **Construct > Coons Surface**.

    - Select the Coons Surface tool in the Surface modeling tab.

3. Click on each of the curves with the Coons Surface tool. The selecting order is not important.

4. Once all the connected curves are selected, the surface is automatically created.
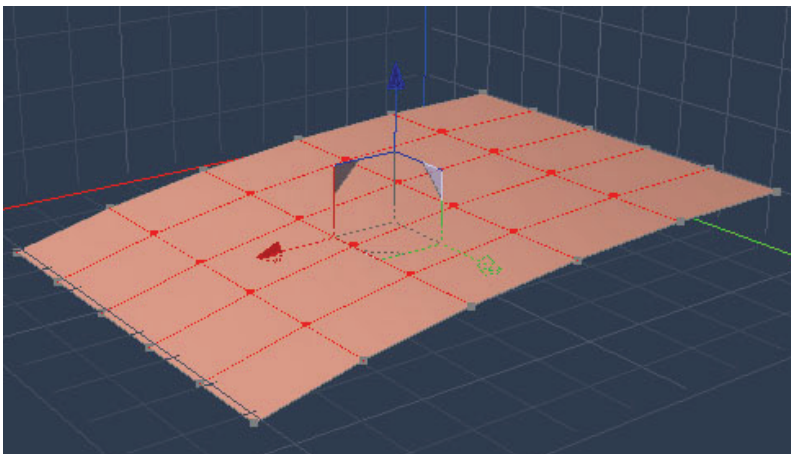
Gordon Surface

The Gordon Surfaces tool is generally used in product design. It allows you to create complex surfaces based on very precise curves, like a custom bottle for example. The surface is defined by a series of latitudinal cross sections and longitudinal profiles. The cross-sections and profiles must be created before the use of this tool.

 NOTE! There is no Close option for the Gordon Surface tool.

To create a Gordon surface:

1. Use any of the 2D vertex object tools to create the cross-sections for your model. You might find it easier to accomplish this while you view your scene in Four Views mode (Top, Left, Front, and Director's Camera views).
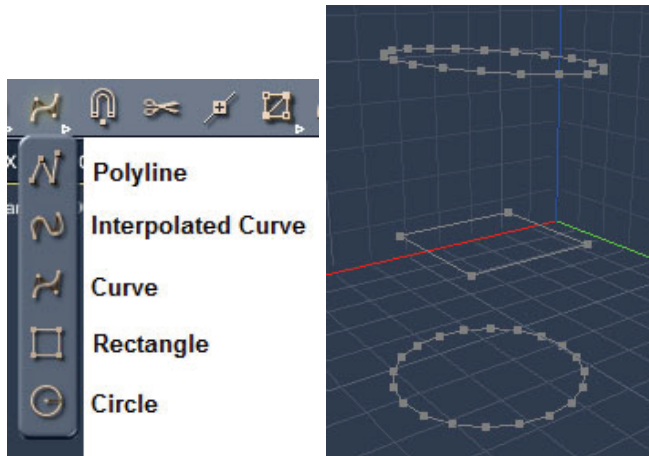
2. Select the Polyline tool. Profiles must be open shapes, so uncheck the **Close** option in the Properties tray if necessary.

3. Create the profile shapes, connecting the cross-sections together. A small square appears beside the cursor when you hover over a connection point.

4. Press **Enter** to complete each profile shape. Profiles are colored red while you create them.
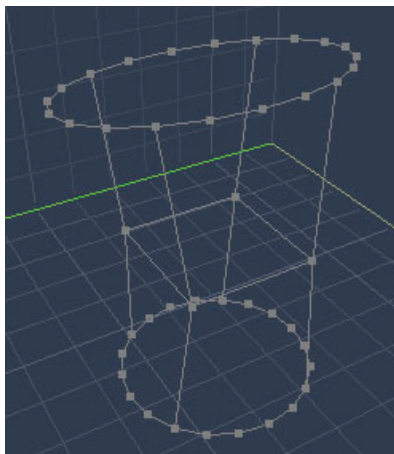
NOTE! Make sure that each point of a profile passes through a point on each of the cross-sections. All sections and profiles must be connected, and the ends of the profiles must connect to the first and last cross-sections. Profiles cannot go past a cross-section.

5. Click outside the shape to deselect all current selections.

6. Do one of the following:

   - Choose **Construct > Gordon Surface**.

   - Select the Gordon Surface tool.

7. Select the cross-sections.

8. Press **Enter** to complete the selection. The cross-sections turn from red to blue.
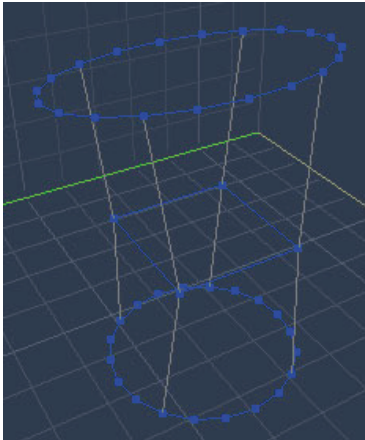
9. Select the profiles, which are colored green during the selecting process.
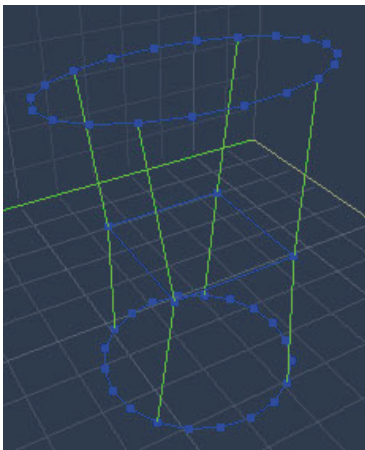
10. Press **Enter** to complete the shape.
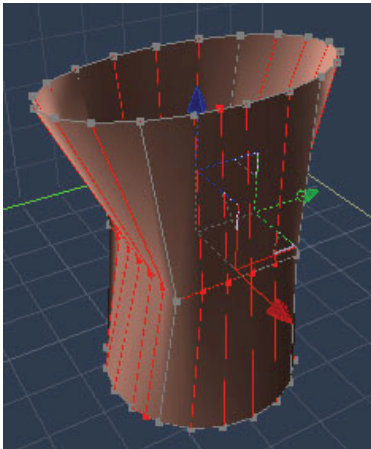
NOTE! If your curves are not properly connected, Carrara generates the surface as best as possible. The results may not be satisfactory if points are missing. Undo the operation, verify your connections, and try again.

Double Sweep Surface

The Double Sweep tool lets you create a surface or volume from a section and two profiles connected to it.

To create a Double Sweep surface:

1. Create three curves: one cross-section curve and two profile curves.
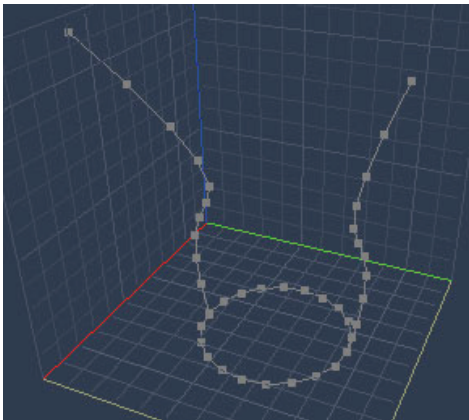
2. Do one of the following:

   • Choose **Construct > Double Sweep**.

   • Select the Double Sweep tool.

3. Click the cross-section curve. It turns blue.

4.   Click the two profile curves (in no particular order. Upon clicking the second profile, the surface is created.
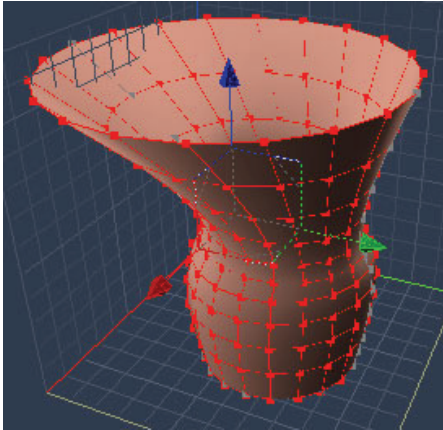
Using Boolean Operations

Boolean operations are used to produce a complex object from two simpler objects. You can perform Boolean operations in the Vertex Modeler and the Spline modeler.

Using Boolean operations enables you to quickly produce certain types of complex models. For example, suppose that you want to create a cylindrical hole through a sphere that originates on the top of the sphere and exits to the sphere's left side.

Using normal modeling techniques, this would be nearly impossible, but with Boolean subtraction it's easy. It's just a matter of aligning a bent pipe and a sphere and then subtracting the two.

Before you perform a Boolean operation, it is important to verify that the objects are aligned the way that you want them. Use the camera orientation controls and take advantage of the scene window's multiple views to look at the objects from a variety of viewpoints. You might also want to check the Properties tray to verify the location of the objects.

There are three types of Boolean operations. In order to perform them, two objects have to be arranged as follows:

- You must have two overlapping objects.

- Both objects should be closed. Holes in the surface of a volume can prevent Boolean operations from producing the desired results.

Note When you perform a Boolean operation on two objects, a single object is produced with its own UV coordinates, so be sure that shading is applied the way you want it.

The Boolean tool can perform the three most commonly used operations (Union, Intersection and Subtraction), and it can also create all the possible operations such as the closed intersections, open intersections, intersection lines, and so on.

There are 12 possible combinations, accessible in the property panel, either by clicking on the chosen icon, or by using the +/- keys to cycle through the choices.

Boolean Unions

The union of two objects is an object whose surface encompasses the visible surfaces of both objects. For example, if you spray-paint two overlapping spheres, the painted surface represents the union of the two spheres.

To join two objects with the Boolean Union command:

1. Align the objects that you want to join.

2. Double-click to select the first object that you want to join.

3. Press the Shift key while you double-click the second object to join.

4. Choose **Construct > Boolean > Boolean Union**.

5. Rename the resulting object if you like.

To join two objects with the Boolean tool:

1. Align and select the objects that you want to join.

2. Click the **Boolean tool**.

3. Click to select the first object that you want to join.

4. Click on the second object. A Boolean operation is performed and a result is displayed.

5. Use the cursor or the +/- keys on your keyboard to select **Union** from the Operation section in the Properties tray.
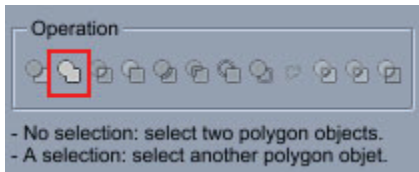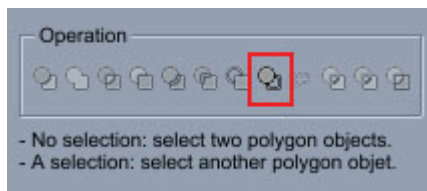

6. Rename the resulting object if you like.

Boolean Intersections

If you take the intersection of two objects, the resulting object is the volume shared by both objects. Carrara provides a number of different ways that you can intersect two objects:

NOTE! In the examples below, the smaller cylinder is object 1 and the larger is object 2.

Intersection

To intersect two objects with the Boolean Intersection command:

1. Align the objects that you want to intersect.

2. Double-click to select the first object that you want to join.

3. Press the **Shift** key while you double-click the second object to join.
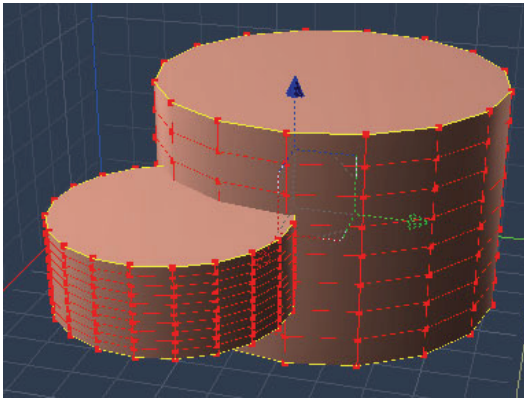
4. Choose **Construct > Boolean > Boolean Intersection**. The intersection object is created.
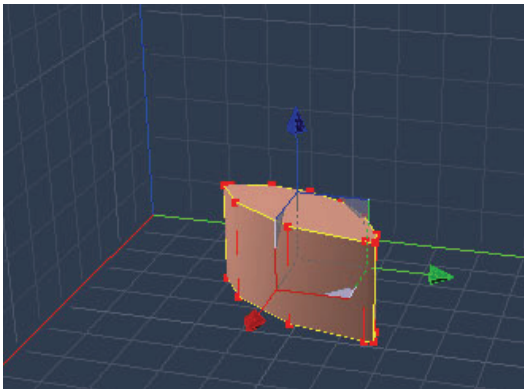
5. Rename the resulting object if you like.

To intersect two objects with the Boolean tool:

1. Align and select the objects that you want to join.

2. Click the **Boolean tool**.

3. Click to select the first object that you want to intersect.

4. Click on the second object. A Boolean operation is performed and a result is displayed.

5. Use the +/- keys on your keyboard or click to select **Intersection** from the Operation section in the Properties tray.

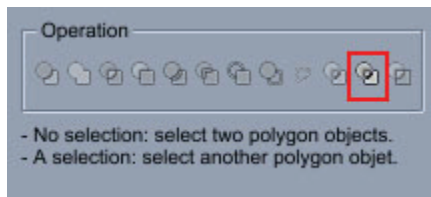

6. Rename the resulting object if you like.

Surface 1 Intersects Volume 2

To intersect surface 1 with volume 2:

1. Align and select the objects that you want to join.

2. Click the **Boolean tool**.

3. Click to select the first object that you want to intersect.

4. Click on the second object. A Boolean operation is performed and a result is displayed.

5. Use the +/- keys on your keyboard or click to select **Surf1 Inter Vol2** from the Operation section in the Properties tray.
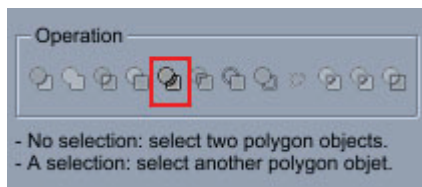
6. Rename the resulting object if you like.
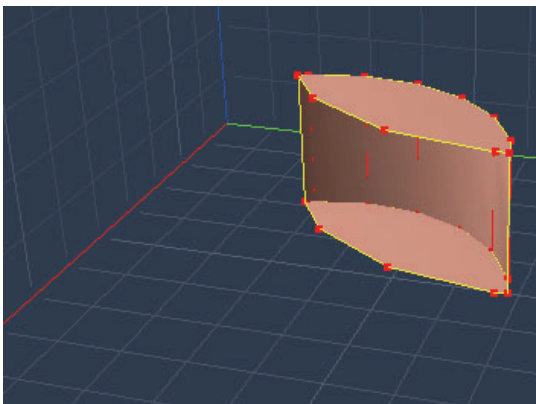
Surface 2 Intersects Volume 1

To intersect surface 1 with volume 2:

1. Align and select the objects that you want to join.

2. Click the **Boolean tool**.

3. Click to select the first object that you want to intersect.

4. Click on the second object. A Boolean operation is performed and a result is displayed.

5. Use the +/- keys on your keyboard or click to select **Surf2 Inter Vol1** from the Operation section in the Properties tray.
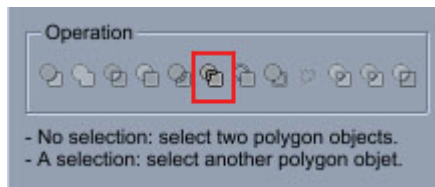
6. Rename the resulting object if you like.
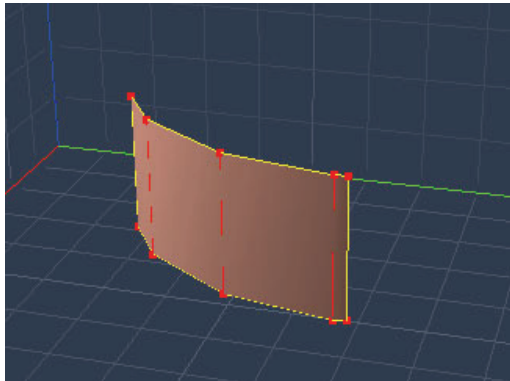
Surface 1 Intersects Surface 2

To intersect surface 1 with surface 2:

1. Align and select the objects that you want to join.

2. Click the **Boolean tool**.

3. Click to select the first object that you want to intersect.

4. Click on the second object. The Boolean intersection is created.

5. Use the +/- keys on your keyboard or click to select **Surf2 Inter Vol1** from the Operation section in the Properties tray.
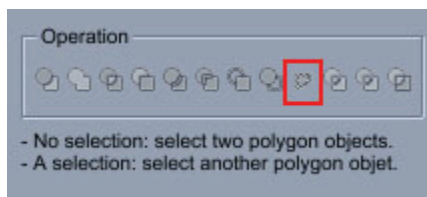
6. Rename the resulting object if you like.
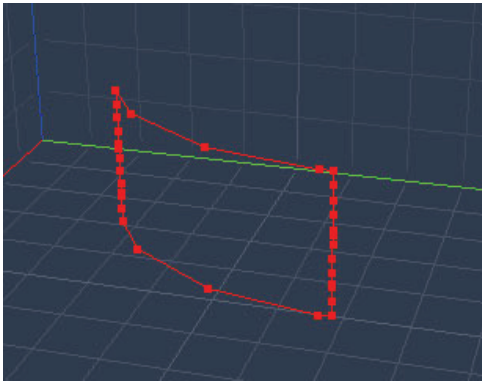
Boolean Subtractions

Boolean Subtractions

Boolean subtraction is similar to the concept of compounding shapes in the Spline modeler. However, compounding shapes only enables you to create holes through extruded objects. Boolean subtraction enables you to work directly with 3D objects and remove overlapping volumes.

To subtract one object from another with the Boolean Subtraction command:

1. Align the objects that you want to intersect.

2. Double-click to select the first object that you want to join.

3. Press the **Shift** key while you double-click the second object to join.
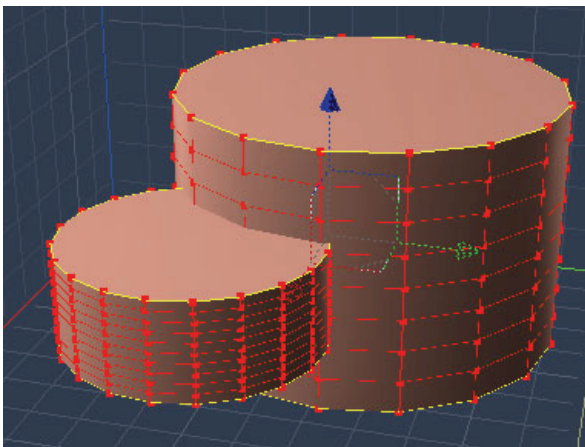
4. Press the **Cmd/Ctrl+Alt** keys while you click a point on the polymesh that represents the object you want to subtract from. The selected point turns yellow.

5. Choose **Construct > Boolean > Boolean Subtraction**. The intersection object is created.
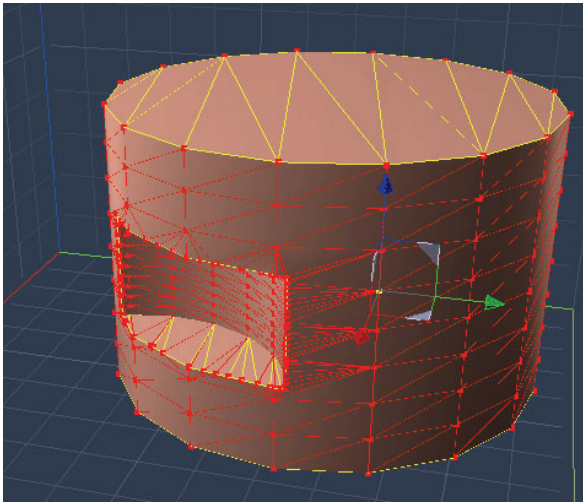
6. Rename the resulting object if you like.

Additional subtraction options are also provided. Refer to the following sections for variations on the Boolean Subtraction options.

Volume 2 Cut by Volume 1

To cut volume 2 by volume 1:

1. Align and select the objects that you want to join.

2. Click the **Boolean tool**.

3. Click to select the object that will be used to cut an area from the second object (Volume 1).

4. Click the object that the area will be cut from (Volume 2). The Boolean object is created.

5. Use the +/- keys on your keyboard or click to select **Vol2 Cut by Vol1** from the Operation section in the Properties tray.
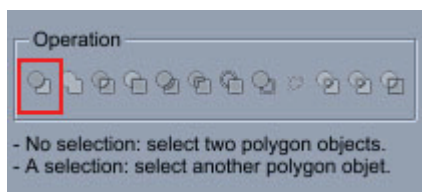
6. Rename the resulting object if you like.
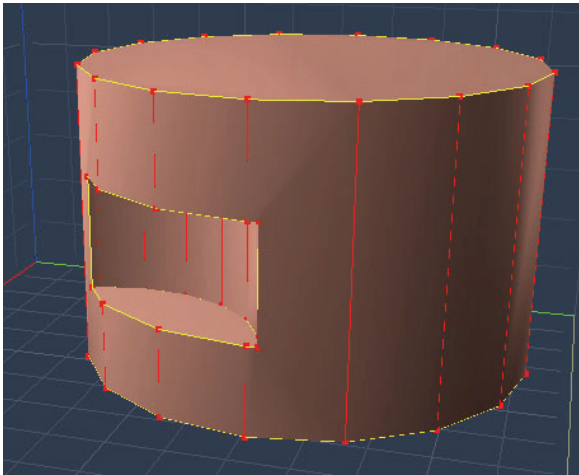
Volume 1 Cut by Volume 2

To cut volume 1 by volume 2:

1. Align and select the objects that you want to join.

2. Click the **Boolean tool**.

3. Click to select the object that the area will be cut from (Volume 1).

4. Click the object that will be used to cut an area from your first selection (Volume 2). The Boolean object is created.
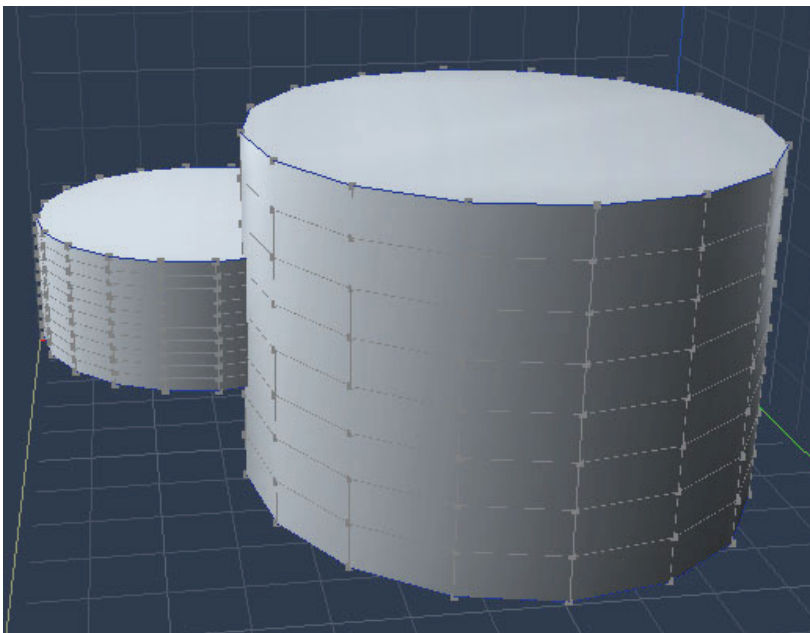
5. Use the +/- keys on your keyboard or click to select **Vol1 Cut by Vol2** from the Operation section in the Properties tray.
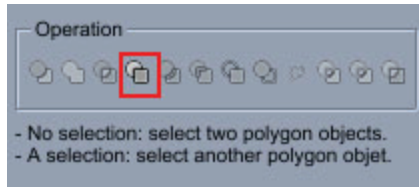
6. Rename the resulting object if you like.
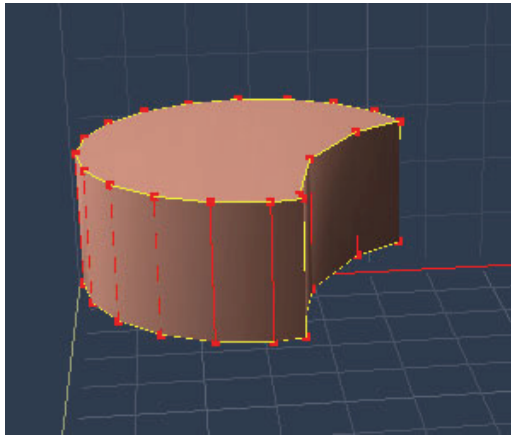
Surface 1 Cut by Volume 2

To cut surface 1 by volume 2:

1. Align and select the objects that you want to join.

2. Click the **Boolean tool**.

3. Click to select the object that the surface will be cut from (Surface 1).

4. Click the object that will be used to cut the area from your first selection (Volume 2). The Boolean object is created.

5. Use the +/- keys on your keyboard or click to select **Surf1 Cut by Vol2** from the Operation section in the Properties tray.

6.   Rename the resulting object if you like.
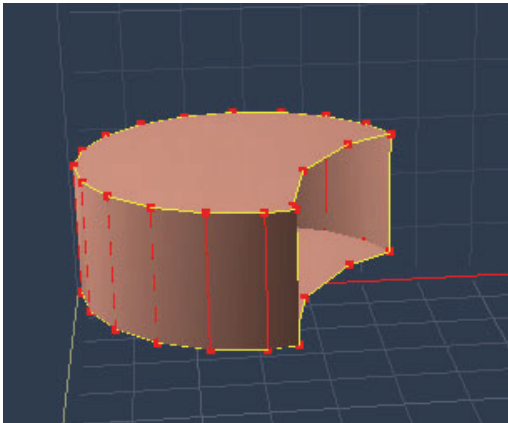
Surface 2 Cut by Volume 1

To cut surface 2 by volume 1:

1.   Align and select the objects that you want to join.

2.   Click the **Boolean tool**.

3.   Click the object that will be used to cut the area from your first selection (Volume 1).

4.   Click to select the object that the surface will be cut from (Surface 2). The Boolean object is created.

5.   Use the +/- keys on your keyboard or click to select **Surf2 Cut by Vol1** from the Operation section in the Properties tray.

6.   Rename the resulting object if you like.
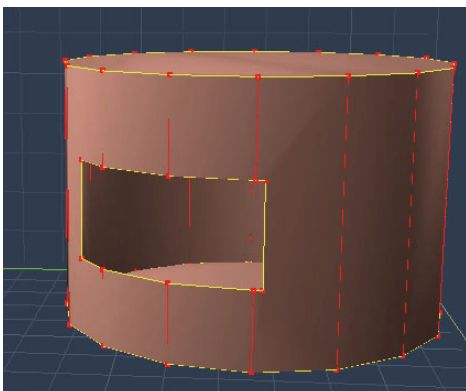
All Parts

To keep all parts of the Boolean intersection:

1. Align and select the objects that you want to join.

2. Click the **Boolean tool**.

3. Click the first object for the Boolean operation.

4. Click the second object. The Boolean object is created.

5. Use the +/- keys on your keyboard or click to select **All Parts** from the Operation section in the Properties tray.

6. Rename the resulting objects if you like.
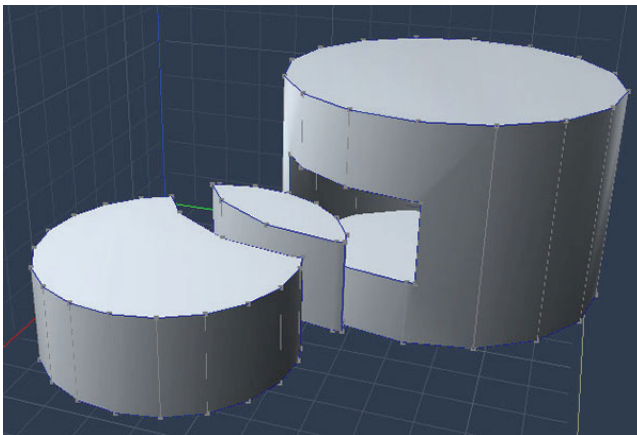
All Parts of Object 2

To keep all parts of the second object:

1. Align and select the objects that you want to join.

2. Click the **Boolean tool**.

3. Click the first object for the Boolean operation.

4. Click the second object. The Boolean object is created.

5. Use the +/- keys on your keyboard or click to select **All Parts of Obj2** from the Operation section in the Properties tray.

6. Rename the resulting objects if you like.
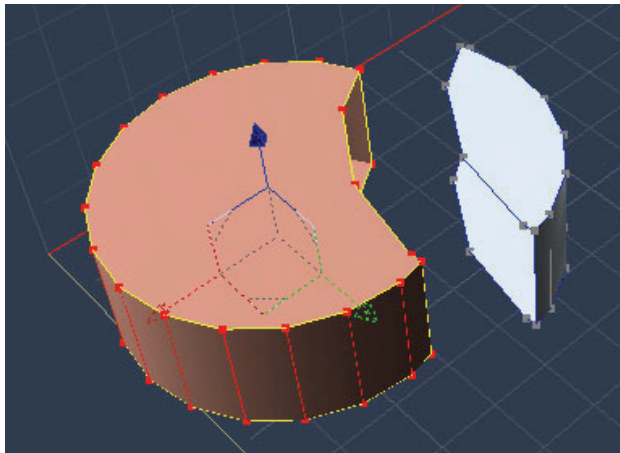
The resulting Boolean operation, with parts moved for demonstration.

All Parts of Object 1

To keep all parts of the second object:

1. Align and select the objects that you want to join.

2. Click the **Boolean tool**.

3. Click the first object for the Boolean operation.

4. Click the second object. The Boolean object is created.

5. Use the +/- keys on your keyboard or click to select **All Parts of Obj1** from the Operation section in the Properties tray.
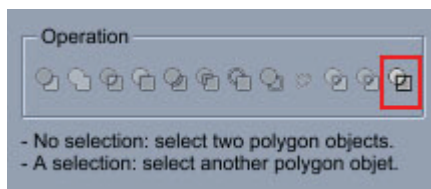
6. Rename the resulting objects if you like.

Using the Cut Tool (Standard, Pro)

The Cut tool is very similar to the Boolean tool in that it allows you to punch a volume with a polyline or a curve.

NOTE! When you perform a Boolean operation, you lose the UV coordinates of the original objects and surface shading can become more complex.

The Cut tool can be used to punch a volume with a polyline or a curve.

To cut an object with a line:

1. Draw the line that must cut the object. If the line has more than three points defining a plane, the cut will take this plane as the reference plane for cutting. The curve will be projected perpendicularly to that plane. Otherwise, the current active plane is used.

2. Select the object to cut.

3. Select the Cut tool.

4. Click on the line.

5. The Boolean operation is performed and a first result is displayed.

6. Click on the operator icon in the property tray or use the +/- keys to select the operation required.

7. Hit Return to stop.

Spline Modeling Overview (Standard, Pro)

The Spline modeler in Carrara lets you create objects by converting 2D shapes into 3D objects. You use the Spline modeler tools to draw 2D shapes, known as cross-sections, and then convert them to 3D objects by extruding them. Once you've created a 3D object, you can refine its shape using an extrusion envelope. This chapter covers spline modeling concepts and describes how to use the Spline modeler.

Spline Modeling Concepts

The Spline modeler is based on a concept called **extrusion**. You create an object by drawing a 2D shape, then drawing a path perpendicular to the shape. Carrara extrudes the shape along the path to form a 3D object.
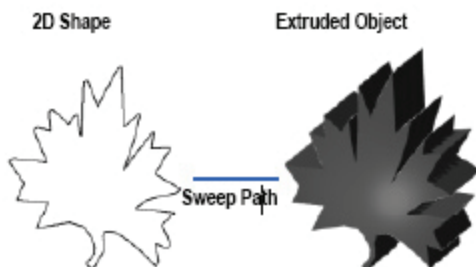
Straight Extrusion

Straight extrusion is the most basic type of spline modeling. In a straight extrusion, a 2D shape is placed on a cross-section and swept along a straight sweep path, creating a "cookie cutter" effect.

The cross-section doesn't have to consist of a single 2D shape:it can consist of several distinct shapes. You can even create an object with a hole in it by extruding a shape along a compound path. For more information on extrusions, refer to "Cross-Sections and the Sweep Path."

Scaling

You can also shape objects by changing the scale of the 2D shape as it's extruded along the sweep path.

Simply changing the scale of a shape produces very basic shapes, but when combined with an extrusion envelope, you can create a much wider variety of objects. For more information on scaling, refer to "Understanding the Envelope".
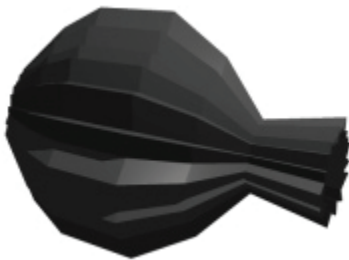
Lathing

You can model many symmetrical objects using a technique known as **lathing**. Carrara creates lathed objects by extruding a **lathe profile** around a sweep path. You can think of the lathe profile as the outline of the object when it's cut in half. For more information on lathing, refer to "Lathing with the Extrusion Envelope."

Lathe Profile   Sweep Path (Lathe Axis)

Cross-Sections

Cross-sections are 2D shapes that act as the skeleton of your object. The Spline modeler extrudes from each cross-section to the next, basing the contours of the object's surface on the shapes in the cross-sections.
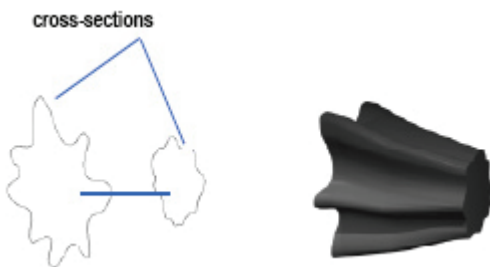
cross-sections

This modeling technique is sometimes referred to as skinning:the Spline modeler stretches a "skin" over the shapes in the cross-sections.

Each cross-section can have any number of 2D shapes. However, the more shapes you add, the more complicated your extrusion gets.

For more information on cross-sections, refer to "Working with Cross-Sections."

Complex Sweep Paths

The sweep path determines the shape and size of the extrusion. A straight sweep path creates a straight object; a more complex sweep path creates objects with bends and curves.
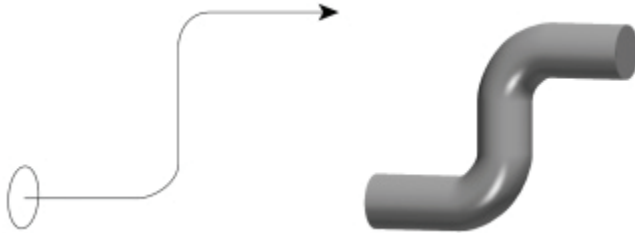
The Spline modeler allows you to draw a true 3D sweep path, meaning it can be adjusted in the X, Y, and Z axes. You can also close the sweep path to create a continuous object like a chain link. Carrara also supplies tools for automatically creating complex sweep paths like spirals. For more information on working with the sweep path, refer to "Drawing a Sweep Path in 3D."

For a mini-tutorial on creating different types of spirals, refer to "Tutorial: Spirals."

Choosing a Spline Modeling Technique

The Spline modeler offers several ways of creating spline objects:lathing and skinning between different 2D shapes, to name two. You must decide which technique is best suited for your object before you begin modeling.

An easy way of determining which technique you must use is to take an imaginary knife and cut your object into slices. The slices you end up with will tell you which technique you should use.

If you get identical shapes, use straight extrusion. If you get congruent shapes that differ only in size, use an extrusion envelope.
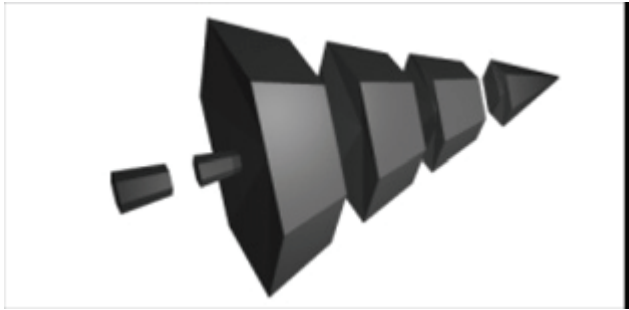
If shapes vary dramatically in size and geometry, you must use a more complex modeling technique. Use lathing if the shapes are symmetrical. If not, use shape numbering and multiple cross-sections.

The Spline Modeling Window

The Spline modeling window defines the drawing environment and provides you with planes for drawing sweep paths, cross-sections, and extrusion envelopes.
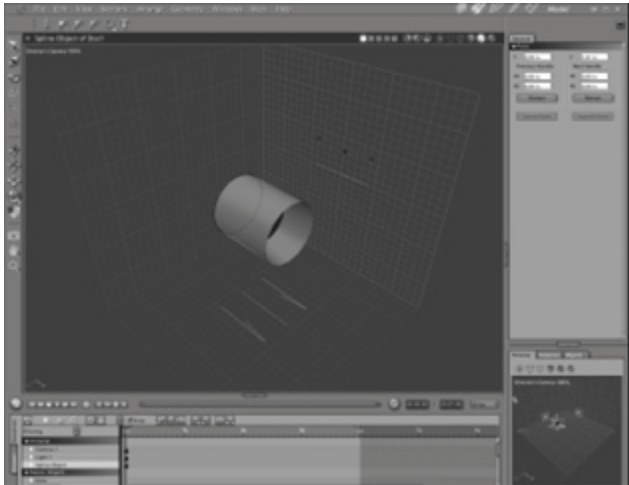
When you are using the Spline modeler, the Spline modeling window opens, new menu commands are available, different tool sets appear, and the Properties tray describes your object. Your view switches to a close-up view of your object.

In addition, the Scene Preview window shows how your object will look in the Assemble room.

As with the Assemble room, the Spline modeling room provides up to four separate panes, each offering a different view of your object.

The Spline Modeler 3D View

The Spline modeler's 3D view shows a working box (also known as the Modeling box) that contains the features for creating cross-sections and paths.
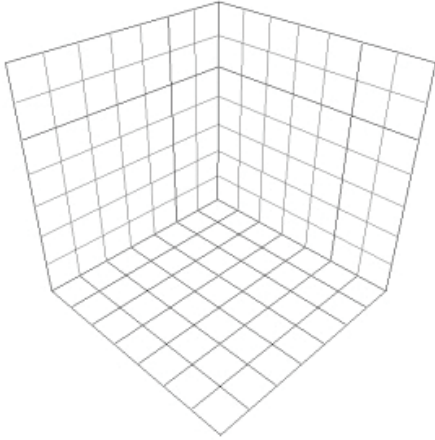
The Drawing plane is highlighted as a darker shade. By default, each side of the working box measures 14 inches, with a grid line drawn every inch. For information on changing these dimensions, refer to "Resizing the Modeling Box."

You can move and rotate the working box along with the object to get a better view of your work. You can also resize the working box and choose whether to resize the object along with it.

The Drawing Plane

The Drawing plane is the plane you're currently working on, whether you're drawing a shape, adjusting a sweep path, or viewing your object. In 3D view, the Drawing plane is highlighted as a darker shade.
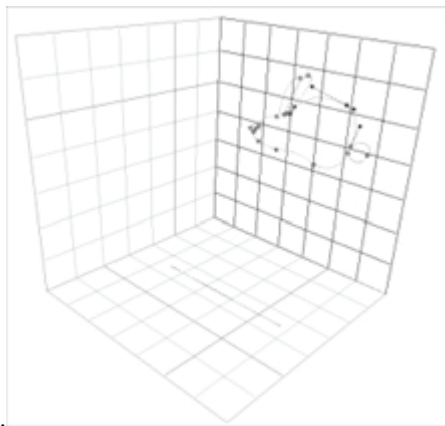
Cross-Section Planes

The cross-section planes are where you create the 2D cross-sections that make up the skeleton of your object. You use the Spline primitives and drawing tools to draw and edit shapes on the cross-section planes.

A cross-section plane contains only one cross-section, which can consist of any number of 2D shapes.
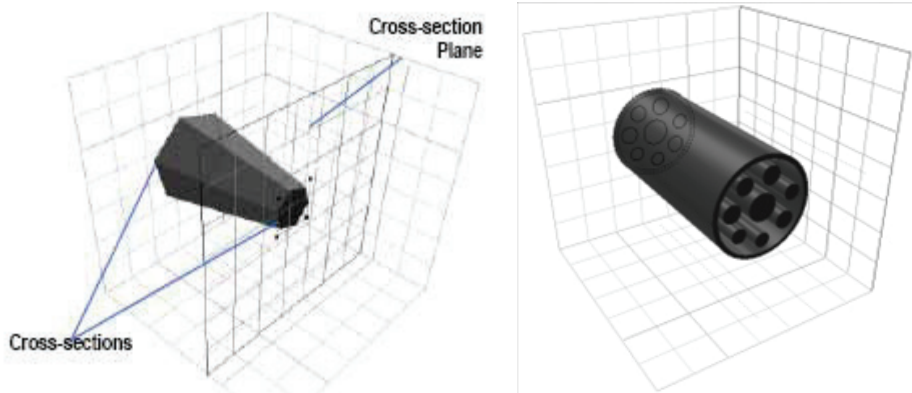
For more information on cross-sections and cross-section planes, refer to "Working with Cross-Sections."

The Sweep Path

The sweep path, also known as the **extrusion path**, is the path along which cross-section shapes are extruded. The path is indicated by two lines that appear on the bottom and side walls of the working box.
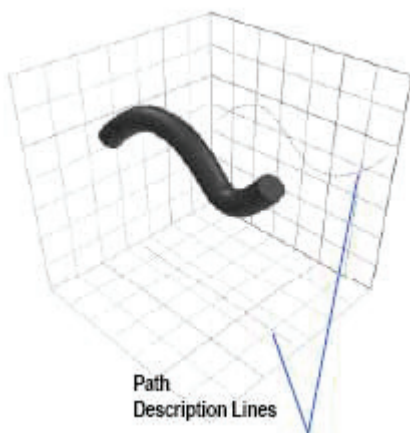
You can think of these lines as horizontal and vertical projections of the same sweep path. They let you edit the sweep path in both the XY and YZ plane. You can draw a sweep path of any shape or angle including curved, straight, or closed paths.

For more information on the sweep path, refer to "Working with the Sweep Path."
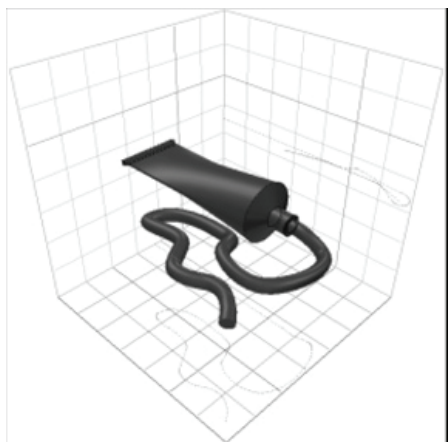
The Extrusion Envelope

The extrusion envelope lets you control the curvature of the 3D object around (as opposed to along) its sweep path. You do this by scaling the object's cross-sections. You can scale cross-section shapes symmetrically or asymmetrically.

Envelope
Description Lines
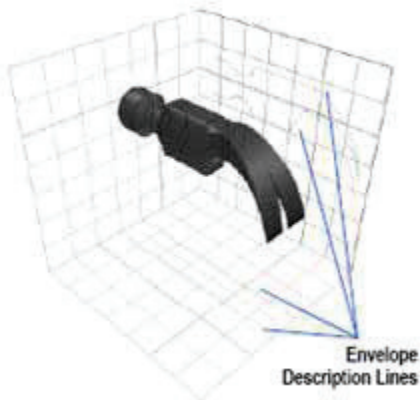
Carrara does not display the envelope when you first open the Spline modeling window. You add it using one of the Extrusion Envelope commands in the Geometry menu. It is indicated by four Bézier curves, two on each sweep path plane. Separate colors are assigned to the YZ and XY plane. Corresponding colored dots appear on the cross-section plane indicating where these curves cut through it. You can scale the object's cross-sections by editing these Bézier curves.

For more information on the extrusion envelope, refer to "Using the Extrusion Envelope."

Setting Preferences for the Spline Modeler

In the Spline modeler's Preferences dialog you can set preferences for the grid, for naming objects, and for resizing shapes and groups of shapes.

To set preferences for the Spline Modeling window:

1. Choose **File > Preferences**. The Preferences dialog appears.

2. Click the menu and choose Spline modeler from the list.

3. Set the following options as needed The options that you can set are as follows:

    - **Resize Shapes And Groups** specifies whether an object is resized from its center or from its opposite corner when you resize by dragging.

- **Grid Settings** specifies grid settings for the Spline modeler.

- **Spacing** adjusts the spacing between grid lines in the box and specifies units of measure.

- **Draw a line every** controls how often grid lines are drawn.

- **Show** toggles the display of grid lines.

- **Snap to** causes objects to "snap" to the nearest grid line as you drag them.

- **Colors** lets you set up a specific color scheme for the Modeling box. When you click a color swatch, the Color Picker appears. Refer to "Color Pickers" for more on choosing colors.

- **Show 3D Object in Preset 2D Views** lets you display the 3D object even in the preset 2D views such as Front, Top, and Left.

4. Once all options have been set, click on **OK**.

Opening the Spline Modeling Window

You open the Spline modeling window by doing one of the following:

- Double-click the object in the Sequencer or the scene window.

- Select a spline object, then click the Model button.

- If you're creating a new spline object, the modeling window opens automatically.

When you're finished modeling, you can save your changes and enter a different room by clicking the appropriate room button.

NOTE! To avoid the memory-consuming accumulation of windows, always close the modeling window when you are finished modeling.

To create a new spline object:

1. Open or create a document in the Assemble room. (For more information, refer to "Creating an Empty Scene.")

2. Depending on the desired location of the spline object, do one of the following:

   - To insert a spline object of default size at the center of the universe, choose **Insert > Spline Object** or drag the Spline Object tool into the Sequencer.

   - To insert a spline object in any other location, drag the Spline Object tool into the scene window, to any location

   - To insert a spline object of a specific size, select the Spline Object tool, then click and drag anywhere in the scene window.

Working with the Modeling Window

As with the document window, the Spline modeling window can simultaneously display up to four separate panes. Each pane gives you a different view of your object.

You can work in any view you want. For instance, you might find it convenient to work in a 2D view when drawing a sweep or extrusion path, then switch to the 3D view when you want to work in 3D. To work in a different view, simply click in the pane.

NOTE! Use the "Show 3D Object in Preset 2D Views" option in the Spline modeler Preferences dialog to hide or show your object in the isometric views of the modeler (Front, Left, Current Section...).

To make your work flow easier, you can change different features of the Spline modeling window:

- Change the display and behavior of the grid. For more information, refer to "Setting Preferences for the Spline Modeler."

- Resize the Modeling box.

Resizing the Modeling Box

The Modeling box opens at a scale consistent with the proportions of objects in the scene. If you want to work with an object at some other scale, you can reset the Modeling box's size. You also have the option to resize the object along with the box.

To set Modeling Box and object size:

1. Choose **Geometry > Modeling Box Size**. The Modeling Box Size dialog appears.
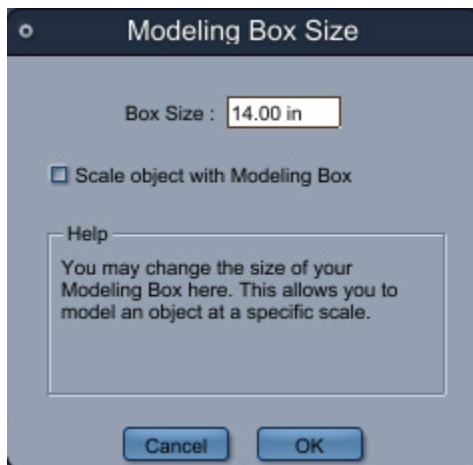
2. Set the options you want:

    - **Box Size** sets the length of one side of the (square) Modeling box.

    - **Scale object with Modeling Box** resizes the object along with the Modeling box.

3. Click **OK**.

Changing Your Point of View

Without changing your point of view to examine an object from all sides, it can be difficult to determine its position relative to other objects in the model. When you perform a operation such as subtraction, this is particularly important:you must be sure the objects are actually overlapping for it to work.

You can dolly, zoom, bank, and track the Director's Camera using the camera controls.

You can also pan and zoom the view in 2D using the pan and zoom controls. For information on using these controls, refer to "2D View Tools."

You can also use the camera list in the upper left corner of the Spline modeling window to switch among the Director's Camera (3D view) and isometric views of the working box's sides. For information on using these controls, refer to "Camera Navigation."

Previewing Objects

You can preview objects in the Spline Modeling window at seven different levels of quality.

For more information on the Preview modes, refer to "Choosing a Preview Mode."

Spline Modeling Tools and Techniques (Standard, Pro)

Creating Shapes and Objects

To create spline objects, you create cross-sections and extrusion paths on the appropriate planes in the Spline Modeling window.

All 2D shapes and paths you draw in the Spline modeler are Bézier curves. A Bézier curve is an interpolated curve whose shape is determined by the relative positions of its vertices and control points. vertices and Each segment of a Bézier curve connects two vertices. The control points (handles) extending from each vertex determine the curvature of the path segments.
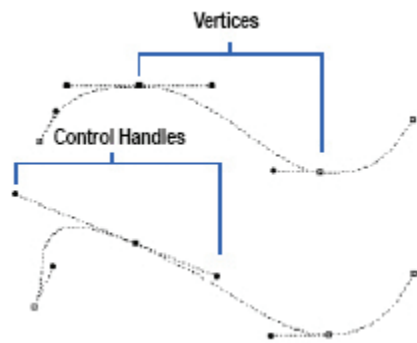
Carrara's drawing tools are similar to those found in traditional Bézier-based 2D drawing applications. The drawing tools enable you to create curves and shapes, and to edit and modify those shapes point-by-point.

Working on the Drawing Plane

This section discusses tools and techniques you'll need when creating objects in the Spline modeler.

Inserting Primitive Shapes

The most basic way to create an object in the Spline modeler is to start with a 2D primitive shape.

You can place a rectangle, rounded rectangle, ellipse, or polygon on a cross-section plane and then apply a spline modeling technique such as extrusion or sweeping.

After creating a shape with one of the Spline Primitive tools, you will not immediately be able to edit its points:you must ungroup it first. For more information, refer to "Grouping Shapes."

To insert a rectangular shape:

1.   Click the Spline Primitive tool and choose the Rectangular Shape tool.

2.   On the Drawing plane, drag from one corner of the shape to the opposite corner. Release the mouse button when the shape is the desired size.

3.   To create a square, choose the rectangle shape and hold down the Shift key while dragging.

To insert a circular shape:

1.   Click the Spline Primitive tool and choose the Circular Shape tool.

2.   Drag on the Drawing plane.

3.   To create a circle, choose the ellipse shape and hold down the Shift key while dragging.

To insert a rounded rectangle shape:

1.   Click the Spline Primitive tool and choose the Rounded Rectangle Shape tool.

2.   Drag on the Drawing plane. The Round Rectangle dialog appears.
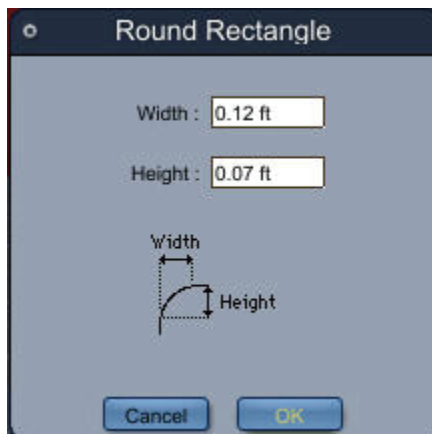
3.   Set the curvature on the corners and click OK.

4.   To create a square with rounded corners, hold down the Shift key while dragging.

To insert a polygonal shape:

1.   Click the Spline Primitive tool and choose the Polygon Shape tool.

2. Drag on the Drawing plane. The Number of Sides dialog appears.
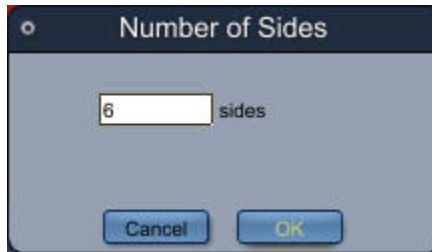
3. Set the number of sides for the polygon and click OK.

4. To keep all angles equal, hold down the Shift key while dragging.

Drawing Shapes

You can create open and closed paths of any shape you like with the Pen tool.

The Pen tool works in two ways: you can click and drag to draw curves, or you can add one point at a time like playing connect the dots. As you add points, Carrara connects them by drawing segments.

Points are classified as either corner points or curve points:

- A curve point's handles are bound together, creating a straight tangent for the path and resulting in a smooth curve.

- A corner point's handles can move independently of one another or retract them completely, allowing you to create abrupt changes in the direction of the path.

Corner Point　　　Corner Point　　　Curve Point
(Handles In)　　　(Handles Out)

To use the Properties tray to transform shapes you create with the Pen tool, you must group them first. For information on grouping shapes, refer to "Grouping Shapes." For information on setting object properties, refer to "Setting Object Properties."

To draw a new path:

1. Deselect all existing paths and shapes by clicking in an empty area of the Drawing plane with the Selection tool.

2. Click the Pen tool.

3. Click anywhere on the Drawing plane to start the new path with a corner point. Or, drag to start with a curve point.

4. Click or drag to add each subsequent point. As you add each point, Carrara draws the segments to connect the path.

To add a corner point:

1. Choose the Pen tool.

2. Click (do not drag) at a point on the Drawing plane.

3. Hold down the Shift key to draw the path in 45° increments.

To add a curve point:

1. Choose the Pen tool.

2. Drag the Pen tool at a point on the Drawing plane. As you drag, a pair of handles extends from the point. By default, each pair of handles is bound together, creating a curve point:the two handles remain parallel to one another.

   - Hold down the Shift key while dragging to move the handles in 45° increments.

   - Hold down the Option/Alt key while dragging to break apart a pair of handles, creating a corner point. You can then move each handle independently. Continue to hold the Option/Alt key down while dragging. If you release the key before releasing the mouse button, the handles snap back together.

When you click a curve handle, the handles extending from the point will move independently.

To close a path:

   - With the Pen tool, click the first point you added.

Editing Shapes

You can edit custom or primitive shapes and paths in the following ways:

   - Add and delete points

   - Delete segments

   - Convert corner and curve points

   - Move points

   - Adjust curves

To view all points on a path or shape:

1. Click the Selection tool.

2. Click a path or shape.

3. All the points on the curve become visible, but none are individually selected. A point appears black when it is selected and white when it is deselected.

   - Hold down the Shift key, then click additional paths to view the points on multiple paths.

   - Hold down the Shift key, then click a path whose points are visible to make them invisible.

   - Click in an empty area of the Drawing plane to make all points invisible.

To select points one at a time:

1. Click the Selection tool.

2. Click a point on a path or shape to select it.

3. The point color changes from white to black and its handles, if it has any, become visible.

   - Hold down the Shift key, then click additional points to increase your selection.

- Hold down the Command/Control key, then click a path to select all the points on the path. Or, double-click the path.

- Hold down the Shift key, then click a selected point to deselect it.

- Click in an empty area of the Drawing plane to deselect all points.

To select points by dragging:

1. Click the Selection tool.

2. Drag a marquee around the points you want to select.

To add points to either end of an open path:

1. Select one of the endpoints of an open path with the Selection tool.

2. Click the Pen tool.

3. Click or drag to add the next point. Carrara draws a segment to continue the path to the new point.

4. Continue adding points until you're satisfied with the path.

To add a point between two existing points:

1. Click the Add Point tool.

2. Click anywhere on a path or shape. Carrara determines whether to add a corner point or a curve point, depending on the shape of the path. The new point is automatically selected so it can be moved with the Selection tool.

3. When you add a point to the sweep path, you can simultaneously add a cross-section plane at that point:just hold down the Option/Alt key as you click.

To delete a point:

1. Click the Delete Point tool.

2. Click a point on a shape or path.

- When you delete a point in the middle of a path, the points on either side of the deleted point become connected by a new segment, changing the shape of the path.

- When you delete the endpoint of an open path, the last path segment disappears, leaving a new endpoint.

- If you delete a point on the sweep path, any cross-section associated with that point is deleted as well.

To delete a segment:

1. Click the Delete Point tool.

2. Click a path segment on a shape. Removing a path segment leaves adjacent path segments unchanged. When you delete a path segment from a closed path, the path becomes an open path.

When you delete a path segment from an open path, the path is split into two separate open paths.

Note The delete segment feature applies to cross-section shapes only:you cannot delete a segment from the sweep path or the extrusion envelope.

To convert a corner point to a curve point:

- Click the Convert Point tool.

To convert a corner point to a curve point:

1. Click the Convert Point tool.

2. Drag a point on a shape or path. As you drag, a pair of handles extends from the point.

To convert a curve point to a corner point using the tools:

1. Choose the Convert Point tool.
2. Click a curve point on a shape or path. The point's handles retract.

To convert a curve point to a corner point using the Properties tray:

1. Select a point on a cross-section plane.
2. You can only use the Properties tray controls for points on a cross-section plane.
3. In the Properties tray, click either the Corner Point or Smooth Point button.

To move points:

1. Click the Selection tool.
2. Select the points you want to move. To move an entire path, select all the points on the path.
3. Drag the selected points to a new location.
   - Hold down the Shift key while you drag to move the points in 45° increments.
   - You can also move points in cross-sections in the Properties tray. Select a point and enter values for X and Y.
   - All selected points move together. As you drag, Carrara redraws the path segments affected by the move.

To adjust a curve interactively:

1. Click the Selection tool.
2. Click a point on a path or shape to select it.
3. Drag the point's handles.
4. As you drag, Carrara redraws the curve.
   - When you move a curve point handle, the opposite handle moves to remain parallel to the one you are moving.
   - Hold down the Shift key while you drag to move the handle in 45° increments.
   - Hold down the Option/Alt key while you drag to break apart a pair of parallel handles. You can then move each handle independently.

To adjust a curve numerically:

1. Select a point on a cross-section plane. You can only use the Properties tray controls for points on a cross-section plane.

2. In the Properties tray, enter values for Next and Previous Control Handle.
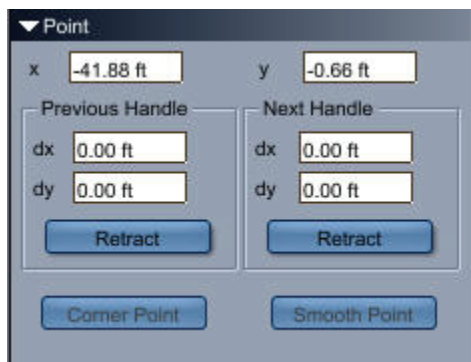
The Next and Previous Control Handle identify the two control points for the selected vertex. The Next and Previous Controls are relative to the selected control point, based on the order in which points were drawn.
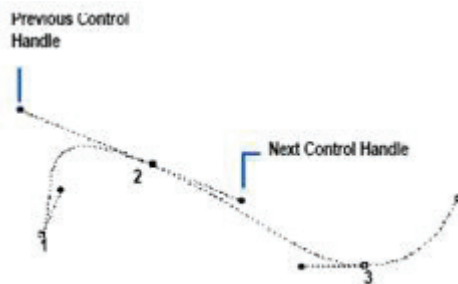
The curve above shows the order in which points were added. Using this order, the Next Control Handle would be to the right of the selected vertex and the Previous Control Handle would be to the left.
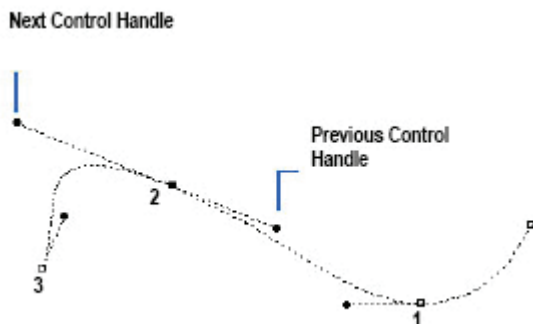
Here the order is reversed so the Next Control Handle would be to the left of the selected point and the Previous Control Handle would be to the right.

The dx and dy values refer to the distance from the selected vertex along the X and Y axis.

Breaking Apart Text Objects

Once you have created text, you can break the text object into individual characters, which you can reposition, rotate, and continue modeling like any other object.

To break apart a text object:

1. Select the text object.

2. Select **Arrange > Convert Text to Outlines**. The letters are separated into discrete cross-section shapes on the Drawing plane.

Working with Shapes

Working on the Drawing plane, you can manipulate shapes and cross-sections in several ways:

- Perform simple operations such as changing the scale, position, and orientation of individual shapes.

- Group shapes in a cross-section to edit them together.

- Compound the shapes on a cross-section to hollow out a 3D object.

This section includes general instructions for working on the Drawing plane. For specific information on cross-sections and sweep paths and how they relate, refer to "Working with Cross-Sections" and "Working with the Sweep Path."

Importing 2D Shapes

Carrara lets you import cross-section shapes, sweep paths, and envelopes in Adobe Illustrator or EPS format. The file can include one or several 2D shapes. You should, however, avoid unnecessary complexity.

Some information from the imported file will be translated or lost:

- Paint characteristics such as stroke and fill are ignored.

- Compound paths are preserved.

- Text is automatically converted to outlines.

For more information on importing files, refer to "Importing 3D Objects."

To import shapes to a cross-section:

1. Click the cross-section plane you want to work with. You can also create a new cross-section plane using **Sections > Create**.

2. Choose **File > Import**. The Import dialog appears.

3. In the dialog, select the file you want, then click **Open**. Carrara places the artwork on the Drawing plane.

4. Click **OK**.

To import a shape as a sweep path or envelope:

1. Choose **Geometry > Extrusion Envelope** to turn on the extrusion envelope. You can use any type of envelope.

2. Click one of the sweep path planes to make it the Drawing plane.

3. Choose **File > Import**. The Import dialog appears.

4. Select the file you want, then click **Open**.

    - If the file you choose contains more than one path, only the first path will be imported.

- Another dialog appears, allowing you to specify whether the path should be used as the sweep path or the envelope.

5. Specify sweep path or envelope.

6. Click **OK**. Carrara imports the sweep path or envelope.

NOTE! Importing a sweep path or envelope replaces the current sweep path and envelope.

Grouping Shapes

You can group one or more shapes using the Group command. Grouping two or more shapes lets you manipulate them all at once.

NOTE! Grouping is allowed only on cross-section planes, not the sweep path.

To group shapes:

1. Select the shape(s) you want to group.

2. Choose **Edit > Group**. A bounding box encompassing the grouped shapes indicates the size and location of the group.

To ungroup:

1. Select the group you want to ungroup.

2. Choose **Edit > Ungroup**.

Scaling Shapes

You can scale a shape or group of shapes directly by using the Resize tool, or numerically by using the Scale dialog. For information on using the Resize tool to scale objects, refer to "3D Object Manipulation Tools."

NOTE! Scaling is allowed only on cross-section planes, not the sweep path.

You can also scale an entire object by scaling the Modeling box. Refer to "Resizing the Modeling Box."

Scaling shapes directly is not always necessary. The extrusion envelope, described in "Using the Extrusion Envelope," can often accomplish the same results more simply and powerfully.

To scale shapes numerically:

1. If necessary, group the curves or shapes you want to resize.

2. Choose **Arrange > Scale**. The Scale dialog appears.

3. Enter horizontal and vertical scale factors.

4. Click **OK**.

Moving Shapes

You can move shapes using standard dragging techniques.

To move shapes manually:

1. If necessary, group the curves or shapes you want to move.

2. Click the Selection tool.

3. Drag the shape wherever you like.

To move shapes numerically:

1. With the Selection tool, click the shape.

2. In the Properties tray, enter values in the top and left fields. Shape position is measured relative to the top and left edge of the Drawing Plane.

Rotating Shapes

You can rotate an object in two ways:

- Free rotate an object in relation to the Drawing plane (2D rotation).

- Rotate an object a specific angle using the Rotation dialog.

The Rotation tool rotates a shape around its center. The Rotation dialog can rotate a shape around either its center or the Drawing plane's center.

NOTE! Rotating is allowed only on cross-section planes, not the sweep path.

To free rotate a cross-section in 2D:

1. Set the Drawing plane to the plane on which you want to rotate the shape. If necessary, group the curves or shapes you want to rotate.

2. Click the Rotation tool.

3. Drag the shape in a circular motion. Hold down the Shift key to rotate the shape in 45° increments.

To numerically rotate a cross-section on the Drawing plane:

1. If necessary, group the curves or shapes you want to rotate.

2. Select the shape(s), and choose **Arrange > Rotate**. The Rotation dialog appears.
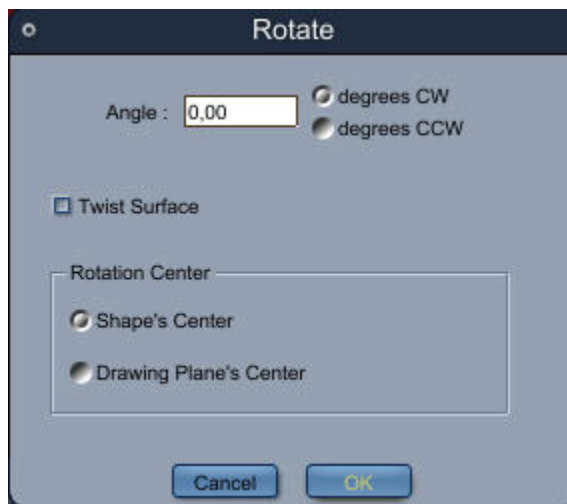
3. Enter the Angle of rotation.

4. Enable degrees clockwise (CW) or counterclockwise (CCW).

5. Enable the Twist Surface option if you want to twist the surface of the object as you rotate. For more information on this option, refer to "Twisting An Object."

6. Enable a rotation center:the shape's center or the Drawing plane's center.

7. Click **OK**.

Beveling Shapes

The bevel menu contracts or expands a shape: it replaces the original shape by one that follows its outline at a given distance.
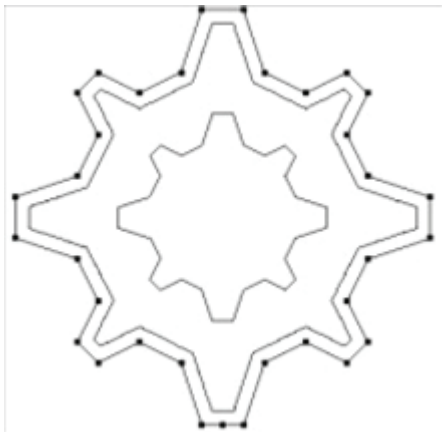
A 0.5 inch bevel was applied to the outer shape to produce the middle shape. In comparison, the inner shape is the result of a 50% scaling of the outer shape.

To apply a bevel to one or more shapes:

1. Select the shapes(s) you want to bevel.

2. If you want to keep the original shape(s), select **Edit > Copy** and then **Edit > Paste**.

3. Choose **Arrange > Bevel**.

4.   Enter a bevel value (a negative value will expand the shapes).

5.   Click **OK**.

Centering Selections

You can center selected shapes on the Drawing plane using the Center Selection command. If several shapes are selected, they maintain their spatial relationship to each other.

Unlike using the Center command in the Sections menu, Center Selection only centers the selected shape, not all shapes in the cross-section. This makes it ideal for creating compound cross-sections.

NOTE! You can also center everything on the cross-section, maintaining the spatial relationships among shapes. For more information, refer to "Centering Cross-Sections."

To center a selected shape on the Drawing plane:

1.   Select the shape or shapes you want to center.

2.   Choose **Arrange > Center Selection.**
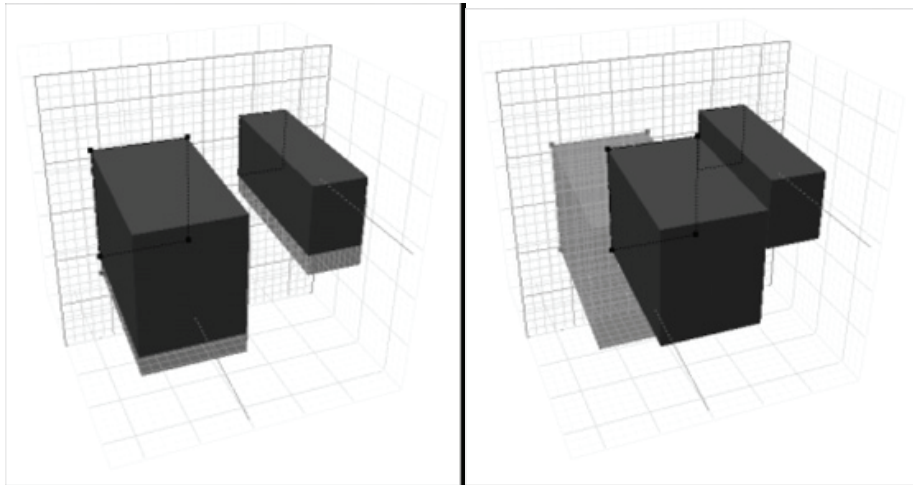
Setting Object Properties

The Properties tray displays the properties of the selected point, control handle, shape, group, or compound cross-section.

You can edit shapes and cross-sections by making numerical adjustments in the Properties tray. When you select a point, Carrara displays the X and Y coordinates.

- When you select a curve point, Carrara displays the X and Y positions of its handles (Control Handle). When these controls are active, you can retract a control handle using the Retract button.

- When you select a grouped cross-section, Carrara shows its height and width, top and left coordinates, and shape number.

- No information is available for ungrouped shapes or for partially selected shapes.

By default, Carrara shows all values in inches. You can change the units of measure using the Preferences: General Scene Settings dialog. For more information, refer to "General Scene Settings Preferences."

To change the properties of a cross-section:

1. Drag open the Properties tray if needed.

2. Select a cross-section group or point.

3. Adjust the values you want.

4. If you selected a closed cross-section, enable the Keep Proportions check box to maintain the shape's aspect ratio as you resize it.

5. If you selected points on a cross-section, do one of the following:

   - Click the Corner Point button to convert the curve point to a corner point.

   - Click the Smooth Point button to convert the corner point to a curve point.

6. If you selected a curve point, click the Retract button to retract the point's handles.

Setting Surface Fidelity

Carrara renders an object by breaking it into hundreds of tiny polygons. This helps the renderer understand the contents of the scene.

The number of polygons Carrara uses for each object is based on the size of the object, its distance from the camera, and the rendering resolution. This calculation usually results in a smooth object.

If an object does not render as smoothly as you'd like, you can increase its surface fidelity. This forces Carrara to break it down into a greater number of polygons.

If none of the objects in your scene renders as smoothly as you would like, you can increase the rendering Silhouette Quality, instead. However, this option uses more memory and increases rendering time. For more information, refer to "Rendering Engines."

The surface fidelity value is resolution-independent. That is, if a particular value yields good results for a given object in a given scene at high resolution, it should yield good results at lower resolutions as well.

To set surface fidelity:

1. Choose **Geometry > Surface Fidelity**. The Surface Fidelity dialog appears.
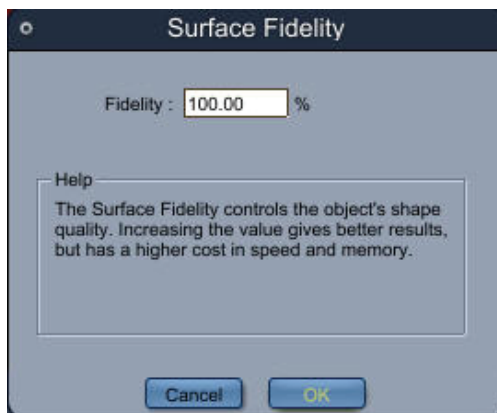
2. Enter a percentage to increase or decrease the object's surface fidelity. The default value is 100%.

3. Click **OK**.

Working with Cross-Sections

A basic spline object, such as a box, has one cross-section at the start of the path. The shape on this first section is swept to the end of the extrusion path. To create more complex forms, you can add any number of cross-sections along the path. Because each cross-section, in turn, can contain any number of shapes, you can use multiple cross-sections to model intricate objects.

As you begin to work with the Spline modeler, you will find that the sweep path and cross-sections are closely tied. Although separated here for the sake of clarity and organization, these concepts go hand in hand. Be sure to read "Working with the Sweep Path."

Creating and Removing Cross-Sections

Creating and Removing Cross-Sections

You can add as many cross-sections as you like to an object. Each cross-section must correspond to a point on the sweep path. If you want to create a cross-section where there is no point, you must first add a point.
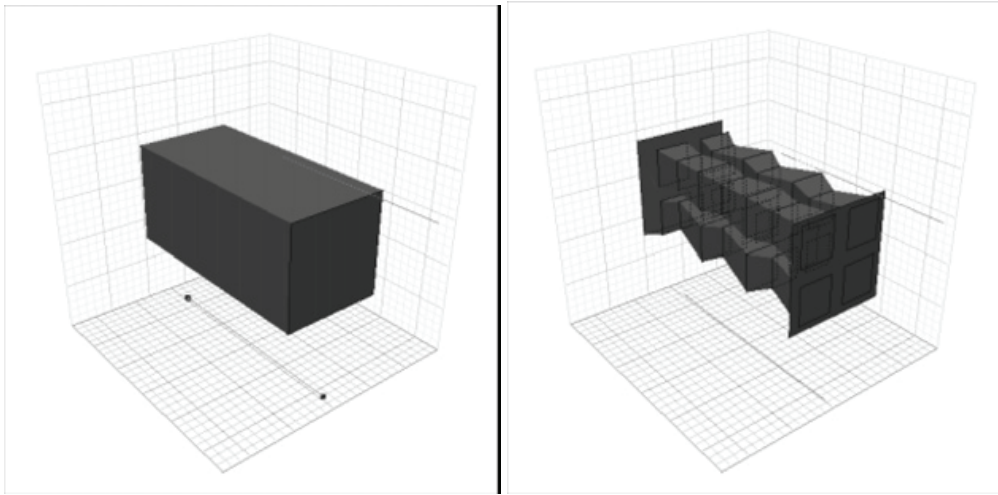
Although modeling with multiple cross-sections enables you to achieve otherwise impossible effects, be careful not to add cross-sections unnecessarily. Editing an object with too many cross-sections can become difficult and tedious. Often, you can accomplish similar results using the extrusion envelope. Refer to "Using the Extrusion Envelope."

To create a new cross-section:

- If there is a point on the sweep path where you want the new cross-section, select the sweep path and choose **Sections > Create**.

- If there is no point, choose the **Add Point** tool. Hold down the **Option/Alt** key and click the sweep path at the point where you want to create the new cross-section. Carrara adds a point to the sweep path and creates a new cross-section at that point.

- If you're working on a cross-section plane, choose **Sections > Create**.

- If there is a shape on the previous cross-section, Carrara copies it to the new cross-section. You can delete the new shape without deleting the new cross-section.

To remove a cross-section:

1. Click the cross-section you want to remove, or select the corresponding point on the sweep path.

2. Choose **Sections > Remove**.

To remove a cross-section and its sweep path point:

- Use the Delete Point tool to delete the point from the path.

Generating Intermediate Cross-Sections

You can have Carrara create a specific number of cross-sections between the current cross-section and the next one.

To generate intermediate cross-sections:

1. Click a cross-section plane.

2. Choose **Sections > Create Multiple**. The Create Multiple Cross-Sections dialog appears.
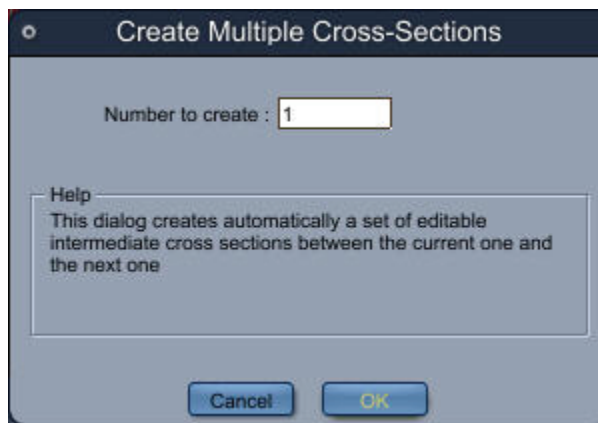
3. Enter the number of sections you want.

4. Click **OK**.

Carrara creates the intermediate cross-sections, spacing them evenly between the current cross-section and the next one. Carrara adds a new point to the sweep path for each cross-section it creates.

The shapes on the new cross-sections are interpolated from the shapes on the existing cross-sections. This process is similar to blending between two shapes in a 2D illustration program:each shape on the new cross-sections is like one "step" in the blend.

Moving Among Cross-Sections

If you have multiple cross-sections, you can move the Drawing plane to any cross-section.

To move to an adjacent cross-section, do one of the following:

- Choose **Sections > Next** or **Previous**.

- Hold down the **Command/Ctrl** key and press the right or left arrow key to move to the next or previous cross-section.

To move to a specific section:

1. Choose **Sections > Go to**. The Go To cross-section dialog appears.
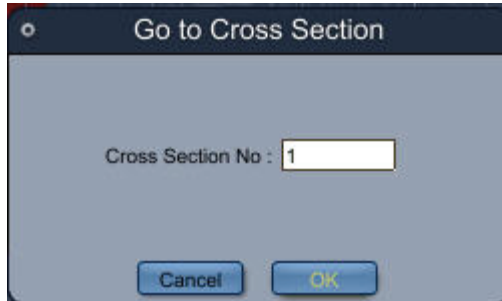
2. Enter the number of the section you want to go to. Cross-sections are numbered from back to front.

3. Click **OK**.

Displaying Cross-Sections

If you have multiple cross-sections, you can display them all at once or show only the current one. Displaying only the current one gives you a better idea of the appearance of your object; displaying all lets you easily identify the cross-section shapes you have to work with.

To show all cross-sections or only the current one:

- Select **Sections > Show> Current** or **All**.

Centering Cross-Sections

Often when you're creating cross-sections, you can end up with a number of sections whose centers aren't aligned, which can lead to unexpected results. It's usually a good idea to center sections when you're finished drawing them.

To center a cross-section on the sweep path:

1. Select the section you want to center.

2. Choose **Sections > Center**. The shape(s) on the cross-section will be centered around the sweep path.

NOTE! You can also center specific selections. For more information, refer to "Centering Selections."

Compounding Shapes

Compounding shapes is like grouping them, with one significant difference:any shape completely enclosed by another in the same compound "cuts away" from the larger shape. This leaves a hollowed out shape, which can be extruded to produce an object such as a pipe or a hollow log.
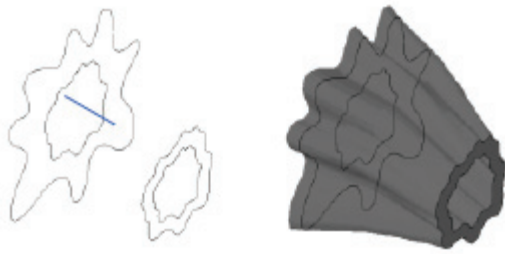
To compound shapes:

1. Select the shapes you want to compound.

2. Choose **Arrange > Combine as Compound**.

To break apart a compound:

1. Select the compound you want to release.

2. Choose **Arrange > Break Apart Compound**.

 NOTE! Compound shapes are allowed only on cross-section planes.

Filling and Emptying Cross-Sections

You can fill a cross-section to make an object "solid." For example, a cylinder with its endcaps filled would be something like a solid can; a cylinder with its endcaps emptied would be a tube that you could look through.

An object's first and last cross-sections are often referred to as endcaps:you can turn endcaps "on" by filling them or "off" by leaving them unfilled.

To fill and empty cross-sections:

1. Click a cross-section plane.

2. Choose **Sections > Cross-Section Options** to open the cross-section Options dialog.
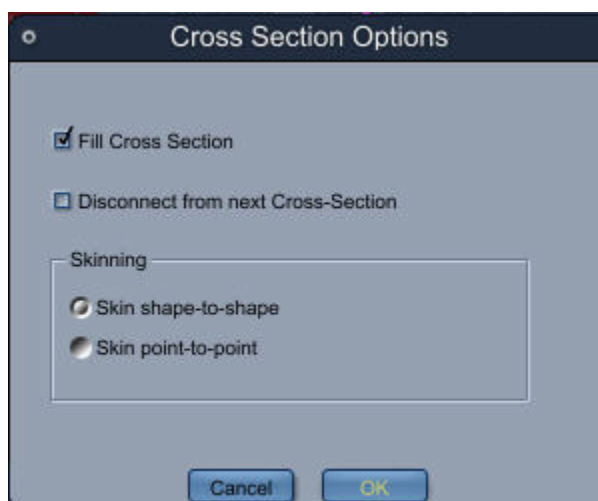
3. Enable Fill Cross-Sections to create solid sections, or disable the option to create empty sections.

You can also use the cross-sections dialog to disconnect cross-sections and choose skinning options. For more information, refer to "Disconnecting Cross Sections."

Modeling with Multiple Cross-Sections

Modeling with multiple cross-sections lets you increase your control over the shape of your object's surface.

With each section you add, you increase your control over how the surface is created. You can add subtle variations in your object's surface by creating more cross-sections with different shapes.
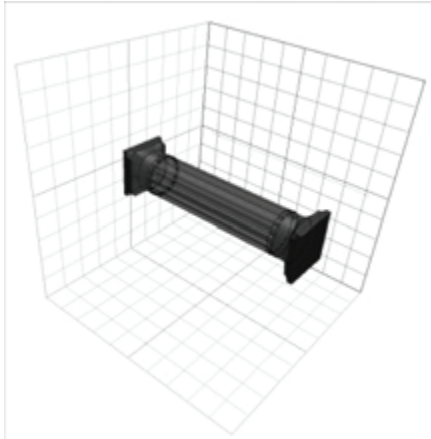
Of course, the more cross-sections you add, the more complicated it becomes to work in the modeler. However, the Spline modeler has some features to help you model with multiple cross-sections, described in this section.

Numbering Shapes

Carrara automatically numbers the shapes in each cross-section, establishing a correspondence between the shapes in adjacent cross-sections. In cross-sections containing multiple shapes, this shape numbering determines which shape sweeps to which other shape.

You can renumber the shapes to change the shape-to-shape correspondence. Each shape's number is displayed in the Properties tray.
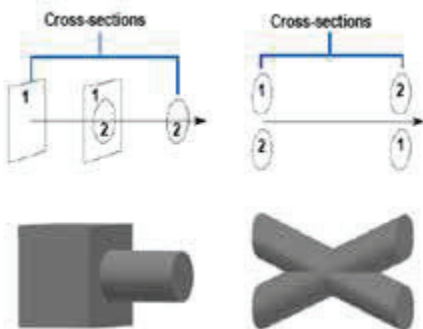
To control shape-to-shape correspondence:

1. Select a shape to number. You must select every point on the shape, or regroup the shape.

2. Do one of the following:

    -   Choose **Sections > Set Shape Number**. The Shape Number dialog appears.
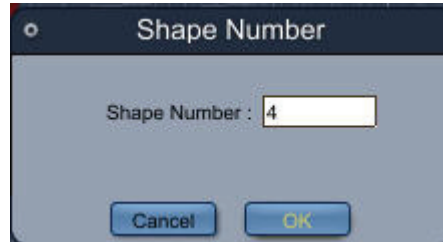
    -   Display the Properties tray.

3. In the Shape Number field, type in the number you want.

    -   If you enter the number of another shape in the same plane, Carrara swaps those numbers.

    -   If you enter a unique number, make sure you assign the same number to the appropriate shape in the adjacent cross-section(s).

    -   Carrara does not extrude any shape that has no correspondence (that is, its number does not match any shape number in an adjacent section).

4. Click **OK**.

Disconnecting Cross-Sections

You can create multiple and "intermittent" objects by disconnecting adjacent cross-sections.
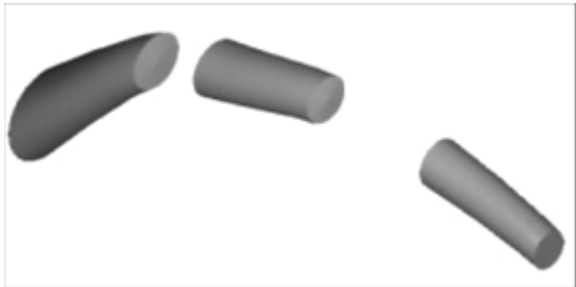
To disconnect cross-sections:

1. Select the cross-section you want to disconnect.

2. Choose **Sections > Cross-Section Options**, or open the properties.

3. Enable the **Disconnect from next cross-section** checkbox.

    Carrara turns off extrusion between the selected cross-section and the next one.

Setting Skinning Options

The Spline modeler automatically extrudes between each cross-section and the next, basing the surface of the object on the shapes in the cross-sections and on the skinning options.

The skinning options make your surface either sharp-edged or smoothed.

Use shape-to-shape
skinning.

Use point-to-point
skinning.
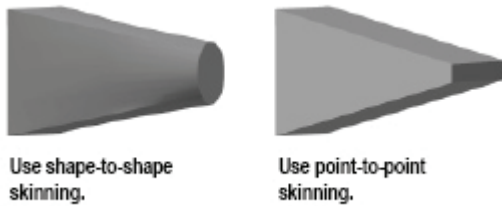
To set skinning options:

1. Choose **Sections > Cross-Section Options** to open the cross-section Options dialog, or open the properties.

2. Set a Skinning option:

    - **Skin shape-to-shape** is especially well suited for creating smooth, organic surfaces whose cross-section shapes are significantly different from one another. In fact, if adjacent cross-sections have different numbers of vertices, shape-to-shape skinning is the only option available.

    - **Skin point-to-point** can be used when adjacent cross-sections contain very similar shapes and you want each vertex in one cross-section to be connected directly to the corresponding vertex in the next cross-section. This option is useful when you model an object that requires straight, sharp edges.

3. Click **OK**.

Twisting An Object

When you rotate a cross-section shape, you can specify that the rotation be applied to the surface of the object to give it a twisted appearance.

To twist an object:

1. Select a cross-section.

2. Hold down the Option/Alt key while rotating the cross-section with the Rotation tool.

3. Hold down the Shift key to rotate the shape in 45° increments.

To twist an object numerically:

1. Select a cross-section.

2. Choose **Arrange > Rotate**. The Rotation dialog appears.

3. Specify the direction and degree to rotate the selection.

4. Enable the Twist Surface option.

    - Carrara twists the surface the specified number of degrees.

    - Use rotation values greater than 360° to specify multiple twists.

Adding Bevels to a Cross-Section

With controls very similar to those used by the text object (as discussed in "Creating Text Objects"), you can add a front and a back bevel to a cross-section. The shapes in the beveled section will stay the same, but one or two automatic cross-sections will be created. In those sections, new shapes will be automatically calculated from those contained in the beveled section, and updated if you modify the originals.

To bevel a cross section:

1. Select the cross section that you want to bevel. The Properties tray displays the options available for that section.
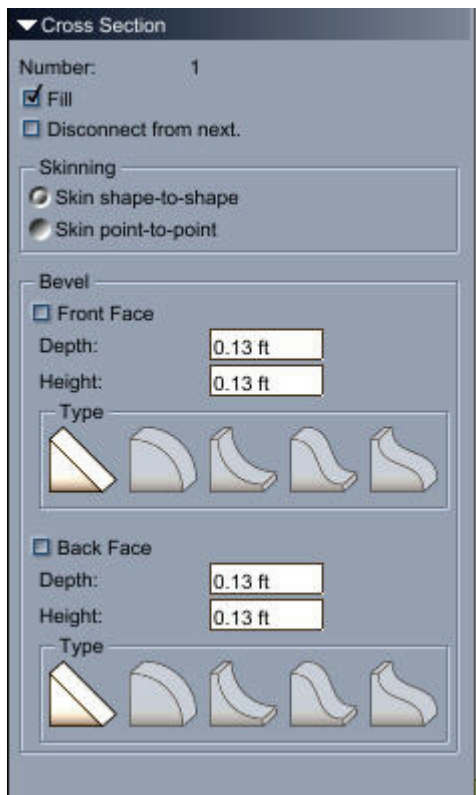
2. Click the Front Face box if you want to add a bevel between the selected section and the next point on the sweep path.

3. Click the Back Face box if you want to add a bevel between the selected section and the previous point on the sweep path.

4. Enter a Depth value for the front/back face. It determines the size of the bevel along the sweep path. This value must be smaller than the distance between the selected section and the next/previous sweep path point.

5. Enter a Height value for the front/back face. This determines the size of the bevel along the cross section plane. A negative value will expand the section's shapes. A positive value will contract them. When using a positive value, it should be much smaller than the size of the shapes in the section.

6. Select a bevel Type for the front/back face. This determines the shape of the bevel by modifying the envelope of the object.

TIP! To get better results, you should set the point-to-point skinning option on the beveled cross-section.

Working with the Sweep Path

The key to working with the sweep path is understanding how the 2D path description lines define the 3D sweep path. There are two path description lines, one on each sweep path plane. Although the path description lines appear to be two separate planes, they are actually projections of the same path.
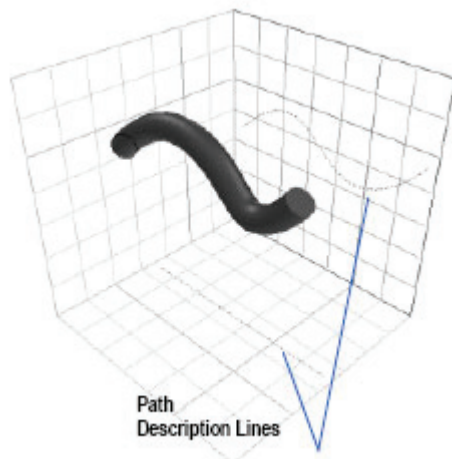
Each point on one path description line corresponds to a point on the other path description line. In fact, each pair of points represents a single 3D point on the sweep path.

The sweep path projection on the ground plane describes the sweep path's horizontal position; the projection on the side plane describes its vertical position.
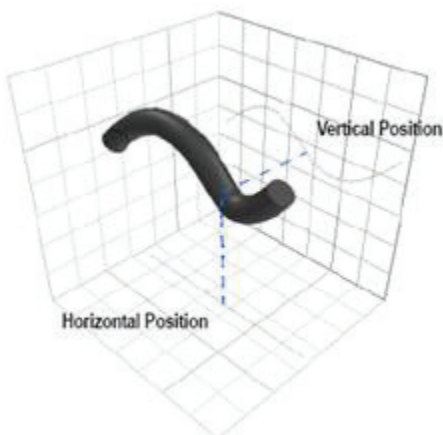
A sweep path curved on the side plane but straight on the ground plane would create an object whose shape curves up and down, but not side to side.
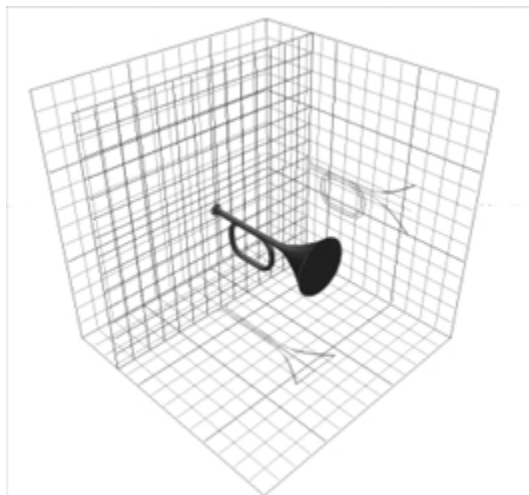
A path that curves on both planes would create an object whose shape curves both side to side and up and down.
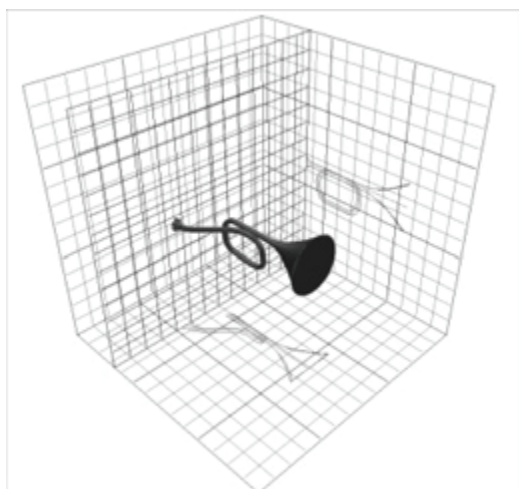
Drawing a Sweep Path in 3D

The best way to start drawing a sweep path is to consider how you want your object to look, both from the side and from the top. An object that has a lot of curves when viewed in profile, but looks straight when viewed from the top, indicates that the sweep path will remain straight on the ground plane and curved on the right plane.

Unfortunately, few objects are this simple. Most objects have some variation from both the top and the profile. You can easily draw this type of sweep path by adjusting the point on the two sweep path projections.

In some cases, simple adjustment might not be enough. You might want to draw the sweep path continuously from the first point to the last, switching back and forth between the two planes as you work.

Cross-Sections and the Sweep Path

When you're working with the sweep path, it's important to understand the relationship between the path and the cross-sections. You can think of the sweep path as an object's spine. Whenever you move a point on the sweep path, any cross-section at that point will move as well.

The opposite is not true, however. When you drag a cross-section shape along the cross-section plane, the sweep path is not affected; you are simply moving the cross-section off its center.
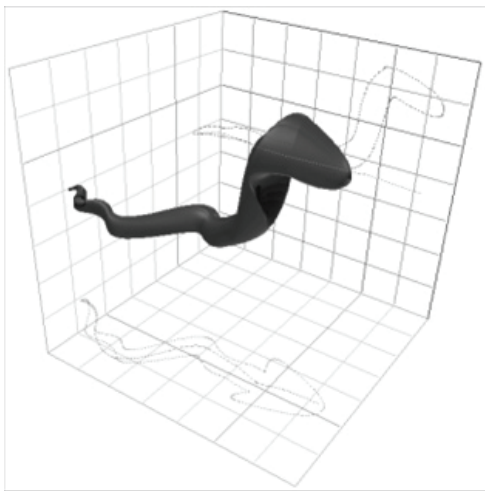
Cross-sections don't have to be centered around the sweep path. You can move a cross-section off the center of the sweep path.

Translation vs. Pipeline

The Spline modeler can extrude cross-sections along the sweep path using one of two extrusion methods: translation or pipeline.

With the translation method, each cross-section remains perpendicular to the ground plane of the Spline Modeling window, regardless of the sweep path's curvature.

With the pipeline method, the orientation of each cross-section depends on the direction of the sweep path at the point where the cross-section is located:all cross-sections remain perpendicular to the sweep path, rather than to the ground plane.

Translation Method   Pipeline Method
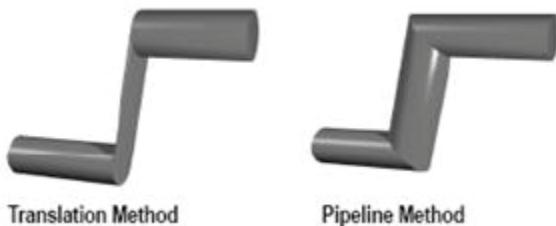
NOTE! Changing methods can reset your sweep path and extrusion envelope.

To choose an extrusion method:

- Choose **Geometry > Extrusion Method> Translation** or **Pipeline**.

TIP! With the pipeline extrusion method, using the Current Section view makes working on the shapes of the current cross-section much easier..

Using Extrusion Presets

Carrara has three extrusion presets, which automatically generate specific types of sweep paths. When you apply an extrusion preset, Carrara replaces the current sweep path and envelope with the preset path.

To straighten the sweep path:

- Choose **Geometry > Extrusion Preset> Straight**.

Carrara moves all of the points on the sweep path into a straight line. Cross-section shapes are unaffected.

Working with Spirals

When you use the Spiral extrusion preset, only the first cross-section of your existing object is kept:the remaining cross-sections, sweep path, and envelope are all replaced.

To make a spiral:

1. Draw a 2D shape on the cross-section plane and select the shape.

2. Choose **Geometry > Extrusion Preset> Spiral**. The Spiral dialog appears.
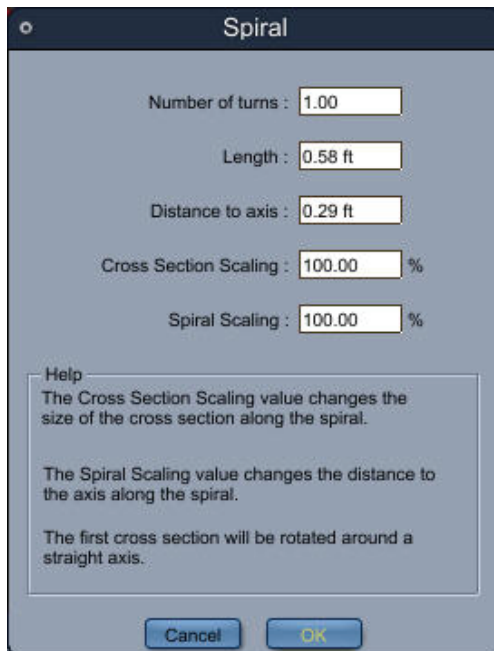
3. Set the options you want.

- **Number of turns** is the number of loops in your spiral. One turn equals 360°.

- **Length** determines the length of the sweep path. This affects how tightly the spiral will be wound.

- **Distance to axis** determines the size of each coil by setting the distance between the cross-section shape and the spiral's central axis. The relationship between the number of turns and the length determines how tightly the coils of the spiral are spaced.
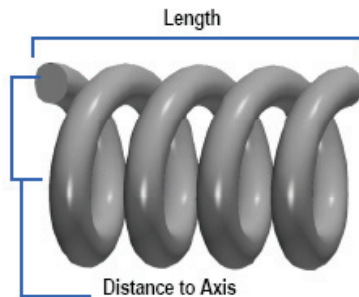
- **Cross-Section Scaling** lets you shrink or enlarge the cross-section shape as it sweeps along the spiral path. Values less than 100% shrink the shape; values greater than 100% enlarge the shape.

- **Spiral Scaling** lets you decrease or increase the distance to the axis as the cross-section shape sweeps along the path. Values less than 100% taper the spiral; values greater than 100% widen it.

4. Click **OK**. Carrara creates the spiral sweep path you have specified and automatically switches to pipeline extrusion.

The Torus Extrusion Preset

The Torus extrusion preset automatically creates a perfect circular sweep path. For more information, refer to "Lathing."

Using the Extrusion Envelope

To a large extent, the contours of a spline object are determined by the cross-sections you place along the sweep path. Wherever you change the size or shape of a cross-section, the object's surface changes accordingly.

To create some complex objects, you might find that you'll need a finer degree of control than cross-sections alone can provide. The extrusion envelope lets you specify how an object's surface should curve from one cross-section to the next, giving you precise control over the object's form.

For the best results, you should model the object as completely as possible using the sweep path and cross-sections, then adjust the extrusion envelope as a final step.

Understanding the Envelope

By default, Carrara does not use the extrusion envelope: it stretches the object's surface over the cross-sections as simply as possible. When you turn the envelope on, it appears as four description lines, two on each sweep path plane. (The different colored line on each sweep path plane is a path description line:refer to "Working with the Sweep Path").

Initially, the envelope conforms to the dimensions of the object's cross-sections, widening and narrowing if the cross-sections vary in size.

Envelope Description Lines
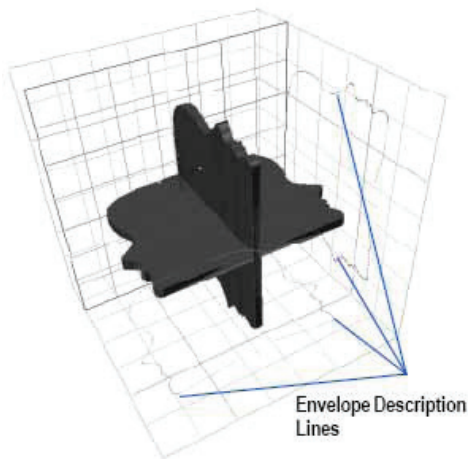
The envelope description lines are Bézier curves. By editing these curves, you can alter an object's contours. The envelope has three modes: Symmetrical, Symmetrical in Plane, and Free.

With the Symmetrical envelope setting, all the envelope lines maintain symmetry.
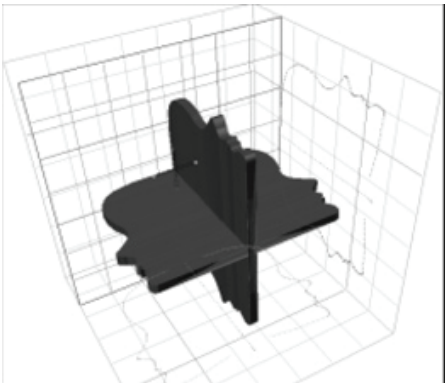
Symmetrical in Plane lets you edit the envelope description lines in pairs.
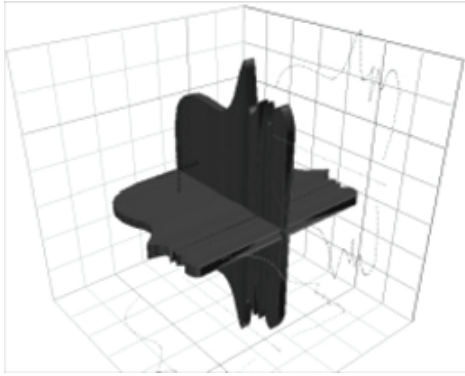
Free lets you edit each line individually. This allows you to model asymmetrical objects.
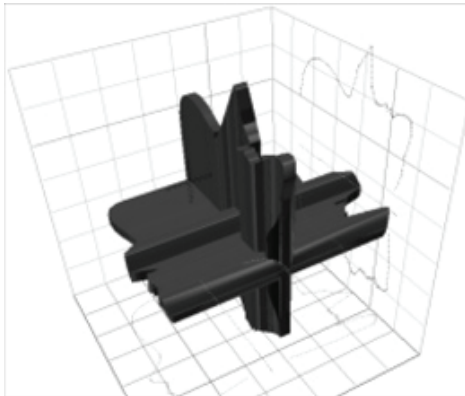
How the Envelope Relates to the Sweep Path

Each point on the extrusion envelope corresponds to a point on the sweep path. When you add a point to the extrusion envelope, Carrara also adds a point to the sweep path. Likewise, when you delete a point from the envelope, Carrara deletes the corresponding point from the sweep path.

Moving an envelope point perpendicular to the sweep path controls the scaling of the object at that point on the path. Moving an envelope point parallel to the sweep path moves the corresponding point on the sweep path as well.

When you are using the pipeline extrusion method, envelope points can move only perpendicular to the sweep path.

How the Envelope Relates to Cross-Sections

Editing the extrusion envelope can also affect an object's cross-section shapes. If you edit the envelope at a point where there is no cross-section, only the surface between cross-sections is affected. However, if you edit the envelope at a point where a cross-section exists, the shapes on the cross-section are scaled accordingly. If you delete a point from the envelope, you will also delete any cross-section located at that point.

When you use the extrusion envelope on multiple-shape cross-sections, the center point is the sweep path:not the center of each shape. If you want to scale a cross-section's individual shapes around their own center points, you should resize each shape individually on the cross-section plane rather than use the extrusion envelope.

Using the Envelope

To use the envelope, you turn it on and set a symmetry constraint other than the default of None. After that, you can directly modify it.

To use the envelope:

- Choose **Geometry >Extrusion Envelope >**and select one of the options described below. If the envelope is not already enabled, the four envelope description lines appear on the sweep path planes.

    - **Symmetrical** uses the same curve for all four envelope description lines:when you edit one line, the others update automatically.

    - **Symmetrical in Plane** uses the same curve for the description lines in either plane:when you edit one line, the other in its plane updates (as a mirror image) automatically.

    - **Free** allows each description line to have a unique curve.

To modify the extrusion envelope:

- Use the drawing tools to edit the envelope description lines.

    - You can add and delete points, as well as adjust existing points and control handles. As you adjust one of the envelope description lines, the other lines are updated in real time.

    - For instructions, refer to "Editing Shapes."

To reset the extrusion envelope:

- Choose **Geometry > Reset Envelope**. Or, choose **Geometry > Extrusion Envelope > None**.

Lathing

Lathing lets you create symmetrical objects by rotating a 2D profile around a straight axis. Rotation can be circular or angular, a full 360º or less.

Because your lathe object is actually a spline object, you can edit it in ways that traditional lathing tools do not allow. For example, you can create a symmetrical lathe object, then deform it using the Spline modeler's other tools.

You can choose from two different lathing methods, which are described next.

Lathing with the Extrusion Envelope

You can create nearly any lathe object by extruding a circle or a regular polygon and using the extrusion envelope to draw the object's lathe profile.

For more information on the extrusion envelope, refer to "Using the Extrusion Envelope."

Cross-section

Envelope Description Line
(Lathe Profile)
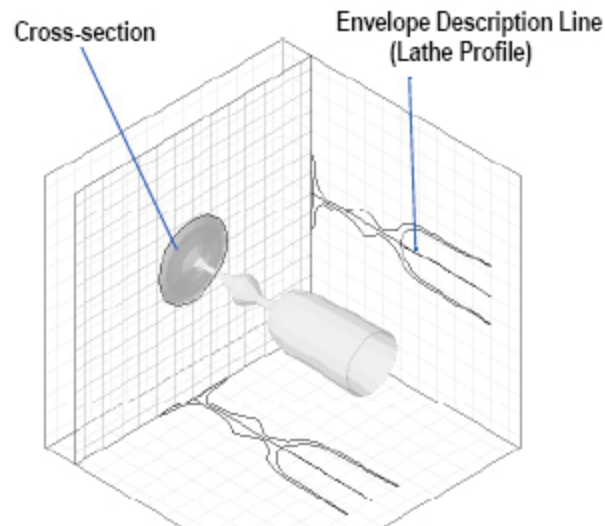
To create a lathe object with the extrusion envelope:

1. Click a cross-section plane.

2. Draw a cross-section using one of the Spline primitive tools or the Pen tool. Use an appropriate size and shape for the object you are creating. The initial diameter of your object depends on the size of the cross-section. For example, the size of the first circular cross-section determines the diameter of the glass's base.

3. Hold down the **Shift** key to create circles or squares.

    o   Choose **Sections > Center** to center the cross-section on the sweep path.

    o   Choose **Geometry > Extrusion Method> Translation**.

    o   Choose **Geometry > Extrusion Envelope> Symmetrical**.

        Don't worry about trying to edit the envelope's description lines:they disappear when you start drawing your own lathe profile.

4. Click one of the sweep path planes to make it the Drawing plane. You might find it easier to draw the lathe profile in a 2D view.

5. Click an empty space on the plane, being careful not to select the sweep path or either of the envelope description lines.

6. Using the Pen tool, place the second point of the lathe profile. To create a curve point, drag the Pen tool.

Click here first.
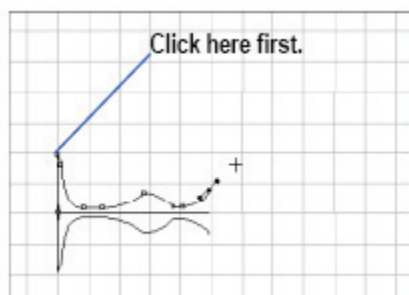
7. Make sure you draw the lathe profile above the sweep path.

8.   Continue drawing the lathe profile, placing additional points with the Pen tool.
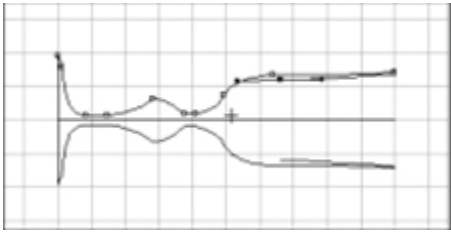
9.   You can also use the drawing tools to edit the points you have already placed. For best results, be careful not to cross over the sweep path as you draw the lathe profile.

Lathing with a Circular Sweep Path

Some lathe objects are difficult to create with the extrusion envelope. An object with a hole in the center can be particularly difficult. To create this type of object, you can draw the lathe profile in the cross-section plane and sweep it around a circular sweep path.

The Torus extrusion preset, available from the Geometry menu, creates a precise circular path.

Metaball Modeling (Standard, Pro)

The Metaball modeler lets you create organic shapes by blending primitives called blob elements. You use **negative** and **positive** blobs to build an object by adding and subtracting primitives, and then finish it with a smooth or a spiky surface. This makes the Metaball modeler ideal for objects such as water puddles, teddy bears, and bushes. You can also open a metaball object in the Vertex modeler and use vertex modeling techniques to define it further.

Metaball modeling lets you fashion organic-looking objects as if you were manipulating clay. You use blob primitives to shape a metaball object, which is unified by a smooth or spiky skin. The blob primitives are not solid 3D primitives, but are defined by a 3D influence field that blends itself with surrounding blob primitives. The blending defines a skin, computed in real-time by the Carrara Metaball modeler.
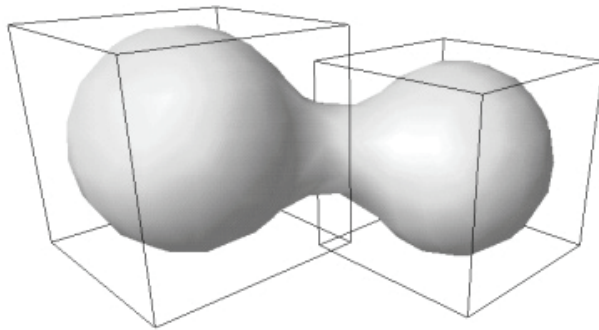
Unlike the geometric primitives in the other modelers, blob primitives can be **negative** or **positive**.

Positive elements add to an object's volume. They are said to be **attractive**, since they blend when they get close together. You set the **surface threshold** to control just how close positive elements must be before they blend. The following image shows surface thresholds of 20% and 80%.
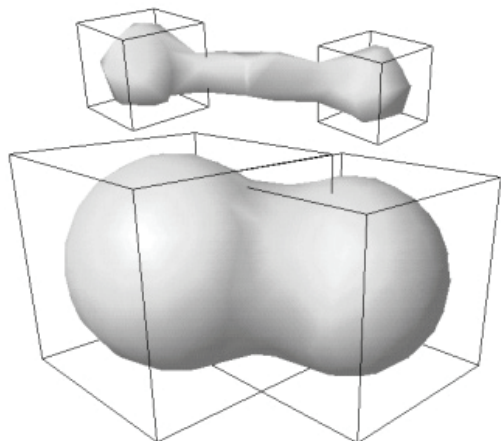
Negative elements are **repulsive**. They push away the skin surface of any surrounding blob elements, making smooth wells or holes in the object. The surface threshold also controls how close negative blobs must be before they push away surrounding blobs.
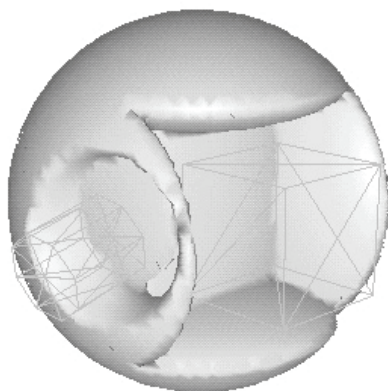
This chapter covers the concepts involved in metaball modeling and describes how to use the Metaball modeler.

The Metaball Modeling Window

The Metaball modeler is where you create metaball objects.
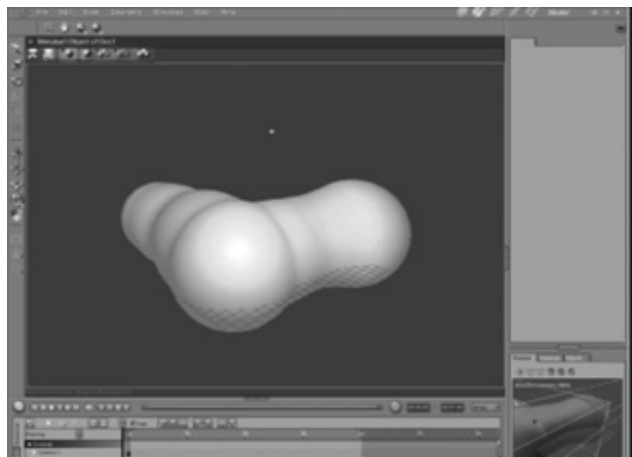
When the Metaball Modeling window opens, new menu commands are available, different tool sets appear, and your view is replaced with a close-up view of the object.

The Metaball Modeler's 3D View

The 3D view of the Metaball modeler's working box has a dark background and colored grids. The origin of the universe is indicated by a trident showing the X, Y and Z axes.
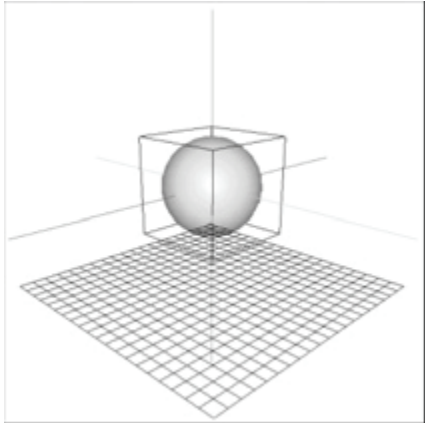
When you move an object, another triplet of colored lines appears to indicate the starting point of the object. In addition, whenever you move an object lines extend from its center to show its movement along each axis.
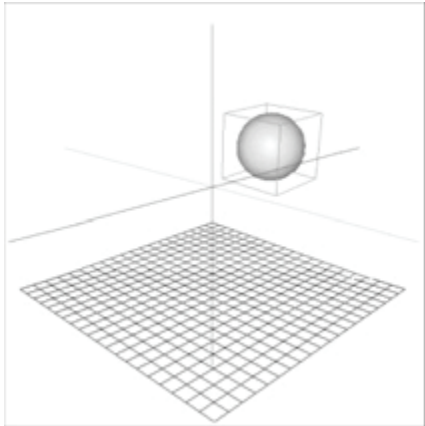
The Ghost Menu

The Ghost menu is a set of interactive commands that appear whenever you right -click (Win) or Control-click (Mac) anywhere in the modeler. The menu remains visible as long as you press the mouse button. You select commands by rolling over them and then releasing the mouse button.

Once a command is selected, it changes the behavior of the cursor. For example, if you select one of the Primitive commands on the Ghost menu, the next click you make adds a primitive blob.

The menu contains commands for adding both positive and negative primitives, moving, scaling and rotating objects as well as commands for viewing your metaball object.
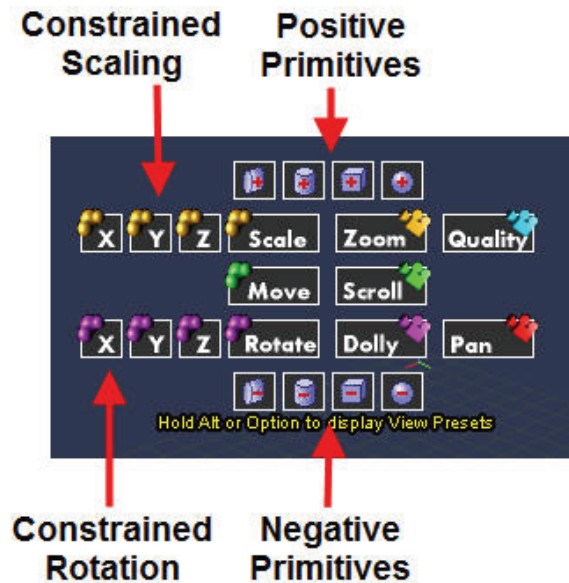
You can also access the Alternate Ghost Menu, which lets you quickly switch between view of your scene. Refer to "Working with the Modeling Window" for more.

To use the Ghost Menu:

- Command/Right-click and roll over the command you want and release the mouse button.

- Option-Command/Alt-Right-click and roll over a view.

Setting Preferences for the Metaball Modeler

In the Metaball modeler Preferences dialog, you can set preferences for the grid, for workspace and object colors, for naming objects, and for selecting blobs in an object.

To set preferences for the Metaball modeler:

1. Choose **File > Preferences**. The Preferences dialog appears.
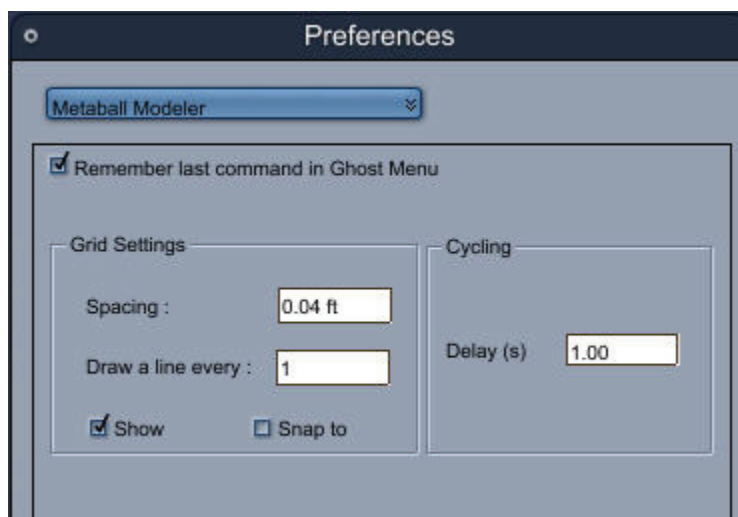
2. Choose **Metaball Modeler**.

3. Set the options you want and click **OK**.

The options for the Metaball modeler preferences are as follows:

- **Remember last command in the Ghost Menu** lets you store the last command you accessed through the Ghost Menu. The next time you open the Ghost Menu, the last selected menu item will be displayed right under your cursor.

- **Grid settings** Sets up the grid for the Metaball modeler without affecting the grid settings used for the Assemble window and other modelers.

  o **Spacing** Adjusts the spacing between grid lines in the box, using the unit of measure you choose.

  o **Draw a line every** Controls how often grid lines are drawn.

  o **Show** When enabled, shows the grid lines.

  o **Snap to** When enabled, causes objects to "snap" to the nearest grid line as you drag them.

- **Cycling** Sets the amount of time before Carrara selects the next blob for Selection Cycling. For more information about Selection Cycling, refer to "Selecting Blobs."

Opening the Metaball Modeling Window

You open the Metaball Modeling window by doing one of the following:

- Double-click the object in the Sequencer or the scene window.

- Select a metaball object, then click the Model button.

When you're finished modeling, you can save your changes and enter a different room by clicking the appropriate room button.

NOTE! To avoid the memory-consuming accumulation of windows, always close the modeling window when you are finished modeling.

To create a new metaball object:

1. Open or create a document in the Assemble room. (For more information, refer to "Creating Scenes.")

2. Depending on the desired location of the metaball object, do one of the following:

  o To insert a metaball object of default size at the center of the universe, choose **Insert > Metaball Object** or drag the Metaball Object tool into the Sequencer.

  o To insert a metaball object in any other location, drag the Metaball Object tool to into the scene window to any location.

Working with the Modeling Window

You can use the Alternate Ghost Menu to switch between the 3D view and the six isometric views.
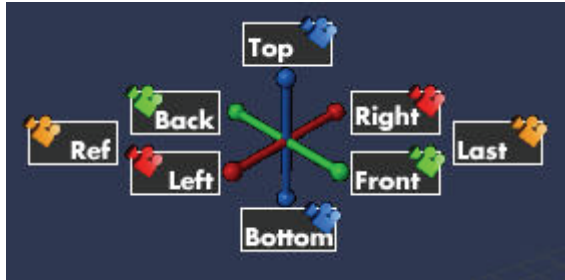
You can work in any view you want. For instance, you may find it convenient to work in a 2D view when aligning blobs, then switch to the 3D view when you want to work in 3D.

To switch between 3D and 2D views:

1. Hold Option/Alt while Command/right-clicking to activate the Alternate Ghost Menu.

2. Roll over the view you want to work in and release the mouse button.

To restore the 3D view to the default:

- On the Alternate Ghost Menu, roll over Ref.

To restore the 3D view to the previous camera position:

- On the Alternate Ghost Menu, roll over Last.

    o If your last view of the 3D window was dollied, panned, or zoomed, the camera position will be the same.

To make your work flow easier, you can change different features of the Metaball modeling window:

- Change the display and behavior of the grid. For more information, refer to "Setting Preferences for the Metaball Modeler."

- Change the Drawing plane (which follows).

- Show and hide the Working Box's planes.

Changing the Drawing Plane

To make your work easier, you can change the Drawing plane.

To change the Drawing plane:

1. Click the Selection tool.

2. Click a plane in the Working Box control to switch between the planes.

Changing Your Point of View

Without changing your point of view to examine an object from all sides, it can be difficult to determine its position relative to other objects in the model. When you blend objects this is particularly important:you must be sure that the objects are actually overlapping for blending to work.

The Metaball modeler also lets you dolly while moving an object.

To dolly while moving a selection:

1. Hold the Command/Ctrl key while dragging.

2. Release the mouse key and continue dragging to move the object.

Previewing Objects

There are several ways in which you can preview objects in the Metaball modeler:

- Choose surface preview quality.

- Choose shading preview mode.

- Set preview precision.

- Assign colors to blobs. This is useful for distinguishing blobs from one another in a blended object.

- Turn the display of negative blobs on or off.

The shading preview mode is independent of the surface preview quality. You can choose between shading previews of Wireframe and Gouraud Shaded.

Note Unless they are turned off altogether, negative blobs always display as Wireframes.

- In **Primitive Preview**, Carrara displays no positive or negative interactions between blob elements. Primitives behave as they do in the Spline modeler, passing through each other when they are placed on the same coordinates. The following images show negative and positive blobs in Wireframe and Shaded mode.
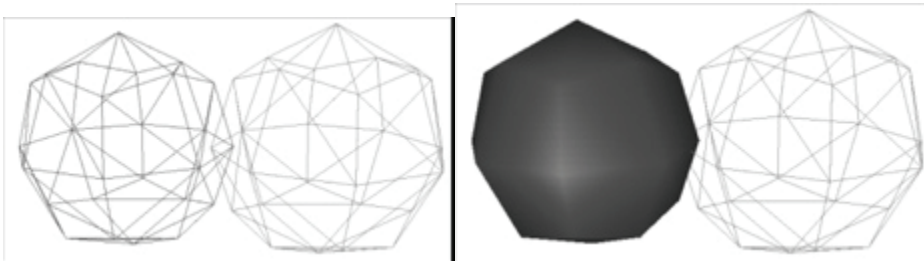
- In **Particles Preview**, interactions between blob primitives are shown and the surface is defined by small triangles, or particles.

- In **Mesh Preview**, interactions between blob primitives are shown and the surface is defined by a mesh. Objects display in white; individual blob colors do not show. The following images show negative and positive blobs in Wireframe and Shaded mode.
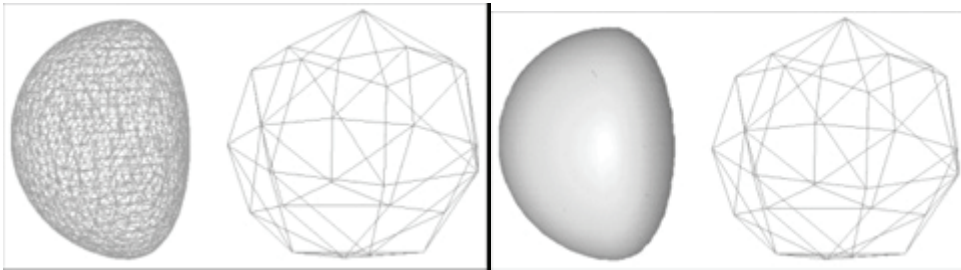
To change the preview quality:

- Choose the Preview Quality tool you want.
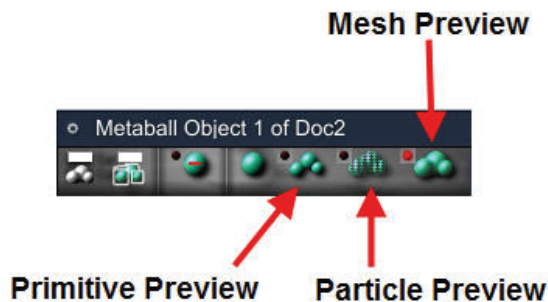
To set preview precision:

1. Drag left or press (-) on the numeric keypad to decrease the precision.
2. Drag right or press (+) on the numeric keypad to increase the precision.

    - In Particles Preview, increasing the precision increases the number of particles in an object.

    - In Mesh Preview, increasing the precision increases the number of polygons used to describe the surface of the object.

    - In Primitive Preview, the precision settings have no effect.

    - Higher rates of precision may slow the redraw rate.

To change the color of blobs:

1. Select the blobs) you want to change.
2. Choose the Assign Color tool. Roll out to the color you want and release the mouse button. All objects you subsequently create will use this color.

 NOTE! In Mesh Preview mode, objects appear white regardless of the color you apply.

To toggle the display of negative blobs:

- Choose the **Display Negative Blobs** tool.

When you turn off the display, you are still able to perform all modeling and editing functions using negative blobs. Selected negative blobs will show their bounding boxes.

Creating Metaball Objects

This section discusses how to use the Metaball modeling tools.

Inserting Blob Primitives

The first step in creating a metaball object is to insert one or more blob primitives.

To insert a blob primitive:

1. Choose the negative or positive pop-up, depending on which type of primitive you want to insert.

2. On the Drawing plane, drag from one corner of the blob to the opposite corner. Release the mouse button when the blob is the desired size. The blob is inserted on the Drawing plane.

NOTE! You can also insert blob primitives from the Ghost Menu.
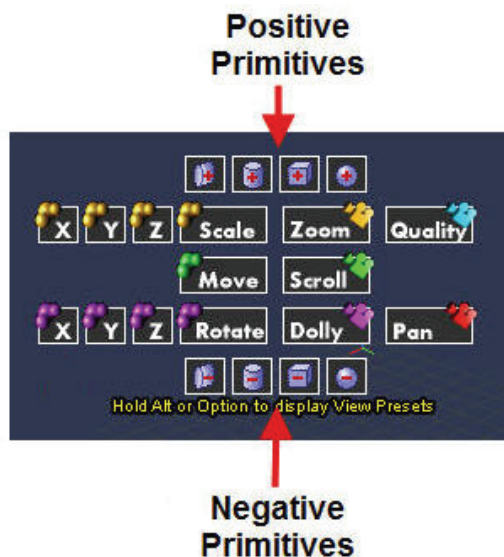
Selecting Blobs

You can select blobs with the Selection tool, or you can select all blobs of a specific color with the Select By Color tool. You can also select each blob in an object cyclically. This is useful if you want to select a blob that is behind or partly embedded in another blob.

To select all blobs of a specific color:

1. Click the Select By Color tool, color swatches appear. Roll over to the color of blobs you want to select.

2. Click a color swatch on the Select By Color tool to pick a group of objects.

For information on coloring blobs, refer to "Previewing Objects."

To cycle through the blobs in an object:

1. Deselect all objects by clicking in an empty area of the workspace.

2. Click a blob in an object and hold the mouse button down. In a few seconds, Carrara selects the next blob in the object, continuing until all the component blobs have been selected in turn. The cycle begins again if you continue to hold the mouse button.

3. Release the mouse button when the blob you want is selected.

To set the amount of time between the change of selected objects, refer to "Setting Preferences for the Metaball Modeler."

Moving Blobs

You can move blobs using standard dragging techniques.

When you move an object, an additional triplet of colored lines appears to indicate the starting point of the object. In addition, as you move, lines extend from the center of the object to show its movement along each axis.
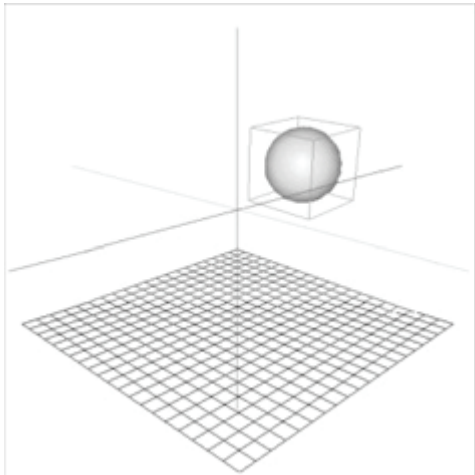
To move blobs:

1. Select one or more blobs.

2.   Choose the Selection tool or choose **Ghost > Move.**
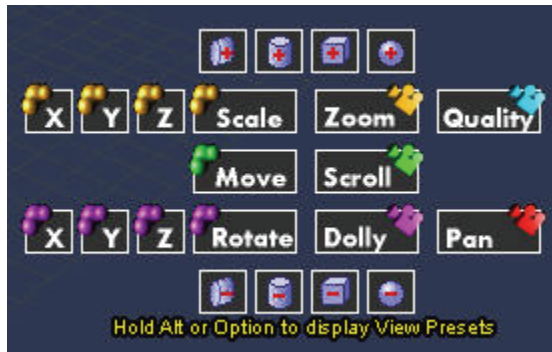
3.   Drag the blob wherever you like. Blobs move parallel to the Drawing plane unless you do one of the following:

- Press the Command/Ctrl key to drag blobs perpendicular to the Drawing plane.

- Hold down the Shift key to constrain the drag angle to increments of 45°.

You can also dolly your view while moving blobs. For more information, refer to "Changing Your Point of View."

Resizing Blobs

You can resize blobs proportionally or on a designated axis.

To scale a blob proportionally:

1.   Select a blob.

2.   Select the Scale tool or choose **Ghost > Scale.**

3.   Drag the mouse left or right to decrease or increase the blob's size.

Carrara displays three colored lines that represent the local axis of the blob element.

To scale a blob on one axis:

1.   Select a blob.

2.   Choose **Ghost > X, Y, or Z** near the Scale button.

3.   Drag the mouse left or right to decrease or increase the blob's size along the selected axis.
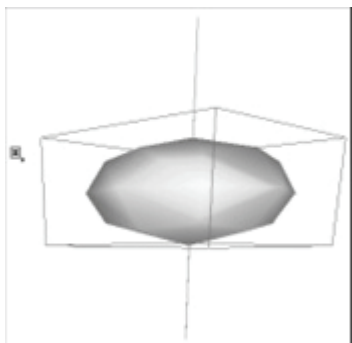
Carrara displays a colored line that represents the local rotation axis of the blob element.

Rotating Blobs

You can rotate an object in several different ways:

- Free rotate in three dimensions.
- Rotate on a designated axis.
- Rotate perpendicular to the Drawing plane.

To free rotate a blob:

1. Select a blob.
2. Select the Rotate tool or choose **Ghost > Rotate.**
3. Drag the mouse to rotate the blob.

   Hold down the Shift key to constrain the rotation angle to 45° increments.

To rotate a blob on one axis:

1. Select a blob.
2. Choose **Ghost > X, Y,** or **Z** near the Rotate button.
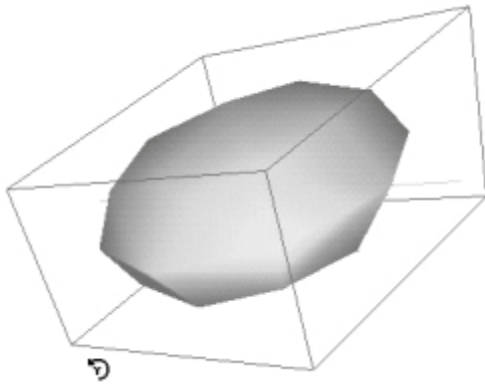
3. Drag the mouse to rotate the blob along the selected axis.

   Hold down the Shift key to constrain the rotation angle to 45° increments.

To rotate a blob perpendicular to the Drawing plane:

1. If you are not already in an isometric view, use the Alternate Ghost Menu to select one.
2. Select a blob.
3. Select the Rotate tool or choose **Ghost > Rotate.**
4. Drag the mouse to rotate the blob.

Duplicating Blobs

You can duplicate blobs, placing the duplicate objects on top of each other or mirrored on the other side of the Drawing plane.

To duplicate a blob:

1. Select a blob.

2. Choose **Edit > Duplicate.**

    The blob is duplicated, with the duplicate placed in the same location as the original.

To duplicate a blob with symmetry:

1. Select a blob.

2. Choose **Edit > Duplicate with Symmetry.**

    The blob is duplicated, with the duplicate mirroring the original object on the other side of the Drawing plane.

Blending Blobs

To blend blobs, simply drag them close together. The proximity required for blobs to blend depends on the surface threshold.

To set the surface threshold:

1. Choose **Geometry > Surface Threshold**. The Threshold dialog appears.
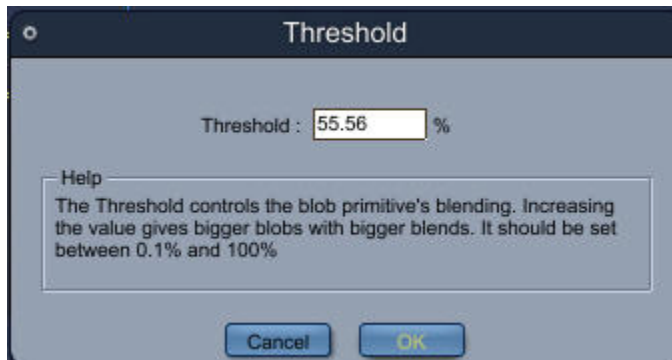
2. Enter a value to change the surface threshold.

    *   A lower value will give smaller blobs and longer blends.

    *   A higher value will give larger blobs and smaller blends.

Examples:

*   Low and high surface thresholds with positive blobs.
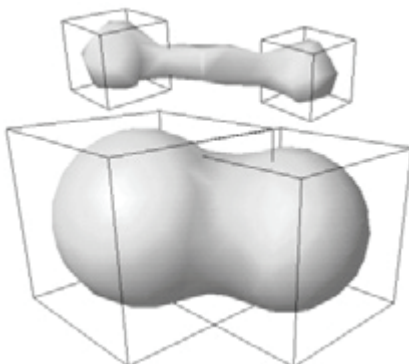
- Low and high surface thresholds with negative blobs.
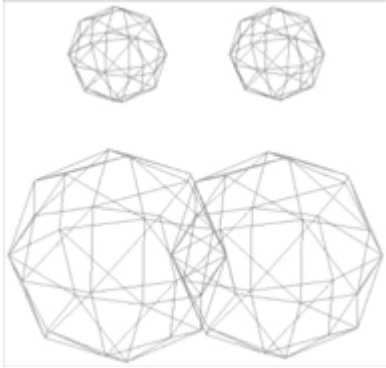
Setting Surface Fidelity

When Carrara renders, it breaks each object into hundreds of tiny polygons. This helps the renderer understand the contents of the scene.

The number of polygons used for each object is based on the size of the object, its distance from the camera, and the rendering resolution. This calculation usually results in a smooth object.

If an object does not render as smoothly as you'd like, you can increase its surface fidelity. This forces Carrara to break it down into a greater number of polygons.

If none of the objects in your scene renders as smoothly as you would like, you can increase the rendering Silhouette Quality instead. However, this option uses more memory and increases rendering time. For more information, refer to "Previewing Objects."

To set surface fidelity:

1. Choose **Geometry > Surface Fidelity**. The Surface Fidelity dialog appears.
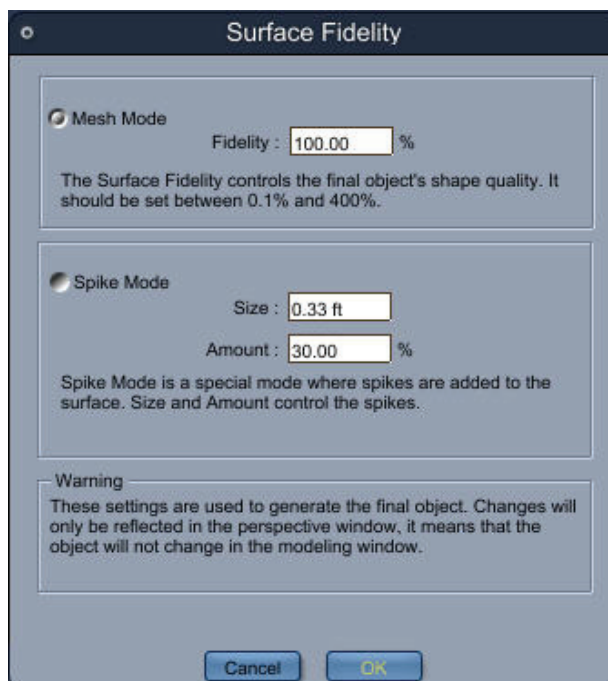
Use the Surface Fidelity dialog to control the smoothness of an object's rendering.

2. Choose the options you want.

   - **Mesh Mode** generates a smooth surface, more or less as seen in the Metaball Modeling window.\

   - **Fidelity** determines the number of polygons in the object. The higher the number, the smoother the surface and the longer the rendering time.

   - **Spike Mode** generates a spiky surface. This is useful for creating furry animals or bushes.

   - **Size** controls the length of the spikes.

   - **Amount** controls the number of spikes.

3. Click **OK**.

Putting It All Together

Arranging Objects

As you build a scene in the Assemble room, you scale and position objects in relation to the global universe and to each other. This chapter describes the techniques you use to arrange objects in your scene.

Arranging is the process of positioning, scaling, and orienting objects. The most significant part of an object's arrangement is its position as it relates to other objects. Its absolute arrangement (relative to the universe) is generally important only for orientation.
For example, it is important to position a glass of wine on a tabletop rather than having it intersect with or float above the table. At the same time, you should also use the glass's absolute arrangement to position a filled glass in its upright orientation, so that the full glass does not appear to be spilling the liquid.

When you arrange one object in relation to another, you may need to perform a series of positioning, orientation, scaling, and alignment operations. Many of these commands operate under the constraints of the working box. Setting the working box appropriately before starting an operation greatly simplifies the procedure. Refer to "Using the Working Box" for further information.

Note Except where noted, all of the positioning, orienting, aligning, and scaling commands work with either simple or grouped objects. For information on grouping objects, refer to "Building a Hierarchical Structure."

Adding and Deleting Objects

You can add existing objects to a scene (from other scenes or from clip art folders) in several ways:

   - Using the Scene Browser. For more information on using the Browser, refer to "The Browser Tray."

   - Dragging between windows.

   - Using the Copy and Paste commands.

   - Import an entire scene. For more information on importing objects from other 3D modeling applications, refer to "Importing 3D Objects."

When you add an existing object, its shading, position, orientation, and size characteristics are brought with it. You can, of course, change any of these in the new scene.

Adding Objects from Other Scenes

You can add objects from one scene into another by dragging an item from the Sequencer Tray or from the Instances tab in the Properties tray.

To add an object from another scene by dragging:

1. Choose **File > Open** to open each scene file.

2. Select the scene that contains the object you want to use.

3. Open the Sequencer Tray, or the Instances tab in the Properties tray.

4. Drag the name of the object you want from the source scene to the destination scene window.
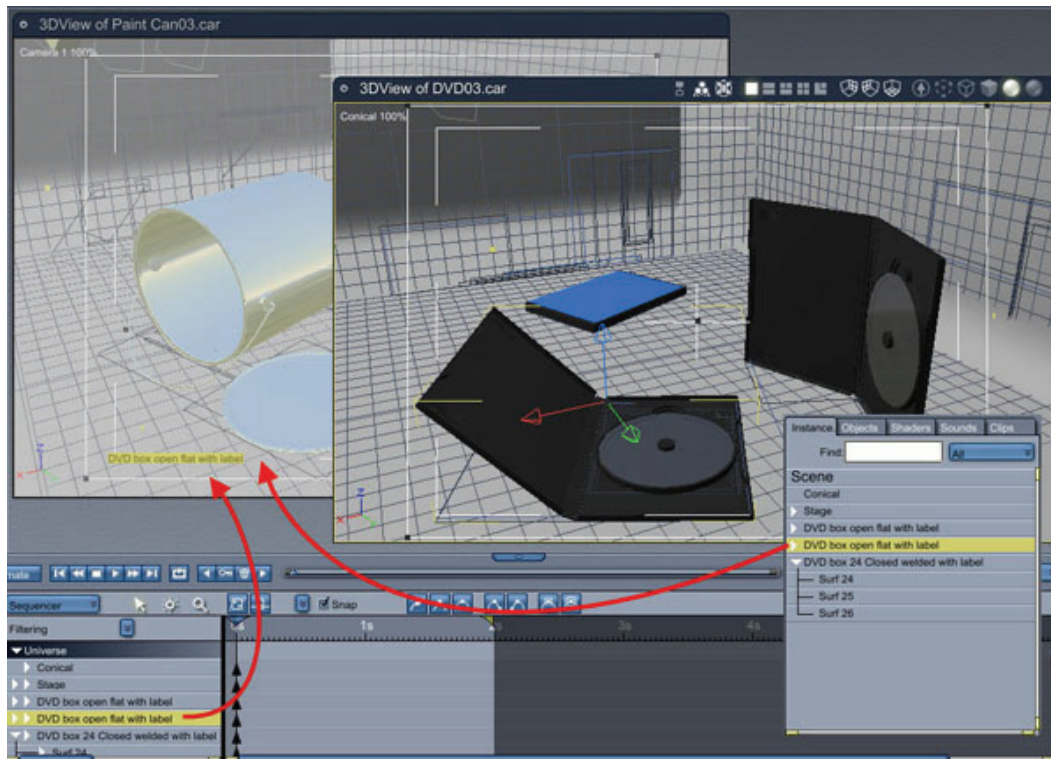
Adding Objects from the Browser

To add an object from the Scene Browser:

1. Create or open a scene.

2. Open the Browser, and select the tab that holds the type of object you want to add.

3. Drag the object's icon from the Browser into the scene window or into the Hierarchy. When you drag into the Hierarchy, you can drag directly onto an object or a group. The object you dragged becomes a child of the object it is dropped onto.
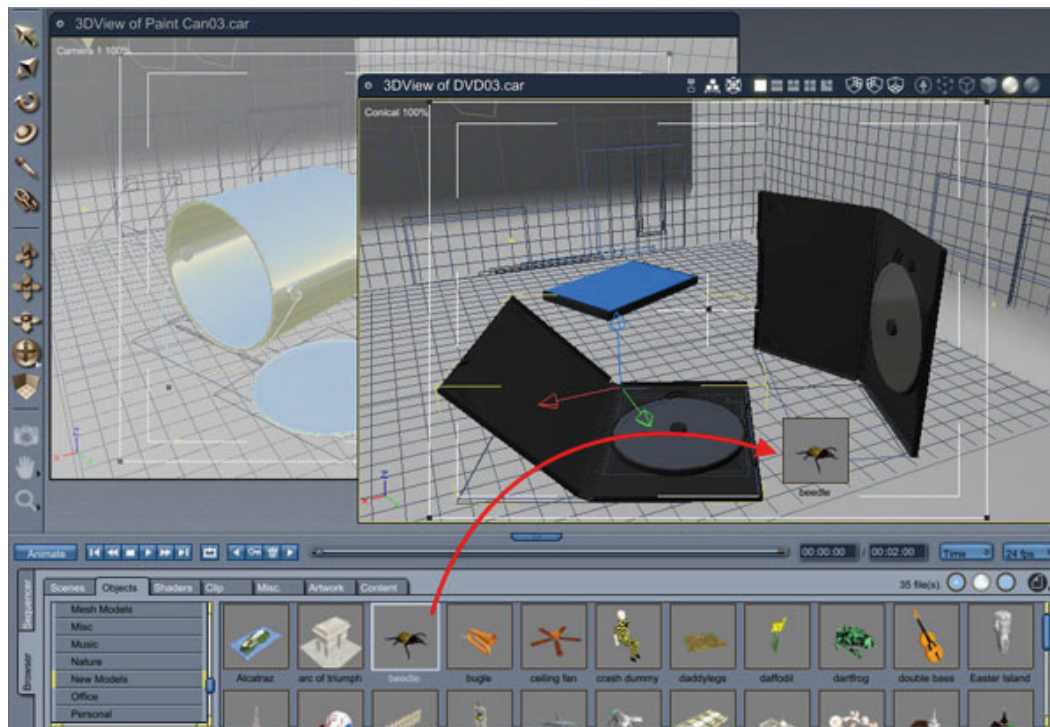
Deleting Objects

To delete an object:

1. Select the object in the scene window or Hierarchy.

2. Choose **Edit > Delete** or press Delete/Backspace

NOTE! When you delete a parent object within a group, you automatically delete all of its child objects. If you want to delete an object but not its children, drag the children to the Universe level before proceeding. For details on parent and child objects, refer to "Linking Objects."

Hiding and Showing Objects

To hide or show an object:

1. Select the object that you want to show or hide, from the scene (if visible), or from the hierarchy in the Sequencer or Properties tray (if either visible or hidden).

2. Choose **Edit > Hide/Show in 3D View.**

Selecting Objects

Selecting Objects in the Assemble Room

In order to arrange objects, you first have to select the object or objects that you want to arrange. Carrara provides several ways to select objects:

When an object is selected, you'll notice the following:

- It displays its bounding box.

- Its projections onto the working box planes are highlighted.

- The name of the object is highlighted in the Hierarchy in the Sequencer tray and in the Properties tray: Instances tab.

Selecting with the Manipulator Tools

To select an item in the document window, click the object preview or one of the projections with a manipulator tool (the Move, Scale, Rotate, or Universal Manipulator tool).

To select objects with a manipulator tool:

1. Choose the manipulator for the type of action you want to perform (Move, Scale, Rotate, or Universal Manipulator).

2. Click the object in the document window, or one of its projections on the working box.

3. To select or deselect additional objects:

- Hold down the Shift key and click other objects to add to the selection.

- Shift+click an object that's already selected to subtract it from the selection.

4. If a number of objects overlap, right-click over the objects. A pop-up menu lists all of the objects beneath the cursor at that point. Select the object you want.

Selecting Objects in the Hierarchy

To select one or more items in the Sequencer tray, or in the Properties tray: Instances tab, click an item or drag a marquee over one or several items.

To select objects in the Hierarchy:

- To select a single item, click the object's name in the Hierarchy.

- To select multiple objects:

    o   Drag a marquee around several contiguous objects.

    o   Hold down the Shift key and click other objects (or drag another marquee) to add to the selection.

    o   Ctrl/Cmd+Click to select additional items that are not contiguous.

    o   Shift+drag or Ctrl/Cmd+Click over items that are already selected to subtract them from the selection.

For complete information on working with the Hierarchy, refer to "Working with Objects".

Selecting Objects in the Assemble Room

In order to arrange objects, you first have to select the object or objects that you want to arrange. Carrara provides several ways to select objects:

When an object is selected, you'll notice the following:

- It displays its bounding box.

- Its projections onto the working box planes are highlighted.

- The name of the object is highlighted in the Hierarchy in the Sequencer tray and in the Properties tray: Instances tab.

Selecting with the Manipulator Tools

To select an item in the document window, click the object preview or one of the projections with a manipulator tool (the Move, Scale, Rotate, or Universal Manipulator tool).

To select objects with a manipulator tool:

1. Choose the manipulator for the type of action you want to perform (Move, Scale, Rotate, or Universal Manipulator).

2. Click the object in the document window, or one of its projections on the working box.

3. To select or deselect additional objects:

- Hold down the Shift key and click other objects to add to the selection.

- Shift+click an object that's already selected to subtract it from the selection.

4. If a number of objects overlap, right-click over the objects. A pop-up menu lists all of the objects beneath the cursor at that point. Select the object you want.

Selecting Objects in the Hierarchy

To select one or more items in the Sequencer tray, or in the Properties tray: Instances tab, click an item or drag a marquee over one or several items.

To select objects in the Hierarchy:

- To select a single item, click the object's name in the Hierarchy.
- To select multiple objects:
    - o Drag a marquee around several contiguous objects.
    - o Hold down the Shift key and click other objects (or drag another marquee) to add to the selection.
    - o Ctrl/Cmd+Click to select additional items that are not contiguous.
    - o Shift+drag or Ctrl/Cmd+Click over items that are already selected to subtract them from the selection.

For complete information on working with the Hierarchy, refer to "Working with Objects".

Selecting Objects in the Assemble Room

In order to arrange objects, you first have to select the object or objects that you want to arrange. Carrara provides several ways to select objects:

When an object is selected, you'll notice the following:

- It displays its bounding box.

- Its projections onto the working box planes are highlighted.

- The name of the object is highlighted in the Hierarchy in the Sequencer tray and in the Properties tray: Instances tab.

Selecting with the Manipulator Tools

To select an item in the document window, click the object preview or one of the projections with a manipulator tool (the Move, Scale, Rotate, or Universal Manipulator tool).

To select objects with a manipulator tool:

1. Choose the manipulator for the type of action you want to perform (Move, Scale, Rotate, or Universal Manipulator).

2. Click the object in the document window, or one of its projections on the working box.

3. To select or deselect additional objects:

- Hold down the Shift key and click other objects to add to the selection.

- Shift+click an object that's already selected to subtract it from the selection.

4. If a number of objects overlap, right-click over the objects. A pop-up menu lists all of the objects beneath the cursor at that point. Select the object you want.

Selecting Objects in the Hierarchy

To select one or more items in the Sequencer tray, or in the Properties tray: Instances tab, click an item or drag a marquee over one or several items.

To select objects in the Hierarchy:

- To select a single item, click the object's name in the Hierarchy.

- To select multiple objects:

    o Drag a marquee around several contiguous objects.

    o Hold down the Shift key and click other objects (or drag another marquee) to add to the selection.

    o Ctrl/Cmd+Click to select additional items that are not contiguous.

    o Shift+drag or Ctrl/Cmd+Click over items that are already selected to subtract them from the selection.

For complete information on working with the Hierarchy, refer to "Working with Objects".

Moving the Hot Point

The hot point is the single point of an object or group that identifies its center of rotation.

The hot point of selected objects and groups appears in the Assemble room as a black point that casts 2D projections.
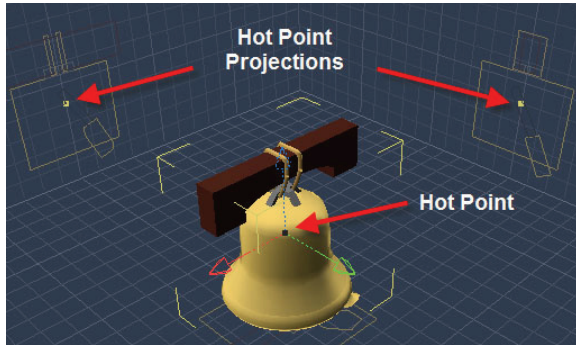
By default, an object's or group's hot point is at the center of its bounding box, but you can move the hot point anywhere in the scene.

The type of operation you want to perform determines where you put the hot point. For example, when you want to show a bell ringing, you need to rotate the bell from near its top. When the hot point is at the center of the bell, the Rotate tool rotates the object in place. However, when you place the hot point near the top, the bell rotates more realistically.
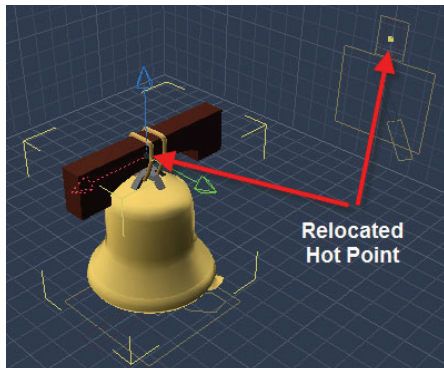
Dragging the Hot Point

You can use the Move tool to reposition the hot point of an object or group.

To move a hot point by dragging:

1. Choose the Move tool.

2. Select the object or group whose hot point you want to move.

3. If you drag the hot point when Caps Lock is off, the object moves with it. Turn on Caps Lock on your keyboard to unlock the object and its hot point.

4. Drag the hot point in 3D or drag one of its 2D projections. The hot point moves parallel to the active plane.

5. Hold down Command/Ctrl to drag the hot point perpendicular to the active plane.

6. Turn off Caps Lock when you are finished.

Moving the Hot Point Numerically

To move a hot point numerically:

1. Select the object or group.

2. Open the **Properties tray: Motion tab: Transform panel**.

3. Enter the desired X, Y and Z coordinates in the Hot Pt. fields. For more information on these settings, refer to "Viewing and Setting Object Properties."

Centering the Hot Point

If you move the hot point for an object or group, you can easily return it back to the center of the object or group (its default position).

To send the hot point to the center of the object:

1. Select the object or group.

2. Choose **Edit > Center Hot Point**. Carrara moves the hot point to the center of the object or group bounding box.

Viewing and Setting Object Properties

When you select an object, you can view its properties in the Properties tray. The Properties tray allows you to rename objects and move, scale, or rotate them based on numerical values.

Naming Objects

By default, new objects are named **Type n**, where **Type** is the object description (Sphere, Text, etc.) and **n** is the number of similar objects in the order created:Sphere 1, Sphere 2, and so forth.

You can change the names of objects and groups. Giving objects and groups descriptive names can make them easier to locate and select.

To change the name of an object or group:

1. Select the object or group.

2. On the **Properties tray: General tab: General panel**, enter any name you like in the Name field.
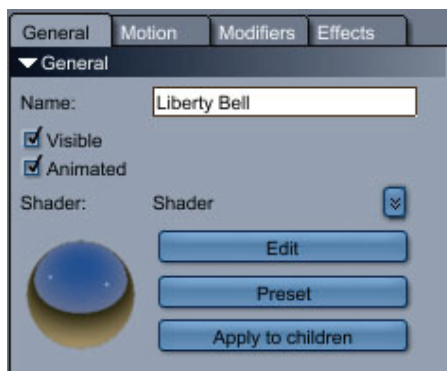
Arranging and Scaling Objects Numerically

You can use the options in the **Properties tray: Motion tab: Transform panel** to perform most of the basic arranging operations discussed in this chapter. You can set properties for an object or for a group. The Properties tray lets you make numerical adjustments:as opposed to dragging or nudging:for moving, positioning, scaling, and rotating an object.
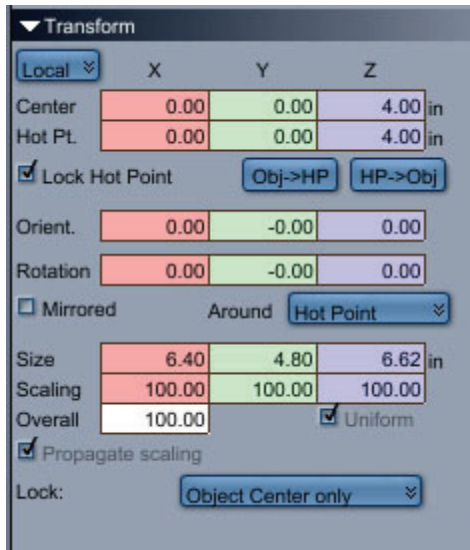
By default, numerical changes use the units of measurement currently specified in the general scene preferences (**File > Preferences: General Scene Settings**) For more information on setting these preferences, refer to "General Scene Settings Preferences."

To arrange objects numerically:

1. Select the object or group that you want to modify.

2. Display the **Properties tray: Motion tab: Transform panel**.

3. Select the Coordinate >System you want to use from the popup menu:

    -   **World** lets you specify the object's arrangement relative to the universe. Use this for absolute positioning, scaling, and rotating operations.

    -   **Local** lets you specify the object's arrangement relative to the center of the object or group bounding box. For instance, you can move the hot point out from the center on a specific axis.

4. From the Lock popup menu, choose **Hot Point and Center** to make the object move when you change either the Center or Hot Point value.
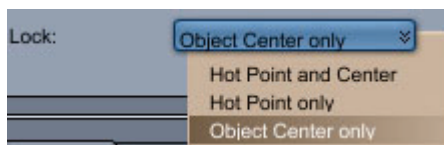
5. Enter new X, Y, and Z values for the object property you want to change:

    -   **Center** coordinates reposition the object from its center.

    -   **Hot Pt** coordinates reposition the hot point numerically as described in "Moving the Hot Point Numerically."

    -   **Orient** coordinates change the rotation of the object in relation to its hot point.

- **Rotation** coordinates rotate the object along one or more axes. For details on orientation, refer to "Rotating Objects."

- **Mirrored**: Check this option to mirror (or flip) the object around its hot point or around the object's center as described in "Mirroring an Object's Orientation."

- **Size**: Sets the dimensions (height, width, and depth) of the object's bounding box.

- **Scaling:** Sets the scaling percentages on each axis. One hundred percent is the scale at which the object was created. Check the **Uniform** option to disable individual scaling axes, and use the **Overall** scale feature instead.

- **Overall**: Scales the X, Y, and Z axes uniformly by the value that you enter, with 100 equaling the original size of the object.

Mirroring an Object's Orientation

You can automatically mirror an object or group's orientation across the ground plane.

To mirror an object or group's orientation:

1. Select an object or group.

2. Position the hot point to describe the location you want for the mirror plane.

3. In the **Properties tray: Motion tab: Transform panel**, enable the Mirror option.

4. From the Around popup menu, select whether you want to mirror the object or group around its hot point, or around its center.
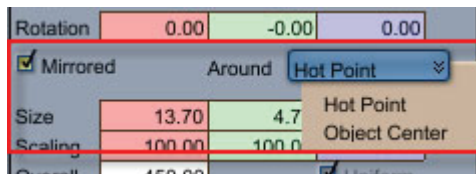
Positioning Objects

In addition to moving, scaling, and rotating objects numerically, you can also use the manipulator tools and other keyboard controls to accomplish similar tasks.

Dragging Objects

The easiest way to move an object is to drag it with the **Move** or **Universal Manipulator** tool.

Dragging operations use the working box planes as the dimensional reference:

- Remember that you can orient the working box to a particular attitude before dragging an object. For information on moving and orienting the working box, refer to "Changing the Working Box."

- Remember also that the planes of the working box extend throughout the universe. The visible grid is merely a reference of the orientation of those planes and does not restrict you to the visible space.

To drag an object:

1.   Choose the Move tool or the Universal Manipulator tool.

2.   Use one of the following procedures:

- Click and drag the object itself to move it parallel to the active plane.

- Ctrl/Cmd-click the object to move it perpendicular to the active plane.

- Drag the object's projection along the desired working box plane.

- Drag the manipulator by the X (red), Y (green) or Z (blue) arrow indicator to move along the desired plane.

- Hold down the Shift key to constrain the drag angle to increments of the rotation angle.

Moving Objects with the Manipulators

When an object is selected with the Move tool or the Universal Manipulator, three colored arrows let you move the object along the global X, Y, and Z directions. As you move the mouse cursor near one of them, the cursor changes to a cross. You can then click and move the cursor in the direction of the arrow. This constrains the movement of the object to the direction indicated by the selected arrow.

To move an object with the manipulator:

1.   Choose the Move tool, or the Universal Manipulator tool.

2.   Use one of the following procedures:

- Drag the manipulator by the X (red), Y (green) or Z (blue) arrow to move along the desired axis.
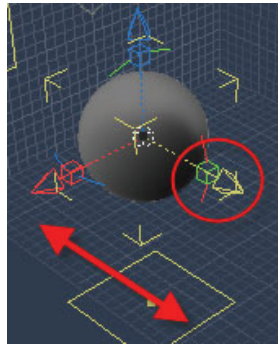
- Ctrl/Cmd+drag the arrow to move perpendicular to the desired axis.
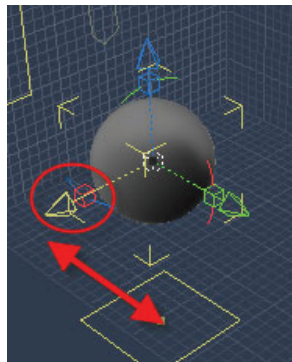
- Hold down the Shift key to constrain the drag angle to increments of the rotation angle.

Centering Objects in the Universe

You can send an object or group to the center of the universe. The object maintains its orientation, and its center (not its hot point) is placed at 0,0,0.

To center an object in the universe:

1. Select the object.

2. Choose **Edit > Send To Origin**.

Nudging with the Keyboard

You can use the keyboard to nudge objects or groups in small steps.

To use the keyboard to nudge an object:

* Press and hold the Command/Ctrl key, and then press the arrow keys to move the object in the active plane.

* Press the Command/Ctrl and Option/Alt keys at the same time to move the object perpendicular to the active plane.

* Press the Command/Ctrl and Shift keys at the same time to move in smaller steps.

Scaling Objects

When you create an object in a modeler, you model it at a particular size. In the scene in the Assemble room, you can scale the object to new dimensions.

Because Carrara allows you to work with real- world units (inches, feet, centimeters, meters, etc.), many artists scale objects equivalent to their size in the real world. For example, you could create life-size models of several boxes:one at 12 x 12 x 15 inches, one at 13 x 18 x 12 inches, and one at 14 x 20 x 14 inches:and a life-size model of a ladder at 48 inches. When you put all the objects together in the same scene, the objects have the correct size relationship.
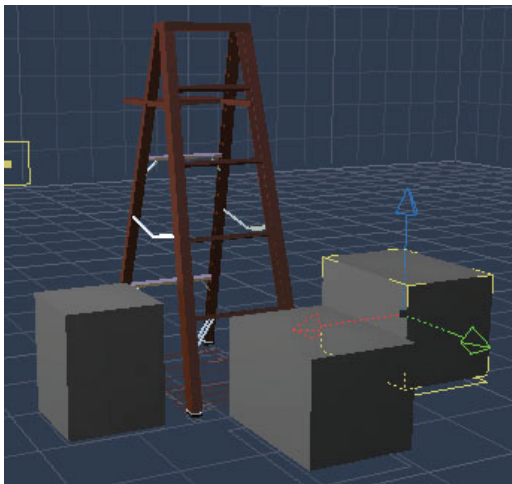
Carrara allows you to work with real world units to maintain true size relationships between objects.

In all cases, you can scale an object proportionately or disproportionately. Scaling in the scene does not change the size of the original object; if you reopen it in a modeler, it will be the same size as when you created it.

You can scale an object or group several ways:

- Use the Scale tool to drag its bounding box. This is useful if you want to scale one object "by sight" and then apply the same scaling factor to other objects.

- Use the Size and Scaling controls (numerical scaling) in the **Properties tray: Motion tab: Transform panel**. For details, refer to "Viewing and Setting Object Properties."

Scaling an Object by Dragging

To scale an object by dragging:

1. Select an object.

2. Choose the Scale or Universal Manipulator tool.

3. Depending on how you want to scale, drag one of the following on the object's bounding box or projection:

   - **Corner** This maintains the object's proportions; it is scaled along all three axes.
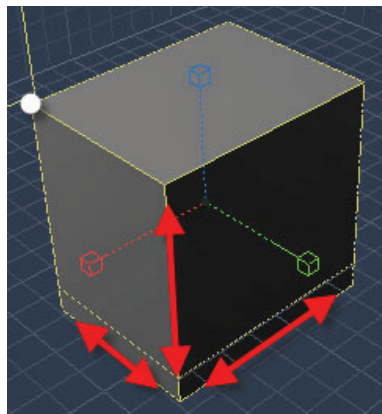
   - **Edge** This scales the object on two of its axes.
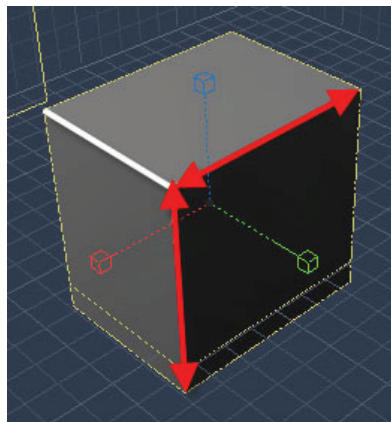
- **Side** This scales the object on one of its axes.
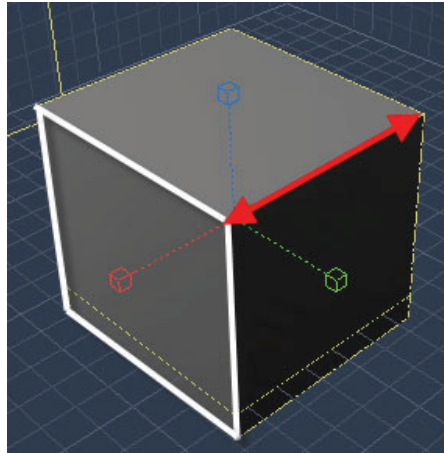
The reference:which remains anchored in place:is the corner opposite to the one you drag. To use the hot point as the reference, open the **Properties tray: Motion tab: Transform panel**. Click the **Lock** popup menu near the bottom of the Transform section, and choose **Hot Point Only**. The hot point stays in place, and all eight corners of the bounding box move to scale the object.
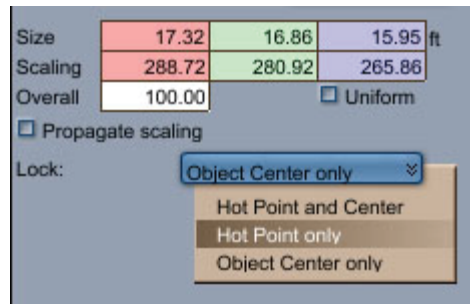
4. When you are satisfied with the object's new size, release the mouse. The object's Scaling factors change as you scale the object. You can use this information to scale other objects by the same amount.

NOTE! Scaling may change the relative positions of two objects. For instance, a glass on a table, if enlarged, may go through the table, and if shrunk, may seem to float over it. To prevent this, use the plane at the bottom of the glass as the scaling anchor—that is, drag a corner at the top of the bounding box, not at its bottom.

Scaling an Object with the Manipulators

When an object is selected with the Scale tool, three colored cubes let you scale the object along its local X (red), Y (green), and Z (blue) directions. As you move the mouse cursor near one of them, the cursor changes to a cross. You can then click and move the cursor in the direction of the line under the cube. This constrains the scaling of the object to that line. If you press the Shift key at the same time, the scaling is uniform (the same in each direction).
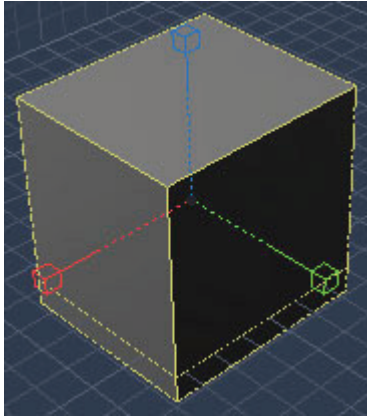
You can still use the bounding box of the object to scale it.

To use the keyboard to nudge the scale of an object:

- With the Scale tool selected:
    - Press and hold the Command/Ctrl key, and use the arrow keys to nudge the object.
    - If you press the Shift key at the same time, the object scales with smaller steps.

Rotating Objects

Most real-world objects have a logical upright position and some have a logical front. For example, airplanes and automobiles have both. In simple terms, think of an object's orientation as the direction it faces.

Carrara interprets the native upright position and front of objects by the way they are modeled.

Note Orientation applies to objects and groups, as well as to lights and cameras.

Carrara allows you to change an object's orientation in several ways: with the Rotate tool (free rotation), and numerically. You can also choose whether to rotate an object around its center or its hot point.

To choose an object's center of rotation:

1. In the **Properties tray: Motion tab: Transform panel: Around** list, select Object Center or Hot Point.
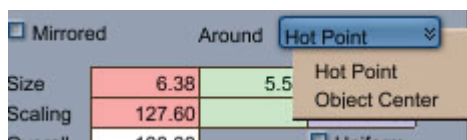
2. If desired, position the object's hot point. For information on positioning an object's or group's hot point, refer to "Moving the Hot Point."

3. Select the object or group.

4. Choose the Rotate tool.

5. Drag the object in a circle.

To constrain rotation to a particular plane:

1. Orient the working box and position the object's hot point. This allows you to control the plane and axis of rotation.

> **NOTE!** For information on aligning the working box to a group or object, refer to "Changing the Working Box". For information on positioning an object's or group's hot point, refer to "Moving the Hot Point."

2. Choose the Rotate tool.

3. Drag one of the object's projections in a circle to rotate the object parallel to that plane. Hold down the Shift key while you drag to constrain rotation to increments of the Rotation Angle.

To orient an object or group numerically:

1. Select the object or group.

2. Enter the X,Y,Z rotation values you want on the **Properties tray: Motion tab: Transform panel**. For more information on these settings, refer to "Viewing and Setting Object Properties."

Using the Manipulators to Rotate

You can use the manipulators to rotate objects.
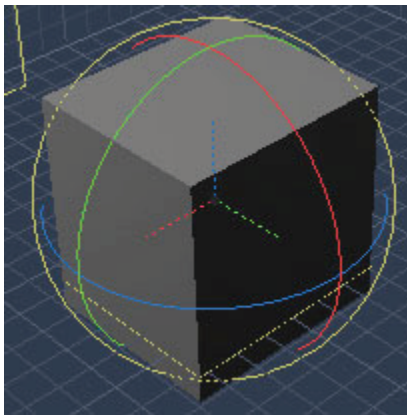
To rotate an object using the manipulators:

- Select an object with the Rotate tool.

- Three colored arcs let you rotate the object around its local X (red), Y (green) and Z (blue) directions.

    o A ring encompasses the three arcs and makes the object rotate around the direction of the camera.

    o As you move the mouse cursor near one of the arcs or the ring, the cursor changes to a cross.

    o You can then click and move the cursor along the arc. This constrains the rotation of the object.

    o To rotate freely in all directions, click inside the ring (without clicking on one of the arcs) and move the mouse.

To use the keyboard to nudge the rotation of an object:

- Press and hold the Command/Ctrl key, and use the arrow keys to nudge the object.

- If you press the Shift key at the same time, the object rotates with smaller steps.

Changing the size of the manipulators:

To easily manipulate big or small objects, the size of the manipulators depends on the size of the 3D View.

To make the manipulators bigger or smaller:

1. Choose **File > Preferences**.

2. Choose 3D View in the pop-up menu.

3. Adjust the **3D View Controls Size** slider.

Arranging Objects with Constraints

By default, when an object has a position constraint, dragging it respects the constraint. For example if the constraint is a ball joint, the object can only be rotated and cannot be moved. This Use Constraints mode can be toggled on and off by using the **View > Use Constraints** command, or by clicking on the Use Constraints icon on the top of the 3DView window.

When the Use constraints mode is off, the objects can be moved and rotated freely, even if they have a constraint. See "Applying Constraint Properties."

Arranging Objects Without Modifying its Children

When you move, scale or rotate an object, its children follow the movement. But if you want to adjust an object in a hierarchy without affecting the children: you have to move the parent and then move back the children to their previous position.

The **Lock Children** mode simplifies that: enable it and you will be able to move, rotate or scale an object without changing its children. This can be useful in a lot of cases, for instance, to adjust the joints of a skeleton.

 NOTE! Be careful when using this mode on animated objects. This mode is only for adjusting, you should avoid using it in the middle of an animation as it may modify the movement of the children of the object you are modifying.

Aiming Cameras and Lights

All of the positioning and orientation commands work on cameras and lights as well as simple objects. The Point At command directs the light beam or camera view toward an object you specify. As the object moves, the light or camera continues to point to it.

NOTE! Because the current camera is not visible as an object, manipulation by dragging isn't possible.

To point a camera or light at an object:

1. Select the camera or light, and the object you want to point at. You may select multiple cameras and lights, but only one object or group. If you don't select a camera or light, Carrara points the current camera.

2. Choose **Edit > Point At**. The light or camera points at the hot point of the selected object or group.
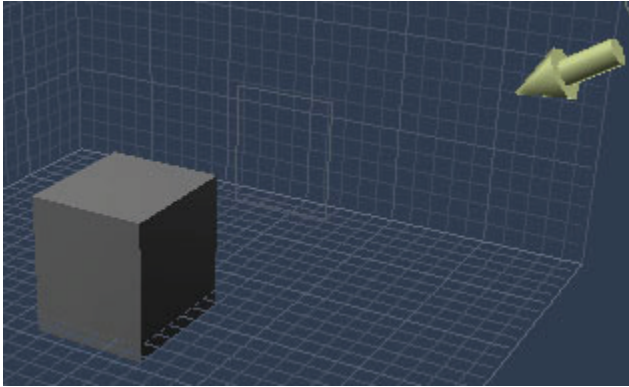
Aligning Objects

Carrara's relative alignment feature lets you arrange objects with respect to each other. You align objects on each axis separately. The selected axis is called the **axis of constraint**.

NOTE! When aligning many objects, you may want to start by aligning only two of them. Once the two objects are aligned, group them, then align a third object to the group. Continue to group and align objects until all are aligned.

To align objects relative to each other:

1. Orient the working box to set the axes of constraint you want to use. For information on orienting the working box, refer to "Changing the Working Box."

2. Select two or more objects. The first object you select becomes the anchor object.

3. Choose **Edit > Align**. The Align dialog appears.
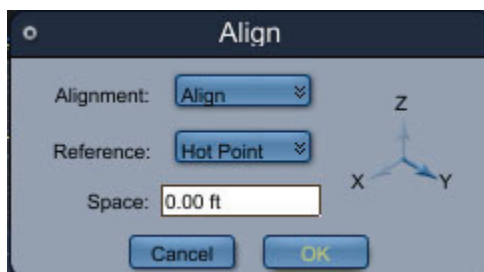
4. Click the X, Y, and/or Z axis indicators in the right side of the dialog to choose one or more axes of constraint. Click again to deselect an axis. If you choose all three axes, the second object will be embedded in the anchor object.

5. For each axis, select an option from the Alignment popup menu. You'll often use a different command on each axis.

- **Align** sets the reference point of each object colinear with the reference point of the anchor object along the axis of constraint.

- **Contact** brings the Max of each object into contact with the Min of the next object along the axis of constraint.

- **Space** puts the specified distance between the reference points of each object along the axis of constraint. Enter a spacing value.

- **Distribute** aligns three or more objects at an equal distance from each other along the axis of constraint. By default, the first two objects you select serve as anchors, while the third and subsequent objects are the ones that move.

6. Choose an option from the Reference menu. Object reference points that are grayed out are unavailable for the selected alignment command.

- **Hot Point** aligns the objects by their hot points (two of which have been moved from center in the following figure).
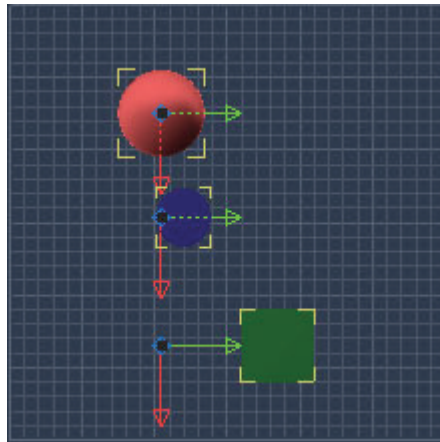
- **Min** specifies the side of each object's bounding box with the lower coordinate value along the axis of constraint.
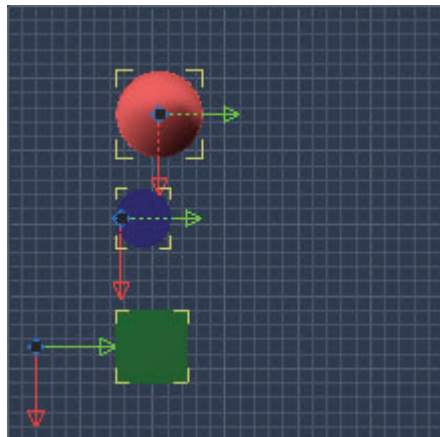
- **Center** specifies the center of each object's bounding box.
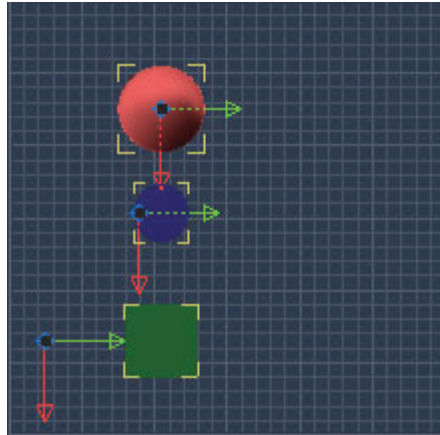
- **Max** specifies the side of each object's bounding box with the higher coordinate value along the axis of constraint.
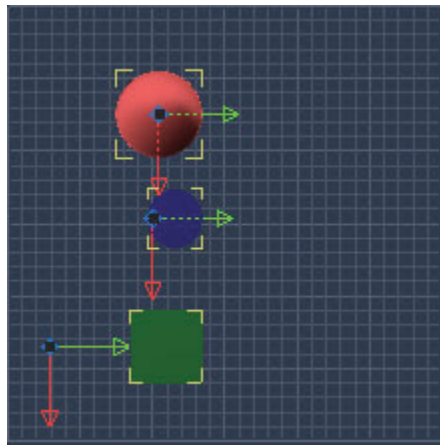
- **Sides** specifies the sides of the object's bounding box.
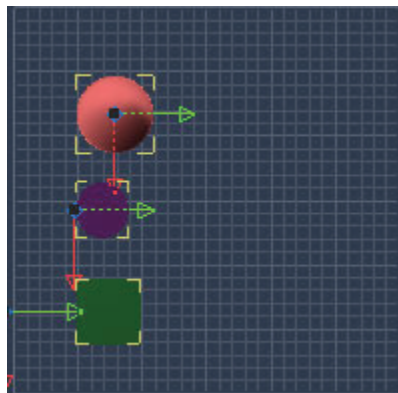
7. If you want to add a buffer around the distribution of the objects, enter a value for the buffer in the Space field.

8. Click OK.

9. When you have achieved the alignment you want in one constraint axis, move on to the next one. (Return to step 3.)

Working with Objects

This chapter explains how to use the hierarchical structure of a Carrara scene to add objects and determine their behavior.

You start building a scene by adding, deleting, and replacing objects.

Jumping into Groups and Editing Objects

You can select an object or group and "jump into" it. Editing an object opens the object in its native modeler, where you can further modify the object.
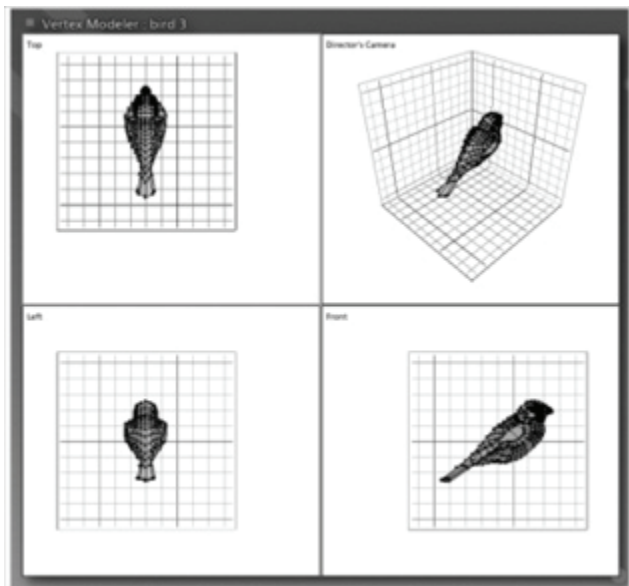
Jumping into a group allows you to rearrange the objects within the group. After you jump out, the objects remain grouped.

To edit an object:

- Do one of the following:
    - Double-click the object in the scene window or the Hierarchy.
    - Select the object in the scene window or the Hierarchy, then click **the Model button to move to the modeler**.

To jump into a group:

- Double-click the group in the scene window.

Carrara opens the group in the local universe. The coordinate system of the local universe is parallel with the walls of the group's bounding box, with its origin at the center.

NOTE! Usually, the Universe level is the root of the Hierarchy section of the Sequencer. However, if you jump into a group, the Hierarchy displays the group as the current root and the contents of that group as the only elements of the Hierarchy section of the Sequencer.

To jump out of a group:

- Click the Jump Out button in the local universe.

Editing an Object in a Different Modeler

Normally, editing opens the object in its native modeler, or the Minimum modeler for primitives and imported objects. You may prefer to open the object in a different modeler.

Working with an object in a different modeler changes its type. This may limit your options for future editing. For example, you can't edit a Spline object in the Vertex modeler, then take it back into the Spline modeler. Also, some modification options may not be available under certain modelers.

Note Carrara constructs 3D surfaces using facets and patches. Facets are triangular. Patches are polygons derived from Bézier curves. Some objects may be entirely facets, others may be a mix of facets and patches. When you open a 3D primitive, Metaball, Text, or Spline object in the Vertex modeler, all patches are converted to facets.

To edit an object in a different modeler:

1. Select the object.

2. Double-click the object to open it in its modeler. **Choose Edit >Convert To Other Modeler.** Carrara opens a dialog that shows your modeler options.

3. Select the modeler you want.

4. If you select the Vertex modeler, drag the slider or enter a value to set a fidelity level for creating facets from patches. More facets creates smoother surfaces, but increases the file size and memory required for the model.

5. When you have set your options, click OK. Carrara opens the object in the selected modeler.

Building a Hierarchical Structure

Working primarily in the Hierarchy, you can structure a scene by grouping and constraining objects. This simplifies arrangement operations and can save you time and trouble.

TIP! You'll find it easier to keep track of objects that you've individually named than it is to manage generically named objects. For information on naming objects, refer to "Naming Objects."

Every object in your scene appears in the Hierarchy section of the Sequencer and in the Instances tab of the Properties. The Hierarchy is shown as a tree of objects, each represented by a listing. The hierarchical structure of a scene changes as you introduce and group objects and create constraints.

In addition to primitives and modeled objects, hierarchy elements include groups, cameras, and lights. The highest level of the scene's hierarchical structure is its Universe list, which encompasses your entire scene. Beneath this level, there may be any number of branches and sub-branches
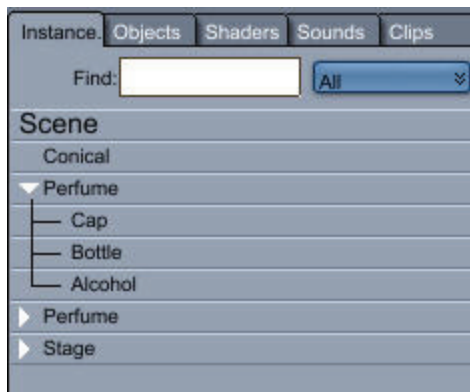
Navigating the Hierarchy

You can expand and collapse the items in the Hierarchy to view more or fewer elements in your scene. You may need to open a series of groups to find a particular object deep in the hierarchical structure of a scene.

To return to the topmost view, close the group at each level until only the Universe level is displayed.

An expand or collapse arrow appears to the left of every group. When the group is closed, the arrow faces to the right. When the group is open, it faces down.

The expand icon is a small arrow pointing to the right. The collapse icon points down.

To open or close a group:

- Click the arrow beside the group.

Once a group is opened or closed, the arrow changes direction.

Changing the Structure

You can combine multiple elements in the Hierarchy in two ways: grouping and linking. You can nest groups within other groups or create a chain with multiple links.

You can reorganize your groups and links by dragging them from one location in the Hierarchy to another. Don't worry about making a mistake. You can drag an element to another level at any time.

You can drag objects individually or select several and drag them at once.

To select multiple items in the Hierarchy:

- Hold down the Shift key and click the items you want to select, or drag a selection marquee around the items you want to select.

To remove a single object from a multiple selection:

- Hold down the Shift key and click the item.

Selecting elements in the Hierarchy applies not only to the selected object, but, in the case of a group, to its contents, and in the case of a parent object, to its children.

Groups and Links

Groups and links are similar:both let you combine multiple objects in your scene so that you can manipulate them as a single unit:but their usage is quite different.

Grouping creates collections of objects that make structuring and navigating your scene more manageable. Groups are static in that they don't change the relationship between the objects in the groups; they only contain them. This is similar to the group function in 2D illustration and drawing programs.

Links let you define a parent-child relationship between objects. By applying various types of links you can tie such objects together so that moving the parent affects the child, but moving the child does not affect the parent.

The exception to this is the Inverse Kinematics behavior applied to a child object in a linked chain. For more information on Inverse Kinematics, refer to "Inverse Kinematics."

Grouping Objects

As you build your scene, you'll want to group related objects. Grouping allows you to control a set of objects as a single unit.

You can arrange a group just as you do a single object. During any positioning or orientation operation, the objects in a group retain their spatial relationship to one another.

How you organize your groups is up to you. You'll often create groups based on the logical context of a set of objects:for example, "bench side, boards, and bolts." When arranged properly and grouped, the set of objects becomes a single object you could name "bench," which you could duplicate, move, or otherwise manipulate as a single object.
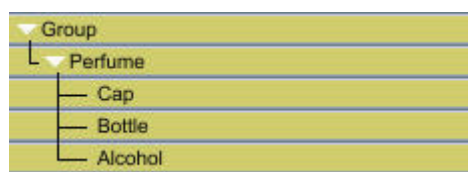
You can also create groups based on the proximity of a set of objects:for example, "all objects on a city block."
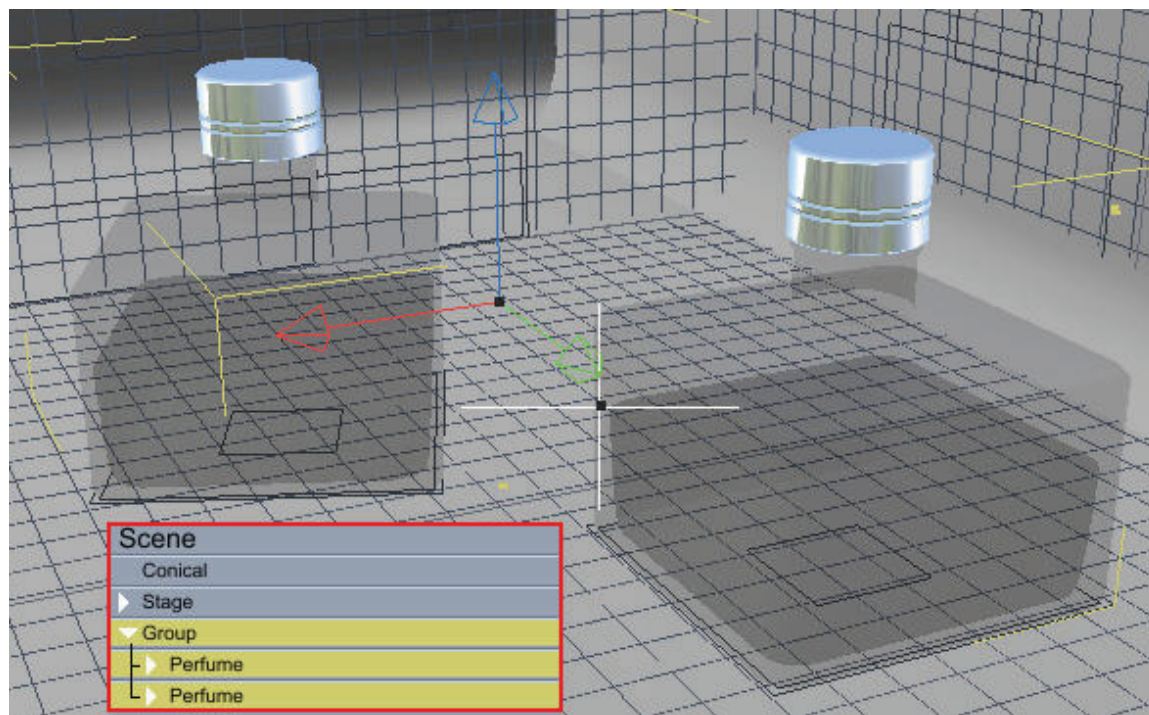
You can nest groups within groups to as many levels as you like. As you build a complex scene, you'll find nesting groups helps manage the multitude of elements.

Don't worry about perfecting the shape of an object before putting it in a group. You can always open the group, then open the object for further modification.

To create a new group:

1. Select the objects you want to group. You can select the objects in either the scene window or the Hierarchy.

2. Choose **Edit > Group**.

A group listing appears in the Hierarchy under the current root object.
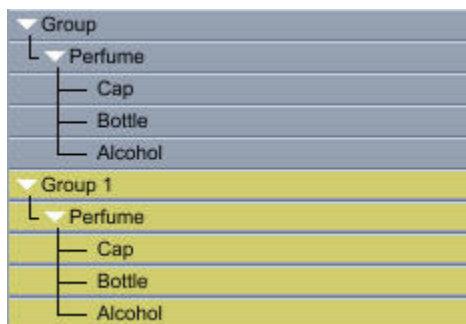
By default, groups are named Group x, where x is a sequential number (1, 2, 3...).

NOTE! You can rename a group by selecting it and changing its name in the Properties tray. For more information, refer to "Using the Sequencer Tray."

To add objects to a group:

- Drag the listings of the objects you wish to add onto the group listing.
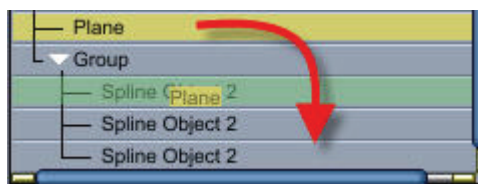
TIP! You can bring an object into the scene and immediately place it in a group. Drag the object from its source (Browser tray or other scene's Hierarchy) and drop it directly onto a group listing.

To remove objects from a group:

- Drag the selected objects to another level in the Hierarchy.

Opening and Closing Groups

You can open and close groups while you work. When the group is closed, you can manipulate the entire group as a single object. When a group is open, you can select and move each component object independently.

This is a great time saver and gives you maximum flexibility. You can easily adjust the objects in your groups, but maintain their places in the scene's hierarchical structure.

Note Opening a group is different from Ungrouping. Ungrouping dissolves the group so that all the components appear at the same level of the Hierarchy.

To open or close a group:

- Click the arrow beside the group listing in the Hierarchy.

A group listing is the root of the group's contents. While you open a group, its contents are displayed on a sub-branch. When you close a group, its content listings collapse into the group listing.

Group Bounding Boxes

In the scene window, the elements of a group are enclosed in a single bounding box, which is visible when the group is closed. When you create a group, the group bounding box is set parallel to the planes of the working box. The group maintains this orientation until you change it.
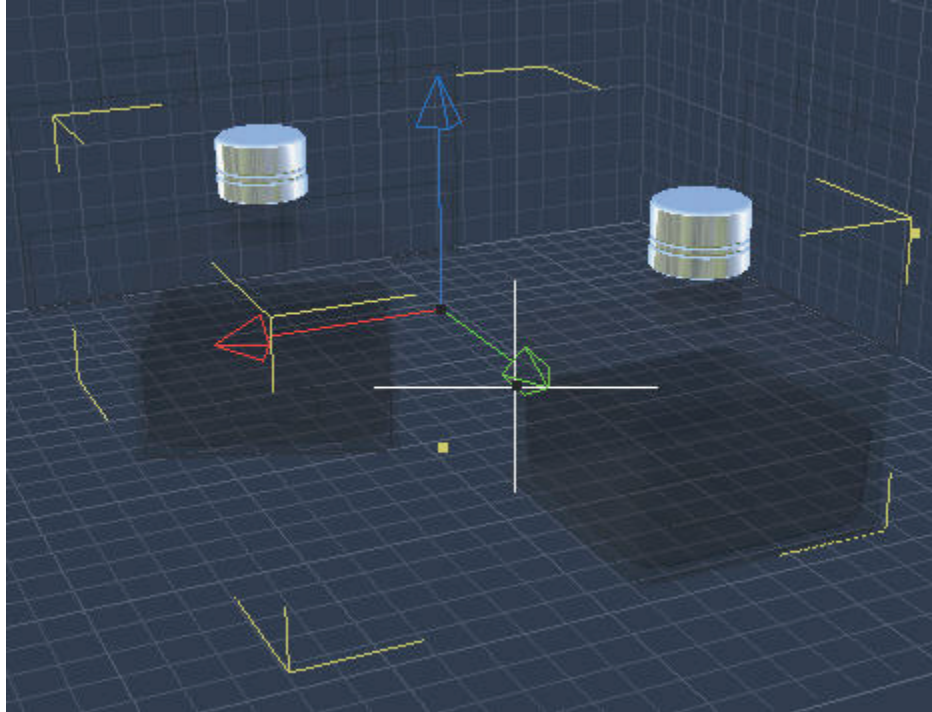
The group bounding box is the smallest box with the same orientation as the working box that encloses all bounding boxes of the group's contents. If you add objects to the group or remove them from the group, the dimensions of the group bounding box may change. The orientation of the group bounding box, however, does not change as you add or remove elements

Jumping In and Out

You can jump into a group to modify the positions of its objects. When you jump into a group, the Hierarchy displays the group listing as the current root and its contents as the only elements of the Hierarchy. Simultaneously, the scene window redraws to display only the contents of this group.

Within a group, a coordinate system called the Local Universe is used. This system's axes are parallel to the walls of the group's bounding box, with its origin at the center.

To jump into a group:

- Double-click the group in the scene window or the Hierarchy.

When you jump into a group, the working box is oriented parallel with the group's bounding box. This enables you to orient objects to the group's Local Universe.

To jump out of a group:

- Click the Jump Out button in the local universe.

When you jump out, the Hierarchy and scene window return to the Global Universe.

Linking Objects

Linking creates a physical connection between objects. In a linked pair, one object is the **parent** and the other is the **child**. When you change the position or orientation of the parent, the child moves with it. However, you can still move or rotate the child independently of the parent.

Linking is used to create articulated structures, such as an arm. The hand is linked to the forearm, which is linked to the upper arm, which is linked to the shoulder.
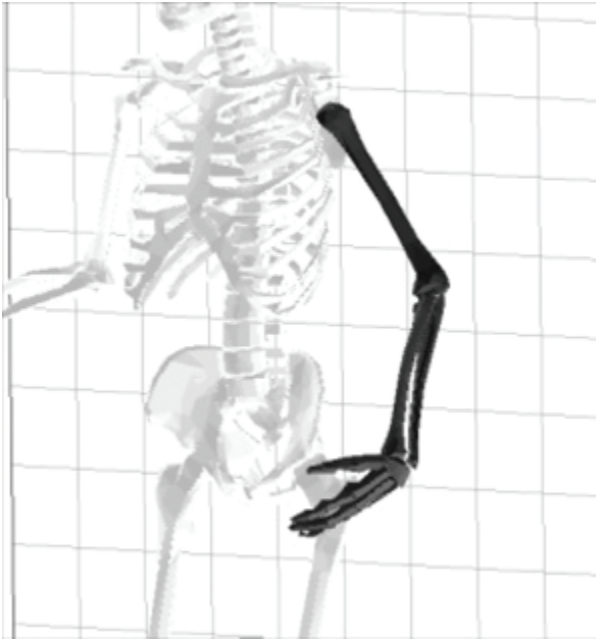
For information on how to constrain the child object's movement, refer to "Applying Constraint Properties."

To link an object to another:

- In the Hierarchy, drag the object or group's listing onto the listing of the desired parent.

To attach an object to a parent that is off screen in the Hierarchy, drag the new element on the first or last visible line of the Hierarchy while holding the mouse button. This automatically scrolls the window's contents to reveal additional elements.
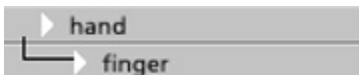
To link an object, drag the object's listing onto the parent.

 NOTE! Duplicating a parent object also duplicates all of its child objects.

To break a link:

- Select the child object's listing and drag it to another level in the Hierarchy.

Note You cannot link a child object to a simple group. To do this, you must first promote the group to a Master object. Then you can link the object (child) to the group (parent). For more information, refer to "Master Object Operations."

Applying Constraint Properties

Linked objects have properties that constrain the child's movement relative to the parent. An excellent real world example of this is the human hand. The hand is linked to the forearm. Its range of motion (relative to the forearm) is limited:you can't bend it forward or back more than 90° or side-to-side more than 45°:anything more would break the wrist. These are the kinds of limits you can impose with constraint properties.
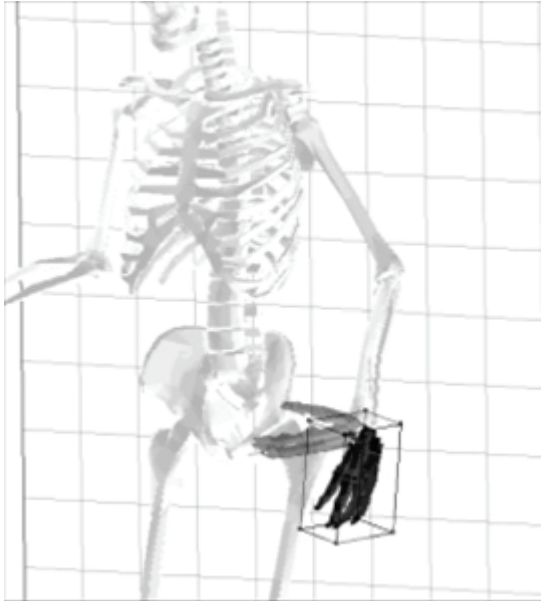
Links become a more powerful tool for creating spatial relationships between objects when you apply a Constraint Property. This is especially true when setting up your scene for animation. Not only can you use constraints to constrain the movement of objects, but by applying the Inverse Kinematics behavior, you can create kinematic chains that allow you to manipulate the parent by moving the child.
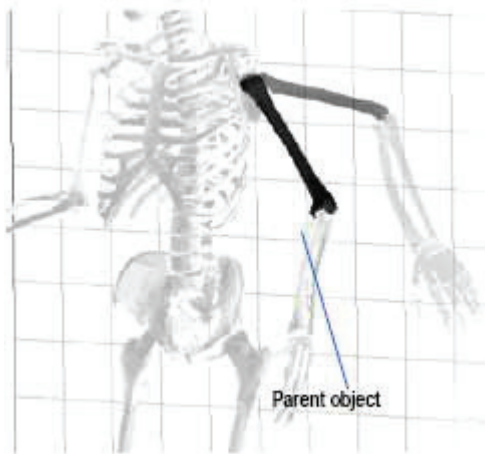
Parent object

To apply a constraint property:

1. Position the child object or group where you want it in relation to the parent.

2. Position the child object or group's hot point where you want it.

3. Select the child object or group.

4. In the **Properties tray: Motion tab: Constraint panel**, choose the type of constraint you want to apply from the list. Each is described in the next section.

5. Choose your options for the specific Constraint Property. For information on doing this, refer to "Setting Constraint Controls."

Types of Constraints

There are several constraint types available: None, 2D Plane, 2D Plane, Axis, Ball Joint, Custom, Shaft, Lock, and Slider. For information on setting rotation controls, refer to "Setting Constraint Controls."

Note You may have difficulty setting up some constraints on joints. As the constraints work in local space, you may have to orient the joints first.

None

None is the default constraint property. You can position the child object anywhere in your scene.

2D Plane

The 2D Plane constraint restricts the movement of your object to a specific plane. This plane is relative to the child object's axis, not the Global Universe. For example, if you rotate the object, the plane that the object moves on tilts. Click the radio button for the plane you want: XY, XZ, or YZ.

Axis

The Axis constraint lets a child object rotate around its hot point on one of the three axes. Examples of the limited axis constraint in the human body are the elbow and the knee. The rotation options are:

*   **Locked** prevents any movement on the selected axis.
*   **Limited** allows you to place limitations on movement in both directions.
*   **Free** lets the object move anywhere along that axis.
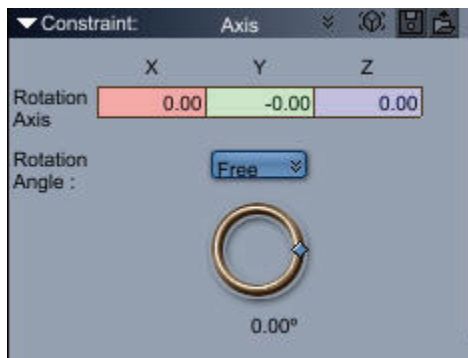
Ball Joint

The Ball Joint constraint lets you rotate the child object around its hot point. By default, the child object can rotate freely around any axis, but you can limit or even lock the rotation on each axis. An example of a limited ball joint constraint in the human body is the shoulder.

Custom

The Custom constraint lets you build your own combination of constraints using sliders and axis rotation controls.
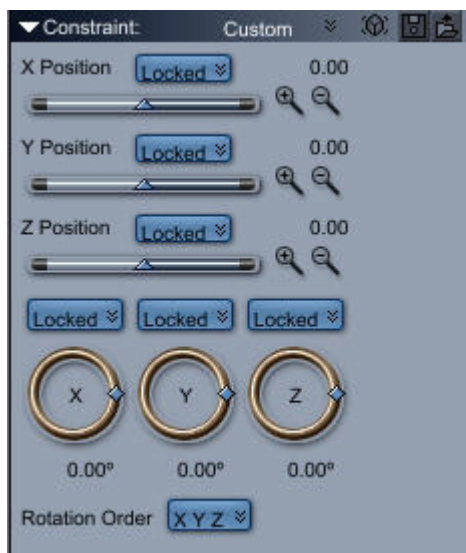
Lock

The Lock constraint locks the child object to the parent object. You cannot select the child object and move it, but it does move in relation to the parent when the parent is moved.

There are no options for the Lock constraint.

Shaft

The Shaft constraint lets the child object both rotate around one of its axes and move up and down the same axis. One way to think of a shaft constraint is to visualize a firehouse pole. The firefighters can slide down the pole while rotating around it.

Choose the axis of rotation and set both rotation and slider controls. For details on setting these controls, refer to "Setting Constraint Controls."
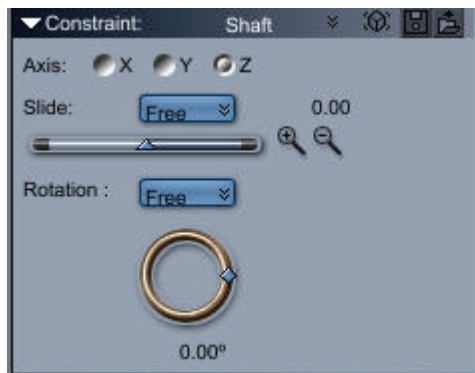
Slider

The Slider constraint sets constraints for the child object's movement along its X, Y, and Z axes. The axis movement options are:

- **Locked** prevents any movement on that axis.

- **Limited** allows you to place limitations on movement in both directions.

- **Free** lets the object move anywhere along that axis.
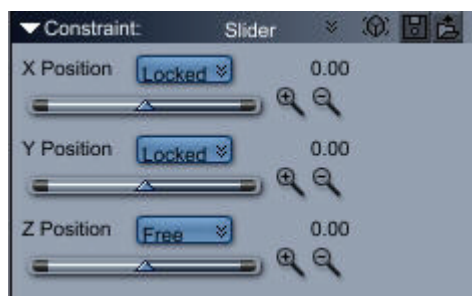
Setting Constraint Controls

All constraint properties use two primary types of controls: slider and axis rotation.

The constraint slider controls are found in the Custom, Shaft, and Slider constraint options.

The Axis rotation controls are found in the Axis, Ball Joint, Custom, and Shaft constraint options.

To set a constraint slider control:

1. In the **Properties tray: Motion tab: Constraint panel**, choose Custom, Shaft, or Slider from the list.

2. For the constraint you've chosen, select the type of slider control:Locked, Limited, or Free. Carrara displays the controls for the chosen constraint.

3. Drag the slider controls to set position limits and a new position.

In Free mode, the slider has one control. Drag the control to move the object along that axis. You can double-click the slider to set the current position numerically.

In Limited mode, the slider has three controls:

- The control on the left sets the outer constraint in the negative direction away from the object's hot point. Drag the control to change the limit.

- The control on the right sets the outer constraint in the positive direction away from the object's hot point. Drag it to set a new limit.

- The control in the middle sets the current position of the object. Drag the control to move the object along that axis between the limits you have set.
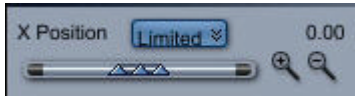
The text field to the right displays the position of the currently selected control.

- If you want more precise control in the slider range, click the plus (+) magnifying glass icon to zoom in on the scale.

- If the slider range is not large enough for you to set the constraint limit where you want, click the minus (-) magnifying glass icon. Click it as many times as necessary to zoom out on the slider scale.

To set axis rotation controls:

1. In the **Properties tray: Motion tab: Constraint panel**, choose Axis, Ball Joint, Custom, or Shaft from the list.

2. For the constraint you've chosen, select the type of axis rotation control:Locked, Limited, or Free.

   Carrara displays the controls for the chosen constraint.
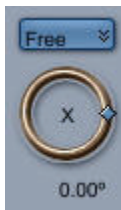
3. Drag the controls to set position limits and a new position.

   - In Free mode, the angle ring has one control. Drag it to rotate the object up to 360° around the chosen axis.
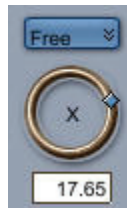
   - In Limited mode, the angle ring displays three controls.

     - The top control defines the rotation constraints on the positive side of the axis. Drag the control to change the limit.

     - The lower control defines rotation constraints for the negative side of the axis. Drag the control to change the limit.

- The middle control sets the current position of the object. Drag the control to rotate the object between the limits you have set.

Controlling Constraints Directly

The Direct Manipulation option for Constraint properties provides an on screen description and control of the objects' motion and rotation range. You can drag handles on the wires to control these properties.

NOTE! There are no Direct Manipulation controls if the constraint type is None.

To use the Direct Manipulation controls for constraint properties:

1. Follow the previous instructions to apply a Constraint property to an object and set constraint controls on it.

2. After you've chosen the constraint type, click the Direct Manipulation button in the Properties tray.
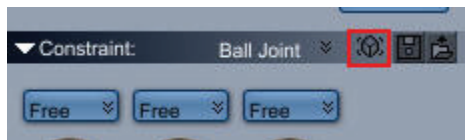
The controls appear on the object as a set of wires with "handles" at certain points. The appearance of the wires depends on the type of constraint.
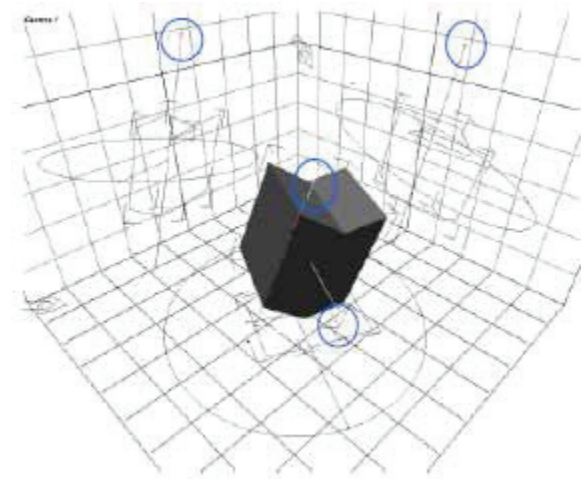
For slider controls, you'll see a straight wire. If the constraint is Limited, the handles at the ends determine the limit for sliding on that axis.

NOTE! If you don't see the wire, it's probably inside the object. Use the constraint slider to extend the range.

To set slider options directly:

1. Drag the handle at the end of the wire to change the limit in that direction.

2. Drag the object to change its position along the wire. The object's hot point slides along the wire until it reaches the end.
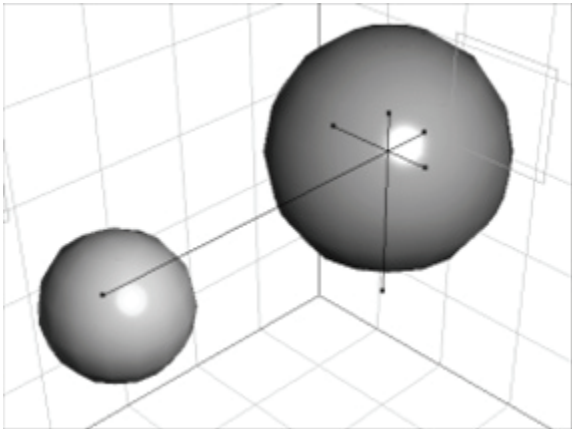
For rotation controls, you'll see a wire arc that describes the object's angular rotation range. The line within the arc describes the current rotation.

To set rotation controls directly:

1. Drag the handles at the end of the arc to change the rotation limit.

2. Drag the center handle on the arc to rotate the object within the range.

Building Chains of Constraints

The parent-child constraint enables you to easily manipulate multiple objects from a number of reference points by creating chains of constrained objects. This feature is useful for creating articulations, especially when used in conjunction with the Inverse Kinematics feature.

For example, if you want to create a fully articulated arm:one that could bend at the shoulder, elbow, wrist, and each of the five finger joints:you should arrange the objects to construct the arm, then constrain them into a chain with the fingers at the deepest level of the Hierarchy and the shoulder at the root.
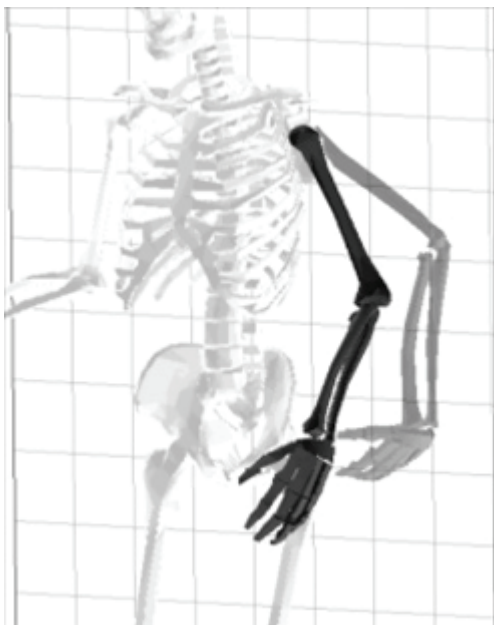
Collision Detection

Collision Detection instructs objects to "respect each other's space." With Collision Detection enabled, you can drag one object toward another, and the object stops when its surface contacts the other object. If you continue to drag, the object snaps through to the other side, but you can't leave it embedded.

Collision Detection is a great feature for bringing the surfaces of two objects together:for example, putting plates and silverware on a dinner table. Without Collision Detection, you might accidentally leave a knife floating just above the table or embed it in the table itself.

To enable or disable Collision Detection:

- Choose **View > Use Collision Detection** or click on the **Collision Detection** icon on the right of the 3DView window.

NOTE! Collision Detection is effective only when you reposition objects by dragging. Nudging and numerical repositioning are not affected by Collision Detection.

Working with Master Objects

Whenever you add a new object to a scene, you create a Master object. Most scenes use one or more duplicates of Master objects. These duplicates are not independent objects but Instances of the Master object. For example, you may create a chair using one Master object called "chair leg" and three Instances of it.
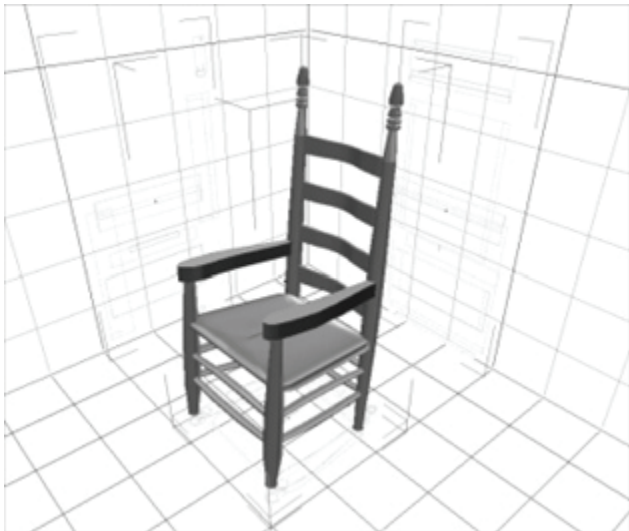
The position, orientation, size, hot point location, and shading of each object Instance may be unique. The shape, however, is common.

Creating an Instance of a Master Object

Every time you add a new object to a scene, you create a Master. You create an object Instance of the Master in one of three ways:

- Use **Edit > Duplicate**.

- Use Copy and Paste.

- Drag the object from the Master category into the scene window.

For information on special duplicating features, refer to "Duplicating Objects."

Objects and Masters in the Hierarchy

The Hierarchy in the Sequencer contains three categories that change your view of the objects displayed in the Hierarchy.

The Universe category displays all the objects and object instances in your scene.
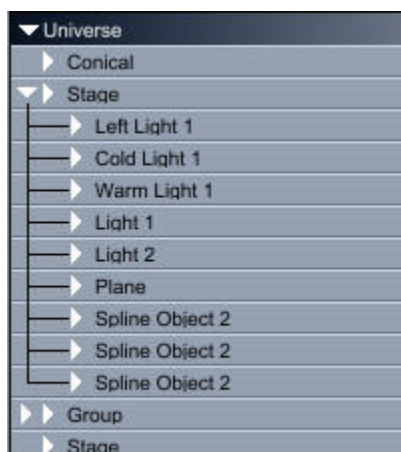
The Master Objects category displays only the Master objects.
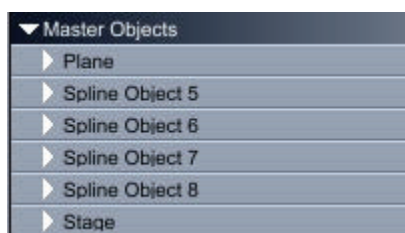
The Scene Effects category displays any rendering effects you may have applied to your scene.
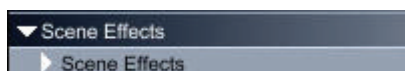
Unlike the Universe category, the Master Objects and Scene Effects categories do not let you structure or group objects.

Editing an Instance

When you edit an object Instance, Carrara displays a dialog in which you can modify the Master object of this class or create a new Master from this Instance.

NOTE! You may have multiple Instances with the same name, but you may not have two Master objects with the same name. When you create a new Master object by modifying an object Instance, Carrara names the new Master object by appending a number to the old name.

Master Object Operations

When you modify a Master object, all object Instances are automatically updated to the new form. Their individual position, orientation, scale, symmetry, and region shading remain unchanged.

If you modify an Instance when you had intended to modify the Master object, you can easily get back on track by replacing the Master with your modified Instance.

To modify a Master object:

1.   In the Hierarchy, click the Master Objects category.

2.   Double-click the listing of the Master object you wish to modify.

3.   Modify the object using the modeler or the set of controls that appears.

To use a group as a Master object:

1.   Select the group.

2.   Choose **Edit > Promote to Master**.

A Master group operates under the same rules as a Master object. If you jump into a Master group and modify its contents in any way:the number of elements, their relative positions, or orientations:those changes will update to the group Instances. If you jump into one Instance of the Master group and modify its contents, you separate this group Instance from its Master, thus creating a new Master group.

Changing an Object's Symmetry

Many real-world objects exhibit symmetry. Airplanes, automobiles, and the human body are a few examples. To help you build complex symmetrical objects, Carrara provides the Flip and Duplicate with Symmetry commands.

The Symmetry Plane

The symmetry commands let you choose the plane across which objects are reflected or flipped.

The attitude of the working box and the Local universe's Z coordinate are important for a successful Flip or Duplicate with Symmetry operation. Flip repositions or reorients the object, while Duplicate with Symmetry makes a second copy in a symmetrical attitude.

- To flip an object in place, position the center of the object at Z=0 in the working box's coordinate system.

- To create a mirrored duplicate precisely alongside the original, set the ground plane of the working box parallel to the plane that will be the center of the symmetrical object. (This will be parallel to one side of the object's bounding box). Move the working box to put working box Z=0 in contact with that side of the object's bounding box.

To flip an object:

1.   Orient and position the working box to put the plane of symmetry where you want it.

2.   Select the object you want to flip.

3. Choose **Edit > Flip**. The Flip dialog appears.
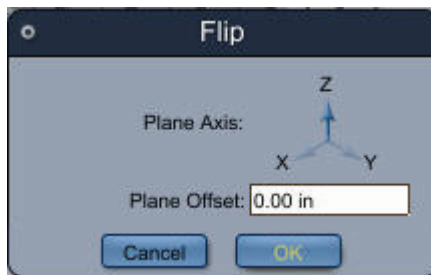
4. Click to select the plane axis along which you want to flip the object.

5. If you want to offset the plane along which the object flips, enter a value in the Plane Offset field.

6. Click OK. Carrara flips the object across the plane of symmetry.

To duplicate with symmetry:

1. Orient and position the working box to put the plane of symmetry where you want it.

2. Select the object.

3. Choose **Edit > Duplicate with Symmetry**. The Duplicate with Symmetry dialog appears.
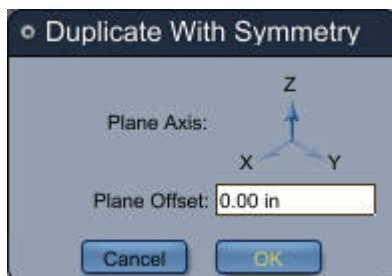
4. Click to select the plane axis along which you want to duplicate the object.

5. If you want to offset the plane along which the object duplicates, enter a value in the Plane Offset field.

6. Click OK. Carrara duplicates the object across the plane of symmetry.

Splitting Objects

You can split objects into their component pieces. For example, you can split a line of text into individual letters or you can break a vertex object into polymeshes.

To split an object:

1. Select the object you want to split.

2. Choose **Edit > Split Object**. The object splits into its component pieces.

Replicating Objects (Standard, Pro)

Duplicating objects can sometimes be a very long process for scenes like a city or a forest, especially when you don't want your objects to be exactly the same. To help you duplicate objects in very little time and using little memory, Carrara 6 provides two tools: the Replicator and the Surface Replicator. They do almost the same thing, replicating your objects, but in different ways.

The Replicator

The Replicator duplicates objects along basic shapes: grid, rectangle, circles, etc. It can be used to create armies, walls or cities. Once you have created the replicator and set the objects you want to replicate, choose the settings in the Replicator modeler.

- The **Grid Cell** determines different things according to the replication method, but basically it is the distance between two neighboring objects.

- The **Autogrid** replication method will set the grid cell to biggest bounding box of the replicated objects.

- The **Random Transform** lets you create objects that are not positioned uniformly by changing rotation, scale or position randomly on each object. The settings are the amplitude of the error on each parameter.

- The **Seamless** option can only be used with the Autogrid replication method and switches the orientation of one of two replicated objects. For example, if you set a terrain as the duplicated object, this will result as a large, continuous terrain.

The Surface Replicator

The Surface Replicator duplicates objects randomly on the surface of another object. You could use this to quickly generate forests, vegetation or rocks on terrain, or hair on a character. Once you have created the replicator and set the objects you want to replicate, choose the settings in the Surface Replicator modeler.
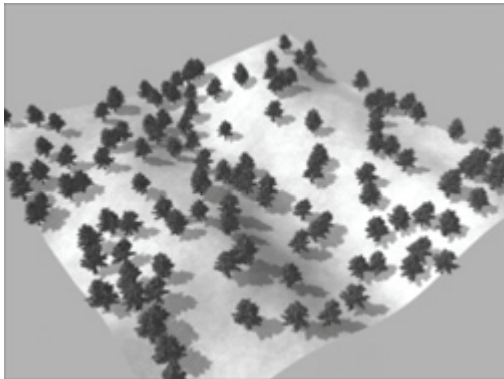
The surface used to generate objects is the surface of the Source Object that can be a plane or a terrain, for example. Use the choose button to select it from the scene. The replicated objects will be linked to the

source object: if you modify the source object, the generated objects will move; if you delete it, the generated objects will disappear.

- The **number of objects** you can choose is only an approximate number as there may not be enough space on the surface to place all the objects. If this is the case, you will see a warning at the bottom of the dialog.

- The **Random Transform** lets you create objects that are not all the same by changing rotation or scale randomly on each object. The settings are the amplitude of the error on each parameter.

- **Align object to normal** determines whether the objects should be orientated like the original object or be orientated like the surface they lay on, using the local normal. To generate a forest you will prefer to uncheck this, where as for hairs it is better to have it checked.

The distribution of the objects can be automatic, but you can also choose globally where you want to put your objects.

- If the source object has several shading domains, you can limit the distribution of replicated objects to a **shading domain**.

- **Use shader** sets the distribution of objects to follow the shader of the generator. The generator will try to place objects on bright areas first and black areas won't get any objects.

- If your objects should not overlap (as is the case most of the time) you can set a **Minimum distance between objects**.

- If you want to make thickets of trees or other clustered objects, you can use **Objects try to cluster**. Specify the **average distance** objects should have in clusters and apply a **Strictness**. High strictness will tend to leave no object by itself.

Using the Replicators

To replicate objects

1. Insert one type of replicator in your scene by the method you choose:

    - **Insert > Replicator** or **Insert > Surface Replicator**.

    - Drag one of these tools to the scene window.

2. Insert the objects to generate as sons of the generator:

    - In the hierarchy move the object or the group you want to generate on the replicator, or;

    - In the replicator properties use the button **Add**.

        If you want to randomly generate different types of objects, such as different types of trees or buildings, reiterate step 2 above. For example, if you want to create a forest with 75% pine trees and 25% maple trees, insert three pine trees and one maple tree as sons of the generator.

3. Choose the generating point on each object: objects are generated from their hot point. For trees you will want to generate them from their base, so move the hot points to the bases of the trees.

4. Edit the generator properties by:

    - Double clicking on the generator, or;

    - Selecting the generator and the switching to the Model Room.

5. In the Assemble Room you can adjust the **display** of the generated objects. This setting won't affect the final render but lets you work with a clean scene.

6. If you are happy with your replication, you can create instances for real to work with it using the **Create** button in the modeling room. But in most cases you should not do this because:

- The properties of the generators won't be accessible

- Carrara might be slower because of all these objects especially the 3D View

Using the Counter

The counter tracks the complexity of the objects and geometry in the scene. The counter displays:

- The selected object

- The number of objects in the scene

- The number of modeled facets

This information may be useful if you intend to export the scene for use in another program. The number of triangles has a direct correlation to file size in the exported document.

To view the counter:

- Choose **Edit > Counter**.

The counter keeps track of the number of objects in the scene and their complexity.

Modifiers

Modifiers are a special set of tools that let you easily manipulate objects in ways that would otherwise require many arranging and remodeling operations.

For example, using the Explode modifier, you can make an object appear to explode by setting only a few parameters. Without the modifier, you would have to remodel the object several times to achieve the same effect. Some modifiers, like Bounce and Spin, automatically animate common types of motion. Others, like Point At and Track, animate one object based on the position or orientation of a second object.

Modifiers can also be animated. You can create complicated animation effects like explosions and dissolves just by applying different modifiers to your objects.

Modifiers also allow you to easily animate the shape of objects and groups without having to edit them in a modeler. They can also be used to animate models you import from other applications.

Applying Modifiers

You can access all the modifiers available in Carrara from the Modifiers tab on the Properties tray. Each modifier has its own settings, but they're all applied in the same way. You can apply one or more modifiers, as well as remove, replace, and reorder modifiers.

To apply one or more modifiers:

1. Select the object or group you want to modify. If you want to apply a modifier to more than one object, you must group the objects together with the **Edit > Group** command, then apply the modifier to the group.

2. If the Properties tray is not open, choose **Windows > Properties** to display it.

3. Click the Modifiers tab

4.   Click the plus (+) icon to select a modifier from the list.
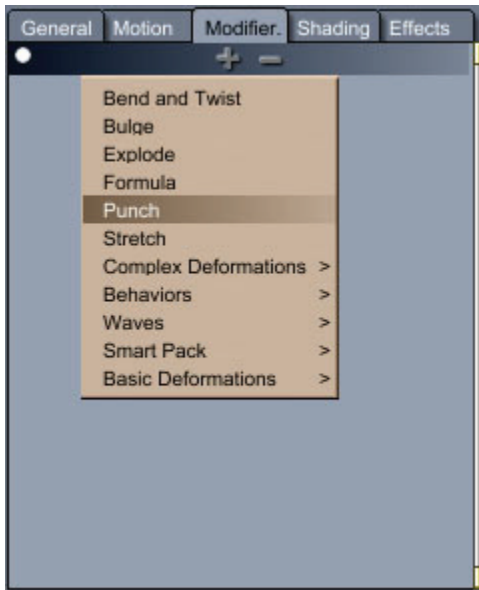
5.   Specify the modifier's parameters by adjusting the sliders and enabling the relevant options.

NOTE! When you apply a modifier, it remains in the object's modifier list for the duration of an animation. However, some modifiers can be turned on and off at different points along the time line.

6.   To apply another modifier, click the plus (+) icon to select another modifier from the list.

NOTE! Some modifiers are exclusive:they cannot be used together. For example, it wouldn't make sense to assign the Point At modifier to an object and also apply Spin:these modifiers combined would produce contradictory results.

To remove a modifier from an object or group:

1.   Select the object or group that carries the modification.

2.   Click **Properties tray: Modifiers tab**.

3.   Click to select the title of the modifier you want to remove. The title bar changes color to indicate it is selected.

4.   Click the minus (-) icon or press the Delete key. The modifier is removed from the list.

To replace a modifier:

1.   Select the object or group that carries the modification.

2.   In the Properties tray, click the Modifiers tab.

3.   Click the Modifier list pop-up menu on the panel of the modifier you want to replace.

4.   Choose the new modifier you want. The modifier is replaced.

If you are using more than one modifier for an object, you can change the order of the modifiers as they appear in the Properties tray. Where two or more applied modifiers conflict, those further down the list override those above.

To change the order of modifiers:

1. Select the object or group that carries the modification.

2. Click **Properties tray: Modifiers tab**.

3. Drag the title bar of a modifier to its new location. A locator line indicates where the modifier will be moved.

Applying Modifiers in the Model Room (Standard, Pro)

The Assemble Room modifiers can also be applied to a selection or an object.

To apply a modifier on a selection:

1. Select the polygons to be affected.

2. Choose the menu **Model > Deform**. The Deform Selection dialog appear.
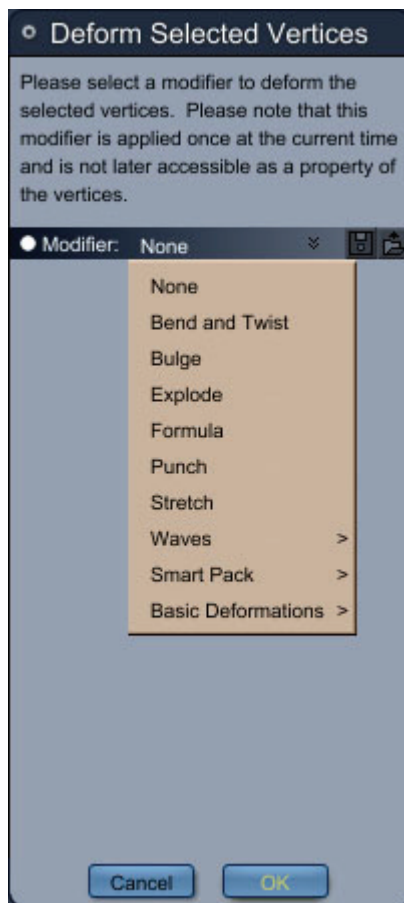
3. Choose a modifier and adjust their settings.

4. Click OK.

Controlling a Modifier Directly

The Direct Manipulation control lets you adjust a modifier's attributes directly on the object. Direct Manipulation controls appear as a set of wires with "handles" at certain points. The handles represent attributes you can change by dragging.

A modifier's title bar contains one of three icons to describe the status of Direct Manipulation. You can turn Direct Manipulation On, Off, or On When Selected (Direct Manipulation is on whenever the object is selected).

- Direct Manipulation On

- Direct Manipulation Off

- Direct Manipulation On When Selected

With some modifiers, you cannot use Direct Manipulation. For those modifiers, the title bar always displays Direct Manipulation off.

To use Direct Manipulation on a modifier:

1. Select an object.

2. In the Properties tray, click the >Modifiers tab and apply a modifier to the object.

3. Click the Direct Manipulation icon in the modifier title bar until the On icon appears. The modifier's control handles appear on the object in the scene.
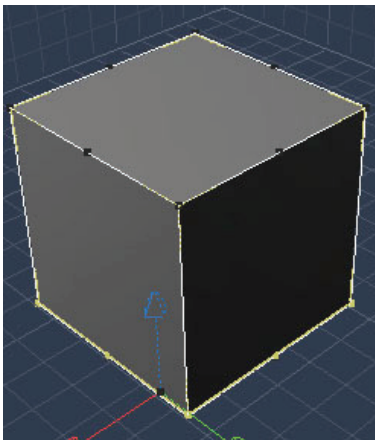
Bend and Twist

The Bend and Twist modifier bends and/or twists the surface of an object. This modifier is particularly useful for fine-tuning complex models such as bent compound shapes, which are otherwise tricky or impossible to achieve.
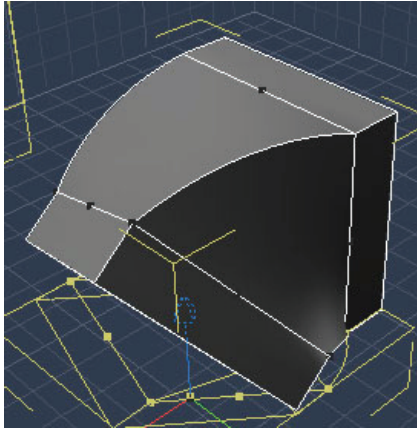
To bend or twist an object:

1. Select the object you want to twist.

2. Click **Properties tray: Modifiers tab**.

3. Choose Bend and Twist from the modifiers list pop-up menu.

4. If you want to use Direct Manipulation, click its icon in the title bar until the **On** or **On When Selected** icon appears. A bounding box with twelve points appears around the object.
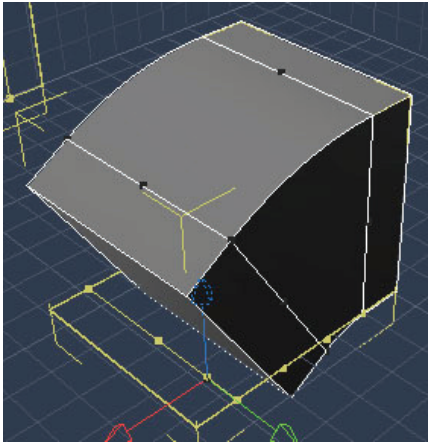
5. Do one or both of the following:

    -   **To twist an object:** Drag one of the corner points in the direction you want to twist. Or, drag the **Twist Angle** slider to control the direction and intensity of the twist.

    -   **To bend an object**: Drag one of the middle points in the direction you want to bend. Or, drag the **Bend Angle** slider to control the intensity of the bend, and the **Bend Axis Angle** slider to control the direction of the bend.

6. Adjust the following properties in the **Properties tray: Modifier tab: Bend and Twist panel**:

    -   **Bend Axis Angle**: Use the Bend Axis Angle slider to control the direction of the bend.

    -   **Limits**: Controls the portion of the object on which the twist or bend is applied. The slider contains two controls. The left control sets the bounding box for one side of the axis, and the right control sets the bounding box limit for the other side. For example, if the slider is set at 25% and 75%, the bend or twist starts at 25% from the bottom of the object and continues to the upper 75% of the object. At 0% and 100%, the entire object is modified.

- **Axis**: Select the radio button for the Axis where you want the modification to occur. The controls move to the selected axis.

- **In**: Check the axes in which you want the bend or twist action to apply.

NOTE! You cannot enable the checkbox for the selected Axis. For example, if the Axis X radio button is selected, you cannot enable the In X checkbox.

- **Flip Along Major Axis**: If you want to reverse the direction of the bend or twist, check or uncheck the Flip along major axis checkbox.

Bulge

The Bulge modifier bulges the middle of an object in or out in a uniform fashion. The bulge occurs most in the middle of the object and not at all at the very top and bottom. The selection bulges uniformly based on the axis and points you choose.
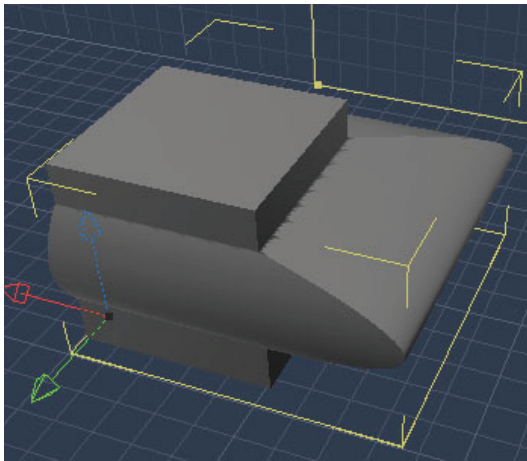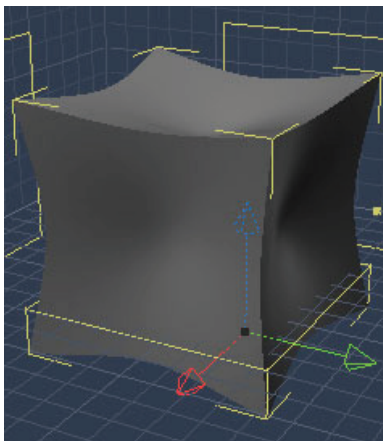

To set Bulge attributes:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab**.

3. Choose Bulge from the modifiers list pop-up menu.

4. Adjust the following options in the **Properties tray: Modifier tab: Bulge panel**:

    - **Bulge**: Controls the direction and intensity of the bulge.

    - **Limits**: Controls the portion of the object on which the bulge is applied. The slider contains two controls. The left control sets the bounding box for one side of the axis, and the right control sets the bounding box limit for the other side. For example, if the slider is set at 25% and 75%, the bulge starts at 25% from the bottom of the object and continues to the upper 75% of the object. At 0% and 100%, the entire object is modified.

    - **Axis**: Select the radio button for the Axis where you want the modification to occur. The controls move to the selected axis.

    - **In**: Check the axes in which you want the bulge action to apply.

NOTE! You cannot enable the checkbox for the selected Axis. For example, if the Axis X radio button is selected, you cannot enable the In X checkbox.

Explode

The Explode modifier causes the object to burst apart.
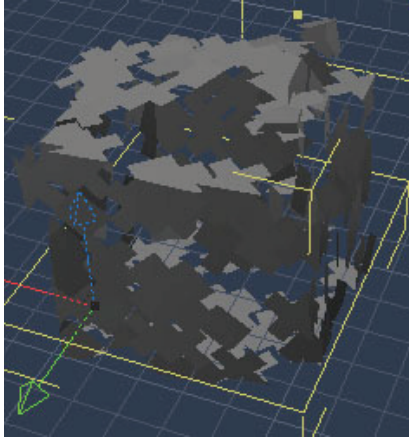
To explode an object:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab**.

3. Choose **Explode** from the modifiers list pop-up menu.

4. Adjust the following options in the **Properties tray: Modifier tab: Explode panel**:

   - **Completion**: Use the Completion slider to control how much the object explodes. In an animation, set this value to 0% in the first frame and 100% in the last. If you apply gravity, the pieces gradually fall to the ground as the explosion progresses.

   - **Size of Pieces**: Use the Size of Pieces slider to control the range of sizes for the pieces created by the explosion. The right control sets the size of the largest pieces, and the left control sets the size of the smallest pieces. With the controls set farther apart there is a greater variety of sizes for the pieces. With the two controls set closer together the difference between the largest and smallest pieces is less dramatic.

   - **Speed**: Controls the speed of the pieces as they move away from the original position of the object. The slider contains two controls. The left control sets the speed for the slowest moving particles; the right control sets the speed for the fastest moving particles. With the left control set to the far left (0%) and the right control set to the far right (100%), you have the greatest variety of speed.

   - **Gravity**: Controls the amount of gravity applied to the pieces. The higher the setting, the faster the pieces fall downward.

   - **Slow Down at End**: Control the rate at which the pieces slow down as they get farther from the center of explosion. A higher value makes the pieces move more slowly at the end of an animation.

   - **Rotational Speed**: control the speed at which the pieces rotate as they move away from the original position of the object. The slider contains two controls. The left control sets

the speed for the slowest moving pieces; the right control sets the speed for the fastest moving pieces.

- **Large Pieces Move Slower**: Enable this checkbox if you want larger pieces to move more slowly during the explosion
- **Pieces Stop at Bottom**: Enable this checkbox if you want all the pieces to stop falling when they reach the bottom of the original object's bounding box.
- **Explode from Top**: Enable the Explode from Top checkbox if you want the object to explode starting from the top and working down to the bottom.

Formula

The Formula modifier uses mathematical equations to modify a selected object. You can use the Formula Editor to input variables, operators and parameters. Formula modifying is a technical process; for more information refer to "Using Formulas in Carrara."

Punch

The Punch modifier punches a dent into one side of an object, or bulges it outward.
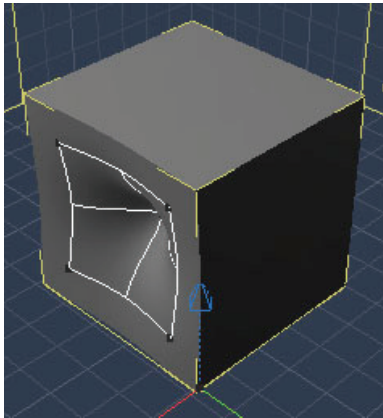
To set Punch attributes:

1. Select the object you want to modify.
2. Click **Properties tray: Modifiers tab**.
3. Choose **Punch** from the modifiers list pop-up menu.
4. If you want to use Direct Manipulation, click its icon in the title bar until the **On** icon appears. A square with five points appears in the specified axis.
5. Adjust the following options in the **Properties tray: Modifier tab: Punch panel**:
   - **Strength**: Drag the Strength slider right to increase the strength of the Punch, or to the left to create a bulge instead of a dent.
   - **Radius**: Drag the Radius slider to the right to increase the area of the object that is punched, or to the left to decrease the area.
   - **Axis**: Select the radio button for the Axis where you want the punch to occur. The punch controls move to the selected axis.
   - **Punch Other Side**: Enable the Punch other side checkbox if you want to place the dent on the other side of the object.

Stretch

The Stretch modifier is particularly suitable for simulating the exaggerated, cartoon-style effects of squash and stretch actions. A squashed object seems to bulge; a stretched object elongates while thinning in the middle, much like chewing gum.
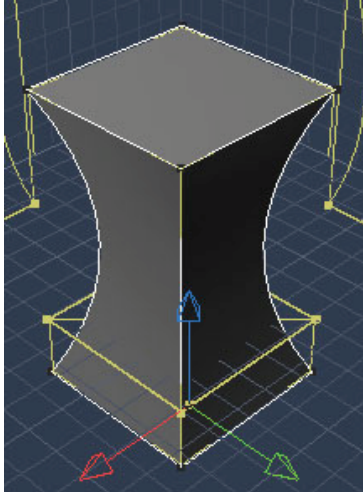
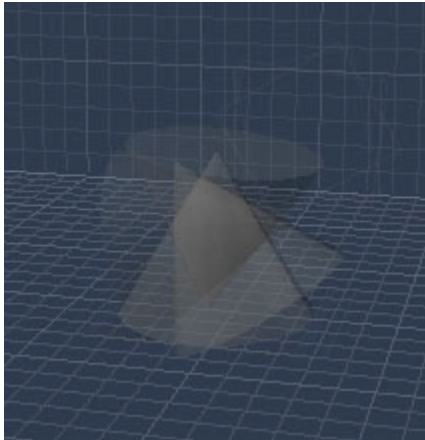To set Stretch attributes:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab**.

3. Choose **Stretch** from the modifiers list pop-up menu.

4. If you want to use Direct Manipulation, click its icon in the title bar until the **On** icon appears. A bounding box with eight points appears around the object.

5. Adjust the following options in the **Properties tray: Modifier tab: Stretch panel**:

   - **Strength**: Adjust the value of the Stretch slider to stretch or squash the object. A value lower than 100 compresses and bulges the object; a value greater than 100 elongates and thins the object.

   - **Axis**: Select the radio button for the Axis where you want the stretch to occur. The stretch controls move to the selected axis.

Basic Deformations

Basic deformations allow you to perform the following modifications to an object:

Offset

The Offset modifier translates an object in relation to its default position, within one or more specified axes.
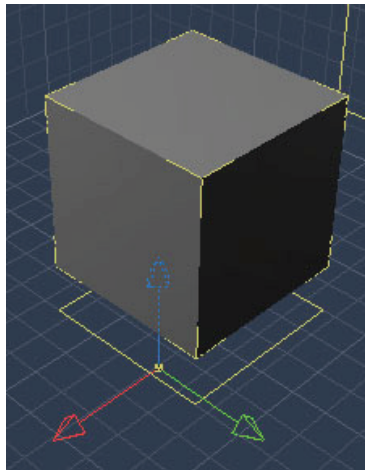
To set Offset attributes:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab**.

3. Choose **Basic Deformations > Offset** from the modifiers list pop-up menu.

4. Adjust the following options in the **Properties tray: Modifier tab: Offset panel**:

    -   **Offset**: Use the slider to adjust the amount of offset from -150% to 150%.

    -   **In**: Check the axes in which you want the offset action to apply.

Rotate

The Rotate modifier turns an object on the specified axis.
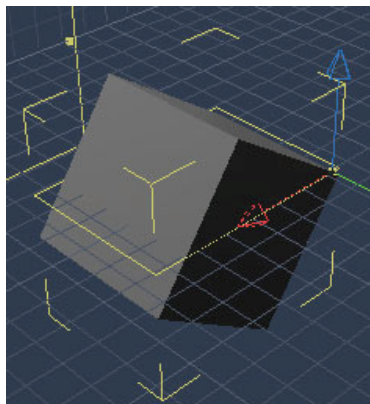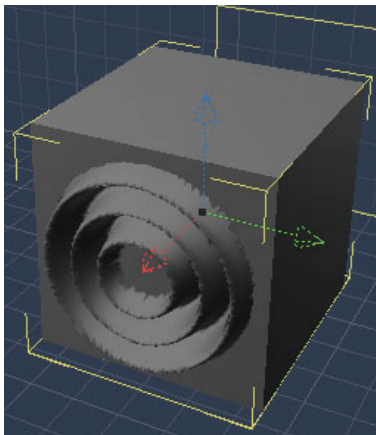

To set Rotate attributes:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab**.

3. Choose **Basic Deformations > Rotate** from the modifiers list pop-up menu.

4. Adjust the following options in the **Properties tray: Modifier tab: Rotate panel**:

    -   **Rotation Angle**: Use the slider to adjust the amount of rotation from -180 to 180 degrees.

    -   **Axis**: Check the axes in which you want the offset action to apply.

Scale

The Scale modifier resizes an object along the specified axes. If all three (X, Y, Z) axes are selected, the object scales uniformly. You can use the Limits sliders to control the area that is scaled.
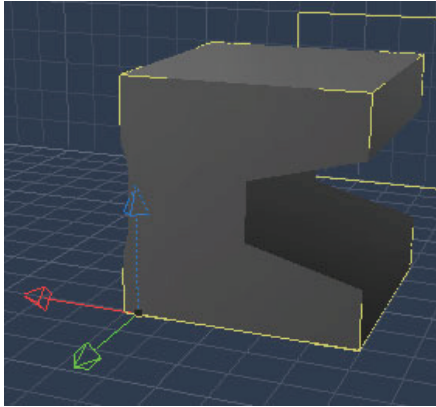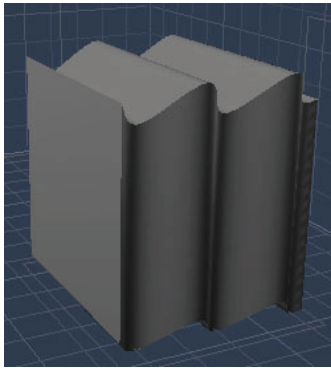

To set Scale attributes:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab**.

3. Choose **Basic Deformations> Scale** from the modifiers list pop-up menu.

4. Adjust the following options in the **Properties tray: Modifier tab: Scale panel**:

   - **Scale**: Use the slider to adjust the amount of scale from 0 to 300%.

   - **Limits:** Use the sliders to control the portion of the object on which the scale is applied. The left control sets the bounding box for one side of the axis, while the right control sets the bounding box limit for the other side. For example, if the slider is set at 25% and 75%, the scale starts at 25% from the bottom of the object and continues to the upper 75% of the object. At 0% and 100%, the entire object is modified.

   - **In**: Enable the checkboxes for each axis on which you want the scale to occur.

Shatter

Shatter is a facet-based modifier you can use to simulate object explosions.
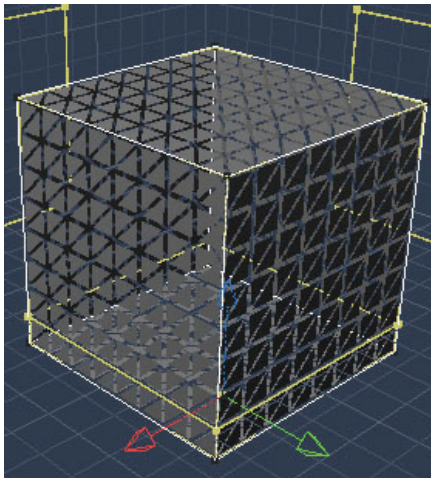
To set Shatter attributes:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab**.

3. Choose **Basic Deformations > Shatter** from the modifiers list pop-up menu.

4. Adjust the following options in the **Properties tray: Modifier tab: Shatter panel**:

    -   Drag the **Scale** slider to the right to increase the scale of the shatter or left to decrease the scale.

Shear

Use the Shear modifier to tilt an object while leaving the top and bottom parallel to one another. This creates a "windblown" effect. You can use the Limits sliders to control the area of the object that is sheared.
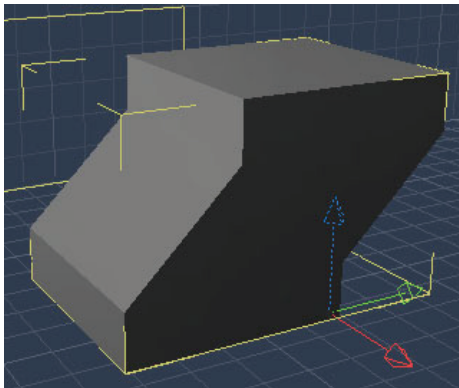
To set Shear attributes:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab**.

3. Choose **Basic Deformations> Shear** from the modifiers list pop-up menu.

4. Adjust the following options in the **Properties tray: Modifier tab: Shear panel**:

    -   **Shear**: Use the Shear slider to control the amount of the shear effect.

    -   **Limits**: Controls the portion of the object on which the shear is applied. The slider contains two controls. The left control sets the bounding box for one side of the axis, and the right control sets the bounding box limit for the other side. For example, if the slider is set at 25% and 75%, the shear starts at 25% from the bottom of the object and continues to the upper 75% of the object. At 0% and 100%, the entire object is modified.

    -   **Axis**: Select the radio button for the Axis where you want the modification to occur. The controls move to the selected axis.

        NOTE! You cannot enable the checkbox for the selected Axis. For example, if the Axis X radio button is selected, you cannot enable the In X checkbox.

    -   **In**: Check the axes in which you want the bend or twist action to apply.

    -   **Flip Along Major Axis**: Check this option to shear the object in the opposite direction.

Taper

The Taper modifier adjusts the scale of an object at one end only.
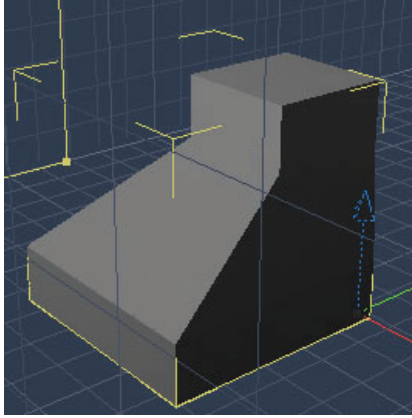

To set Taper attributes:

1. Select the object you want to modify.
2. Click **Properties tray: Modifiers tab**.
3. Choose **Basic Deformations> Taper** from the modifiers list pop-up menu.
4. Adjust the following options in the **Properties tray: Modifier tab: Taper panel**:

    - **Taper**: Use the Taper slider to control the amount of the shear effect.
    - **Limits**: Controls the portion of the object on which the taper is applied. The left control sets the bounding box for one side of the axis, and the right control sets the bounding box limit for the other side. For example, if the slider is set at 25% and 75%, the taper starts at 25% from the bottom of the object and continues to the upper 75% of the object. At 0% and 100%, the entire object is modified.
    - **Axis**: Select the radio button for the Axis where you want the modification to occur. The controls move to the selected axis.

      NOTE! You cannot enable the checkbox for the selected Axis. For example, if the Axis X radio button is selected, you cannot enable the In X checkbox.
    - **In**: Check the axes in which you want the taper action to apply.
    - **Flip Along Major Axis**: Check this option to taper the object in the opposite direction.

Complex Deformations

In the Modifiers area, you can perform complex deformations on objects.

Atomize

The Atomize modifier replaces the surface of the selected object with small balls which are then slightly scrambled or wiggled.
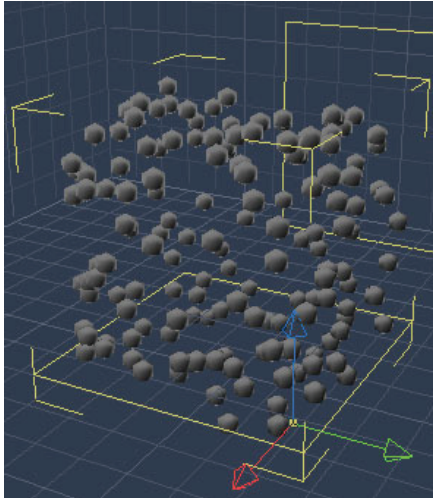
To set Atomize attributes:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab.**

3. Choose **Complex Deformations> Atomize f**rom the modifiers list pop-up menu.

4. Adjust the following options in the **Properties tray: Modifier tab: Atomize panel**:

    -   **Wiggle Effect**: Controls the amount that the particles move.

    -   **Particles Density**: Drag the slider to the left to decrease the density, or to the right to increase the density. A setting of 10 makes the object solid again.

    -   **Size of Particles**: Drag the Size of Pieces slider to the right to increase the size of the balls on the object's surface. Drag to the left to decrease the size of the balls.

Black Hole

The Black Hole modifier creates a vortex effect, similar to the way water swirls as it is being sucked down a drain. In an animation the Black Hole modifier creates a whirling, circular motion that tends to form a cavity or vacuum at the center of its action.
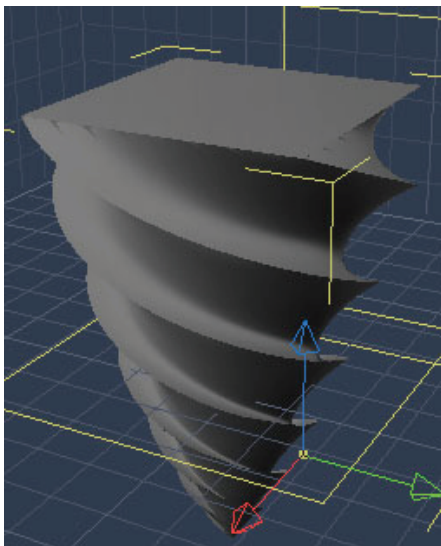
To set Black Hole attributes:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab**.

3. Choose **Complex Deformations> Black Hole** from the modifiers list pop-up menu.

4. Adjust the following options in the **Properties tray: Modifier tab: Black Hole panel**:

   - **Completion**: Controls the state of the vortex. Set this value to 0% at the beginning of an animation, and 100% at the end.

   - **Winding**: Controls the rotation as the Black Hole deforms. A negative setting creates a counter-clockwise rotation.

   - **Spin Speed**: Drag the slider to the right to make the object turn faster as it travels down the vortex.

   - **Depth**: Use the slider to control how far below the object the center of gravity is placed. A low setting results in little gravity pull. A high setting results in a stronger pull.

Dissolve

The Dissolve modifier reduces the object to triangles or polygons then diminishes the object so that it gradually fades away or disintegrates. You can use this modifier to create interesting sci-fi effects such as a transporter, or a mummy dissolving before your eyes in a horror movie.
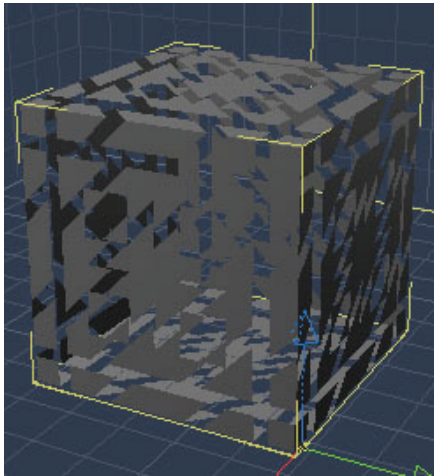
To set Dissolve attributes:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab**.

3. Choose **Complex Deformations> Dissolve** from the modifiers list pop-up menu.

4. Adjust the following options in the **Properties tray: Modifier tab: Dissolve panel**:

   - **Completion**: Drag the slider to the right to dissolve more. In an animation, set this value to 0% in the first frame and 100% in the last.

   - **Size of Pieces**: Drag the slider to the right to increase the size of the pieces into which the object dissolves. This slider controls the size of the pieces the object dissolves into.

Spherical Morph

The Spherical Morph modifier turns the selected object into a sphere.
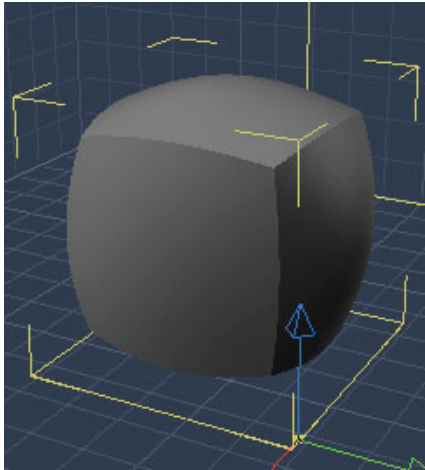
To set Spherical Morph attributes:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab**.

3. Choose **Complex Deformations> Spherical Morph** from the modifiers list pop-up menu.

4. Adjust the following setting in the **Properties tray: Modifier tab: Spherical Morph panel**:

   - **Completion**: Drag the slider to the right to make the object more like a sphere. In an animation, set this value to 0% in the first frame and 100% in the last frame.

Spike

The Spike modifier adds needle-like pointed spikes to the selected object. Use the Spike modifier to add stubble to a character's chin or create a field of grass blowing in the wind.
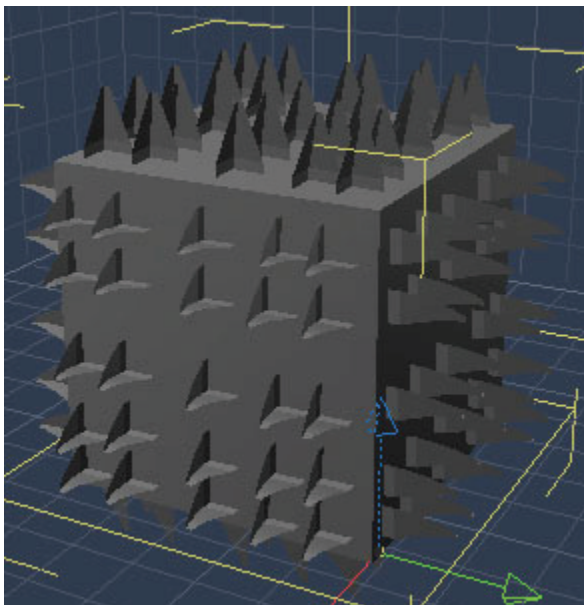
To create Spikes:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab**.

3. Choose **Complex Deformations> Spike** from the modifiers list pop-up menu.

4. Adjust the following options in the **Properties tray: Modifier tab: Spike panel**:

    - **Spike Density**: Drag the slider to the right to apply a greater number of spikes to your object.

    - **Length**: Drag the slider to the right to make the spikes longer.

    - **Radius**: Drag the slider to the right to set a larger radius for the spikes.

    - **Messiness**: Use the Messiness slider to control how wavy the spikes appear. A low setting results in straighter spikes while a high setting results in wavier spikes.

    - **Flow**: Use the Flow slider to control the wiggles of the spikes during animations. This slider works with the Messiness slider. The higher the value, the more the spikes appear to move in an animation.

    - **Gravity**: Use the Gravity slider to control the strength of gravity on the spikes. The higher the value, the more the spike will bend downward.

    - **Keep Original Object**: Enable the Keep Original Object checkbox if you want to keep the original object while adding the spikes around it. Disable it to replace the object with spikes.

Warp

The Warp modifier takes the object's surface points that are farthest from the center of the object and moves them out farther. At the same time, surface points that are nearest the center of the object move in closer to the center.

NOTE! The Warp modifier has no effect on spheres because all the surface points are the same distance from the center.

To set Warp attributes:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab .**

3. Choose **Complex Deformations> Warp** from the modifiers list pop-up menu.

4. Adjust the following option in the **Properties tray: Modifier tab: Spike panel**:

- **Strength**: Use the Strength slider to control the amount of warping. A negative setting moves surface points near the center away from the center and surface points far from the center in toward the center. A positive setting moves surface points far from the center farther away from the center and points near the center closer to the center.

Behavior Modifiers

Behavior modifiers allow you to give objects sets of instructions that give them an "activity" during an animation. Behaviors reduce the number of key frames you need to create and allow you to create actions that are not otherwise possible. The behavior modifiers basically fall into two categories:

- The **Bounce**, **Spin**, and **Shake** behaviors assign complex actions to an object.

- **Point At** and **Track** define or modify the motion of one object based on the location or motion of another object.

NOTE! For information on the IK Terminator and Inverse Kinematics modifiers, see "Inverse Kinematics." (Standard, Pro)

Bounce

The Bounce behavior instructs an object to bounce up and down along the global Z-axis. You can set the upper and lower limits for the bounce, and specify the number of bounces per second.

NOTE! The bounce behavior does not control an object's lateral movement (along the X and Y axes). However, if the object is already moving along the XY plane, it bounces while moving.

The Bounce behavior is useful when you have a specific effect in mind and wish to specify Bounce parameters (like height and frequency) by hand. However, natural bouncing is much easier to create using physics. For details, refer to "Physics (Standard, Pro)."
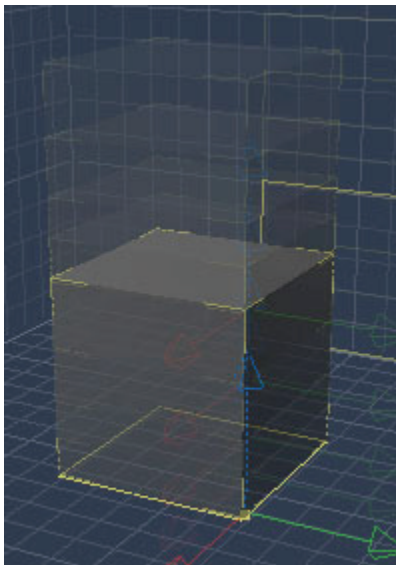
To set bounce options:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab**.

3. Choose **Behaviors> Bounce** from the modifiers list pop-up menu.

4. Adjust the following settings in the Properties tray: Modifier tab: Bounce panel:

    - **Bounce per Second**: Drag the slider to the right to increase the number of bounces per second.

        TIP! For a realistic effect, you can change the number of bounces per second at the beginning and end of your animation.

            - **Floor**: In the text field, enter the lower limit of the bounce. This value is the distance from the global universe Z=0 plane that is the lower limit of bouncing.

            - **Height**: In the field, enter the distance above the floor level that is the upper limit of bouncing.

            - **Start** and **Stop**: Enter values in the Start and Stop fields to limit the bounce to a particular range in time.

Point At

The Point At behavior instructs an object to orient itself "toward" another object through all or part of the animation.

Use the Point At behavior to get a camera or light to follow a particular object or group. You can also use it to animate an object that always faces or points at another, such as a satellite dish as it tracks a satellite, or a sunflower as it follows the path of the sun.
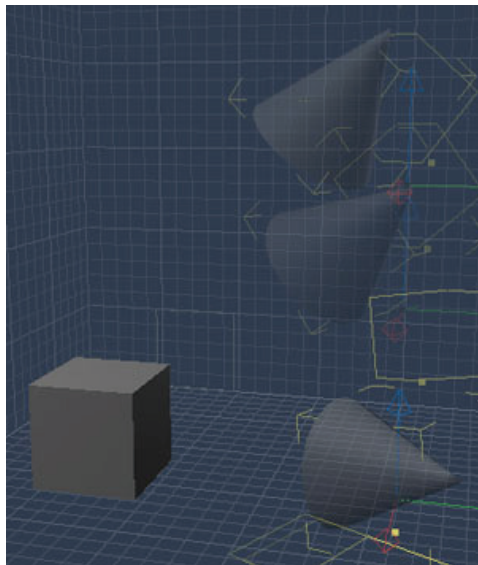
NOTE! Edit >Point At differs from the Point At modifier in two respects. First, **Edit >Point At** only operates on cameras and lights. Second, it only makes the change "right now," so it's not an animation tool.

To set attributes for Point At:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab**.

3. Choose **Behaviors> Point At** from the modifiers list pop-up menu.

4. Set the following options in the **Properties tray: Modifiers tab: Point At panel**:

   - **Towards**: Click the button to open a dialog where you can select the object that should be followed.

   - **Axis**: Select an Axis radio buttons to choose which face of the object should point at the target object.

      NOTE! It is not necessary to set the Axis for a camera or light. By default, the face with the light beam or the camera lens is selected. Any other setting points the light beam or camera lens at 90° or 180° to the target.

      NOTE! For objects other than cameras or lights, you'll need to specify which face of the object should be oriented toward the target. For example, if you're working with a "pointing finger" object, select the axis that describes the direction the finger points.

      - **Enabled**: Check this option to turn the behavior on and off at specific points in an animation. A key frame is created each time you turn the behavior on or off.

Shake

The Shake behavior instructs an object to jump in an animation. You can set the number of jumps per second, as well as the amplitude of the jumps.
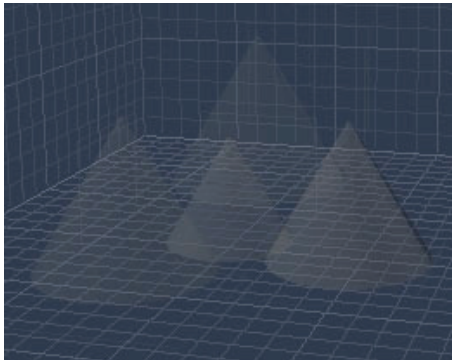
To set Shake attributes:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab**.

3. Choose **Behaviors> Shake** from the modifiers list pop-up menu.

4. Set the following options in the **Properties tray: Modifiers tab: Shake panel**:

    - **Jumps per second**: Drag the slider to the right to increase the number of times an object jumps in a second.

    - **Amplitude**: Enter the length of the jumps.

Spin

The Spin behavior instructs an object to rotate on one of its axes. You can set the number of cycles of the spin per second.

NOTE! To make the object spin in the opposite direction, enter a negative value.

To set Spin attributes:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab**.

3. Choose **Behaviors> Spin** from the modifiers list pop-up menu.

4. Set the following options in the **Properties tray: Modifiers tab: Spin panel**:

    - **Cycles per second**: Drag the slider to the right to increase the number of times the object spins in a second. (A cycle is one complete 360° rotation.)

    - **Axis**: Select the axis of rotation.

    - **Start** and **Stop:** Enter values in the fields to limit the spin to a specific time range.

Track

The Track behavior instructs an object to "shadow" another as its moves through the universe.
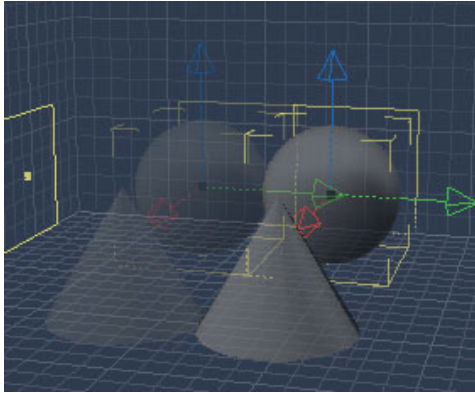
To set Track attributes:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab**.

3. Choose **Behaviors> Track** from the modifiers list pop-up menu.

4. Set the following options in the **Properties tray: Modifiers tab: Track panel**:

   - **Towards**: Click to open a dialog where you can select the object that should be tracked.

   - **In**: Enable the checkboxes for each axis on which you want the tracking behavior to occur.

   - **Track Rotation**: Check this option if you want the object to track the rotation as well as position.

   - **Enabled**: Check the box to toggle the behavior on and off at specific points in the animation. A key frame is created each time you turn the modifier on or off.

Waves

Carrara includes several types of Waves modifiers so you can add waves to objects.

Circular Wave

The Circular Wave modifier distorts an object with a wave effect that ripples the surface in a circular fashion.

To set Circular Wave attributes:

1.   Select the object you want to modify.

2.   Click **Properties tray: Modifiers tab**.

3.   Choose **Waves> Circular Wave** from the modifiers list pop-up menu.

4.   Set the following options in the **Properties tray: Modifiers tab: Circular Wave panel**:

-   **Height of Waves**: Drag the slider to the right to create a higher wave.

-   **Frequency**: Use the slider to control the number of waves that occur within a given time interval.

-   **Phase**: Use the slider to control the position of the wave within a given time interval.

-   **Limits:** Use the sliders to control the portion of the object on which the wave is applied. The left control sets the bounding box for one side of the axis; the right control sets the bounding box limit for the other side. For example, if the slider is set at 25% and 75%, the wave starts at 25% from the bottom of the object and continues to the upper 75% of the object. At 0% and 100%, the entire object is modified.

-   **Wave Function**: Select the 1 - cos or sin radio button to toggle between a cosine and sine wave pattern.

-   **Axis**: Select the radio button for the axis on which you want the wave to occur.

Linear Wave

The Linear Wave modifier distorts an object using a wave effect that ripples the surface in a uniform fashion.

To set Linear Wave attributes:

1.   Select the object you want to modify.

2.   Click **Properties tray: Modifiers tab**.

3.   Choose **Waves> Linear Wave** from the modifiers list pop-up menu.

4.   Set the following options in the **Properties tray: Modifiers tab: Linear Wave panel**:

-   **Height of Waves**: Drag the slider to the right to increase the height of the waves.

-   **Frequency**: Use the slider to control the number of waves that occur within a given time interval.

-   **Phase**: Use the slider to control the position of the wave within a given time interval.

- **Limits**: Use the slider to control the portion of the object on which the wave is applied. The left control sets the bounding box for one side of the axis; the right control sets the bounding box limit for the other side. For example, if the slider is set at 20% and 80%, the wave starts at 20% from the bottom of the object and continues to the upper 80% of the object. At 0% and 100%, the entire object is modified.

- **Wave Function**: Select the 1 - cos or sin radio button to toggle between a cosine and sine wave pattern.

- **Axis**: Select the radio button for the Axis where you want the wave to occur.

   NOTE! You cannot enable the checkbox for the selected Axis. For example, if the Axis X radio button is selected, you cannot enable the In X checkbox.

- **In**: Enable the checkboxes for each axis on which you want the wave to occur.

Wave

The Wave modifier distorts the object by pulling it along an imaginary wavy sweep path. Use this modifier to simulate air or water currents.
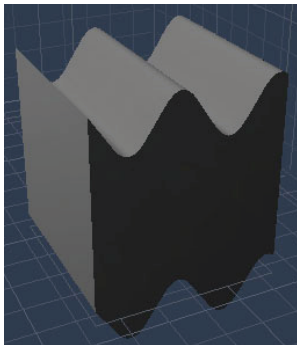
To set Wave attributes:

1. Select the object you want to modify.

2. Click **Properties tray: Modifiers tab**.

3. Choose **Waves> Wave** from the modifiers list pop-up menu.

4. If you want to use Direct Manipulation, click its icon in the title bar until the **On** icon appears. A bounding box with a number of control handles appears around the object. Drag the **Number of Waves** slider to the right to increase the number of waves applied to the object.
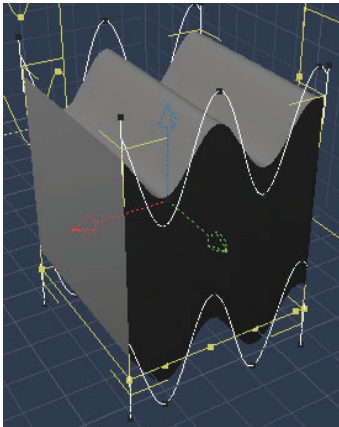
5. Drag one of the corner handles to adjust the height of the wave. Dragging away from the object increases the wave's height. Or, use the **Height of Waves** slider to set the wave height. Dragging to the right increases the wave height.

6. Drag the center handles, or use the **Phase** slider to control the wave position within a given time interval.

7. Select the radio button for the **Axis** where you want the wave to occur.

8. In the Shape section, choose a shape for your wave. The available shapes are:

- **PlanarA**: Wave moves along the plane A.
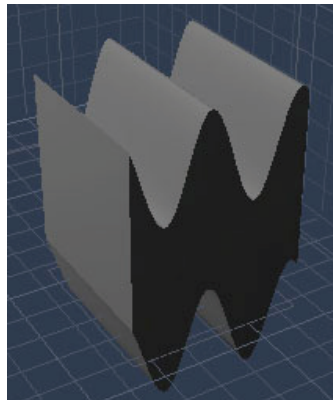
- **PlanarB**: Wave moves along the plane B.
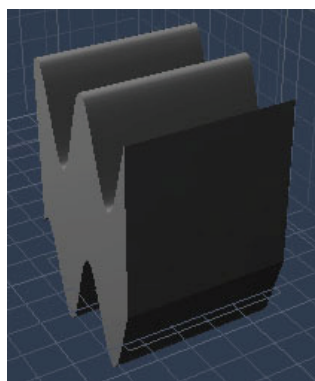
- **Radial**: Wave moves from center point outward.
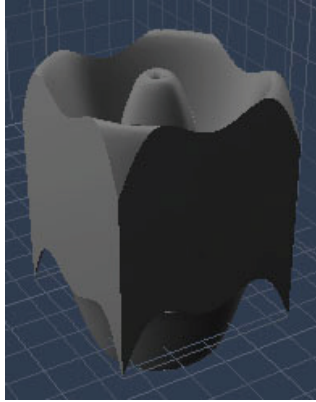
- **Cylindrical**: Wave moves around the outside of the object as if it were in a cylinder.
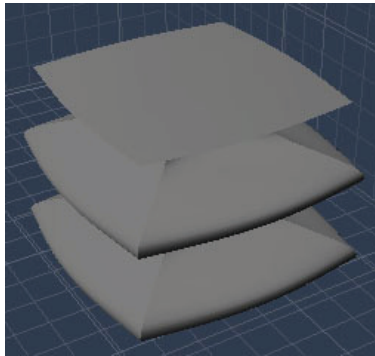
The direction in which the wave is applied depends on which Axis radio button is selected.

Smart Pack

The Smart Pack modifiers are covered in the section that follows. There are two modifiers in this category:

Morpher

Morpher allows one object to morph into another. The only restriction is that targets must have the same number of vertices as the object being morphed. See "Using Morph Targets," for the preferred way of creating morph targets (Standard, Pro).

To use the Morpher:

1. Create your base shape, using the Spline, Metaball, or Vertex modeler.

2. If necessary, convert your object to a vertex object and open it in the Vertex modeler.

3. If the polygons are not triangulated, select all of the polygons in your model and triangulate the polygons with the **Model > Triangulate Polygon** command.

4. Return to the Assembly room. Choose Edit > Duplicate to create a copy of the object.

5. Select the duplicate and switch back to the Vertex modeler. When Carrara asks if you want to create a new master or edit the master, choose **Create new master**.

6. Modify the geometry but **do not add any vertices**. Repeat steps 2 - 5 as many times as desired; there is no limit on the amount of morphs you can apply.

7. In the Assembly room, select the object to be morphed, and the morph targets you created. Choose **Edit > Group** to group the object to be morphed and the targets into one group.

8. Scale your targets down to 1% and move them out 10000 in any direction. Do not hide your object! An error will be generated if the targets are set to invisible.

9. Add a morpher modifier to your base object. Enter the name of the target in the target field. If the object was found and has the right number of vertices, the name you typed in the field will remain, otherwise 'Error' or 'Not Found' will appear.

10. Use the morph slider to change the degree of the effect. Negative values also work.

If you edit one of your targets, and it doesn't seem to modify your base object, just hit the 'reload now' checkbox on that morph.

Noise Deformer

This deformer adds noise to the geometry of an object. It can be used to simulate dilapidation and decay. This example illustrates the effect:
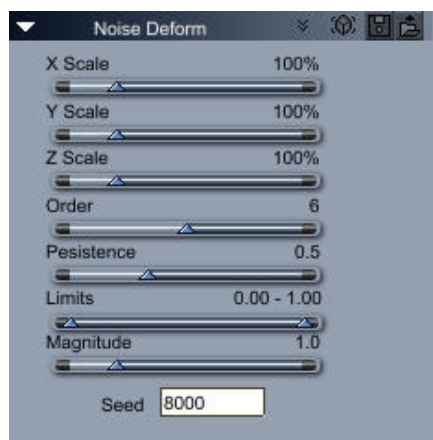
These settings will deform an object.
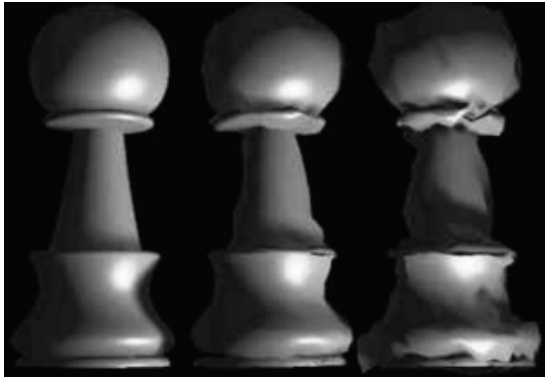
Set the following options:

- **Seed** This value "seeds" the random number generator.
- **Scaling** Scales the noise each direction.
- **Order and Persistence** Please refer to the introduction to noise present in the section covering Noise Factory. See "Noise Factory."
- **Limits** Upper and Lower limits on the Magnitude of the noise.
- **Magnitude** Changes the magnitude of the disturbance.

Using Formulas in Carrara

Carrara allows you to create mathematical formulas that in turn create custom features. Using formulas, you can define your own objects, deformers, tweeners, shaders, gels, backgrounds, and backdrops. The interface for creating formulas is common for all items. The valid input and output variables differ for each item you create. Formula editing becomes technical quickly. Higher math or programming experience is recommended.

Creating Formula Objects

Formula objects are 3D objects you create using mathematical equations. The Formula Editor lets you enter parameters, variables, and operators to create unique objects.

To create a formula object:

1. Do one of the following:

    - To create a formula object of default size at the center of the universe, choose **Insert > Formula**. (Standard, Pro)

    - Click the Formula tool and drag it into the scene window or Hierarchy section of the Sequencer.

2. Double-click the object. The Formula controls appear.
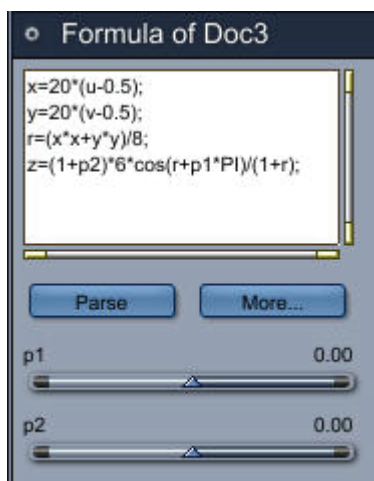
3. Set the Formula controls to create an object of the shape you want. Enter a formula into the editing panel on the top, or edit the existing one.

- **Parse** tests and compiles the formula. If there are any errors, Carrara gives you an error report.

- **More** displays the Formula Editor.
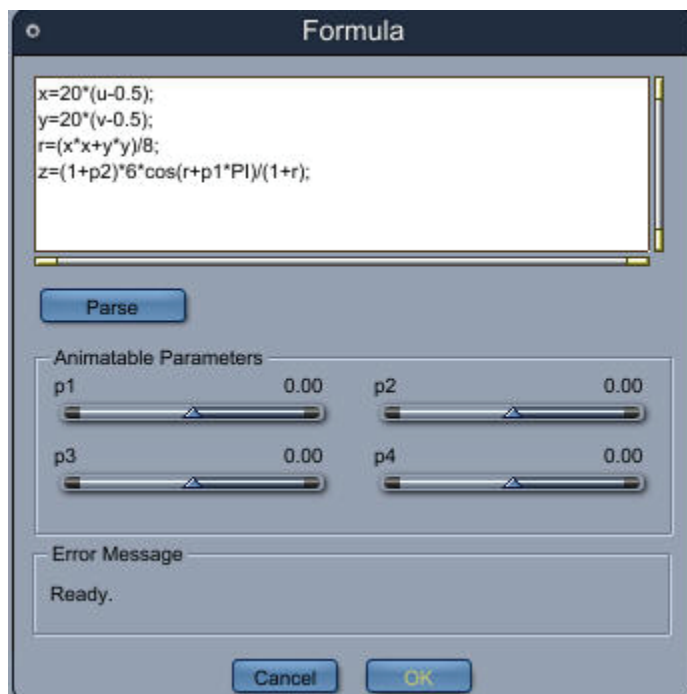
4. When you are finished, return to the Assemble or Storyboard room.
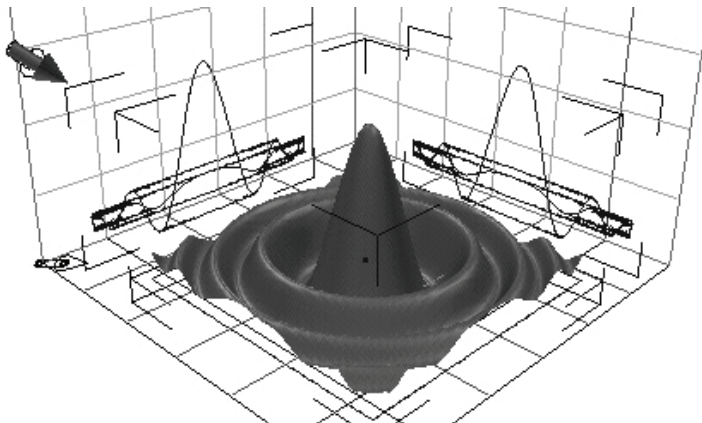
5. When your formula object is finished you can edit it by adjusting its parameters.

Creating Formula Items

The method for creating a formula item depends on the item you want to create. You can create the following items using formulas:

- Primitive objects
- Deformers
- Tweeners
- Shaders
- Gels
- Backgrounds and Backdrops

To create a Formula primitive:

1. Drag the Formula Primitive icon into the scene, or choose **Insert > Formula**.

2. Double-click the object to display the formula controls, or click the Model room button.

To create a Formula deformer:

1. Click the **Properties tray: Modifiers tab**.

2. Choose **Formula** from the pop-up. The formula controls appear in the Modifiers section.

To create Formula tweeners:

1. Double-click the gap between key events on the time track. Carrara opens the Transition Options dialog.

2. Choose Formula from the pop-up. The formula controls appear in the Transition Options dialog.

To create Formula shaders:

1. Display the Shader Tree Editor by double-clicking a shader in the Browser.

2. Select the shader tree component where you want to use a formula.

3. Select **Pattern Functions > Formula** from the list. The formula controls appear on the selected branch of the shader tree.

To create Formula gels:

1. Select a light other than Distant light.

2. Display the **Properties tray: Effects tab: Gel pane l.**

3. Choose Formula from the list. The formula controls appear in the dialog.

To create Formula backgrounds or backdrops:

1. In the Properties tray, select the scene, then click the Effects tab.

2. Select Background or Backdrop.

3. From the (Background or Backdrop) pop-up, choose Formula. The formula controls appear in the Effects section in the Background or Backdrop controls.

Formula Controls

The formula controls include a readout of the formula itself, sliders for adjusting parameters, a Parse button, and a More button.
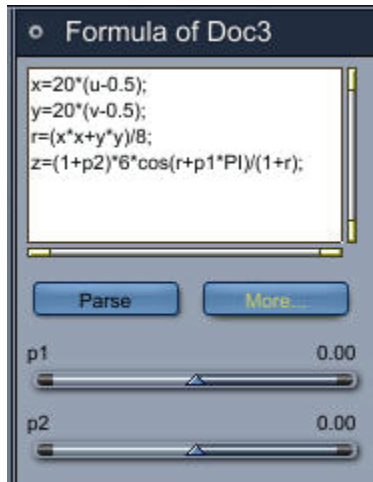
For a few items, the controls also include a preview of sorts. (For the tweener formula, the preview is a graph.)
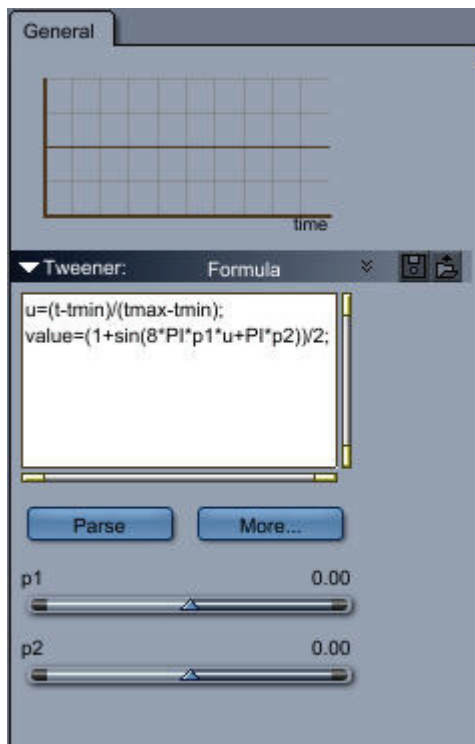
Editing Formulas

To create or edit a formula:

1. In the formula controls, click More. Carrara opens the Formula Editor.

    o You can type into the window to build your formula. Be careful, the editor is case-sensitive.

    o Each statement must end with a semicolon.

2. When you are finished in the Formula Editor, click OK.

3. In the formula controls, click the Parse button. Parsing compiles the formula to run.

4. If appropriate to this formula, you can adjust the parameter sliders.

Formula Operators

sin(x)	sine of x
cos(x)	cosine of x
tan(x)	tangent of x
asin(x)	arcsine of x
acos(x)	arccosine of x
atan(x)	arctangent of x
atan2(y,x)	arctangent of y/x
sinh(x)	hyperbolic sine of x

cosh(x)   hyperbolic cosine of x

tanh(x)   hyperbolic tangent of x

sqrt(x)   square root of x

pow(x,y)   x raised to the power of y

exp(x)   exponential

log(x)   logarithm

log10(x)   base-10 logarithm

floor(x)   floor of x (The largest integer that is less than or equal to x)

ceil(x)   ceiling of x (The smallest integer that is greater than or equal to x)

abs(x)   absolute value of x

mod(x,y)   **x** modulus y

Comparison Operators

<   less than

<=   less than or equal to

==   equal to

!=   not equal to

>   greater than

>=   greater than or equal to

&&   logical "and"

|| logical "or"

Conditional

variable=((comparison)?(value if TRUE):(value if FALSE))

For example:

a=((b>1)?1:b);

The expression (b>1) is the comparison. If it's true, the formula returns a=1. If it's false, the formula returns a=b.

Constants

PI

E

Animatable Input Parameters

p1, p2, p3, p4

These parameters can be set between -1 and +1.

Variables

The variables valid for a particular item depend on its type. Refer to the section below that describes the variables for the formula item you are creating. To define a variable, just use it. Its initial value will be 0.

Formula Shader

Output Variables

value (output is a grey level) or red, green, blue (output is a color)

The expected output is between 0 (black) and 1 (white or color maximum intensity).

You can use any of these output variables, however, the program can't use value' and any of red', green', and blue' simultaneously. If the shader is used in a color based channel, then color output variables will have priority (meaning if value' and red' are defined, then only red' will be used). The opposite applies when using the shader in a value based channel, like Shininess. (If value' is defined, value' will be used).

Input Variables

u and vu and **vu and v**	These are parametric mapping coordinates.
x, **y** and **z**	These are the local space coordinates of the point being shaded.
nx, **ny** and **nz**	These are the normal local coordinates (normalized vector).
X, **Y** and **Z**	These are the global space coordinates of the point being shaded.
NX, **NY** and **NZ**	These are the normal global coordinates (normalized vector).

Formula Deformer

The Formula Deformer iterates on all the points of an object, deforming them one by one.

Output Variables

dx, **dy** and **dz** The local coordinates of the deformed point. The range is -1 to +1 (0 is the center.)

Input Variables

x, y, z The local coordinates of the point to be deformed.

Formula Tweener

Output Variables

value

Input Variables

t, tmin, tmax t is time. One t unit is 1/1800th of a second. t is between tmin and tmax.

Formula Backdrop

Output Variables

value or red, green, blue

Refer to "[Formula Shader](#)."

Input Variables

u,v (u,v) defines a point on the backdrop. The range is between 0 and 1. (0,0) is the lower left corner, (1,1) the upper right.

Formula Background

Output Variables

value or red, green, blue

Refer to "[Formula Shader](#)."

Input Variables

x,y,z (x,y,z) is the global coordinates of the direction vector, from the camera eye to the background. The vector (x,y,z) is normalized and each coordinate ranges between -1 and +1.

Formula Gel

Output Variables

value or red, green, blue

Refer to "[Formula Shader](#)."

Input Variables

u,v (u,v) defines a point on the backdrop. The range is between 0 and 1. (0,0) is the lower left corner, (1,1) the upper right.

Formula Primitive (Object)

Output Variables

x,y,z local coordinates of the point defined by (u,v).

u,v The parametric space coordinates. The range is between 0 and 1.

Using Scene Effects

Carrara provides some environment options to help complete the appearance of your scene in renderings. These options include Ambient Light, Atmospheres, Backgrounds, and Backdrops.

You'll find the scene effects option in the **Properties tray: Effects tab** after you select Scene from the Instances tab or from the Sequencer tray.
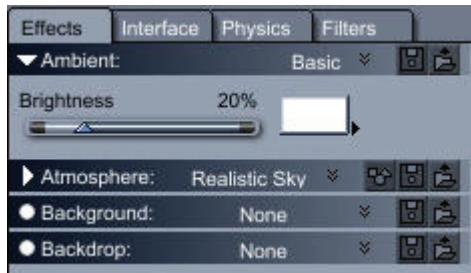
To apply a scene effect:

1. Click the working box to deselect any selected objects, or select Scene from the Properties tray: Instances tab.

2. Click the **Properties tray: Effects tab**.

3. Select the effect you want: Ambient, Atmosphere, Background, or Backdrop.

4. Set the options for the effect in the panel that drops down.

NOTE! The results of Scene Effects settings are visible in the final rendering or when you render an area with the Test Render tool. You can return to these settings, make changes, and render again.

Ambient

Ambient light is uniform throughout the scene. It radiates in every direction, having no specific origin and casting no shadows.

Ambient light is useful for scene building, but you will want to turn it down or even off for rendering, since it tends to reduce the feeling of depth by washing out shadows and contrast. Instead, concentrated light sources (from specific lights) bring out shadows and highlights and emphasize your subject's form and surface features.

TIP! A properly placed distant light is actually more effective than ambient light for simulating a daylight scene.

Ambient light characteristics are scene settings. Choose color and brightness to achieve the effect you want. Remember, a strong ambient light level washes out the effects of your specific lights.

TIP! Use a warm color to simulate indoor light and a cool color to simulate outdoor light.

To set ambient light:

1.   Display the **Properties tray: Effects tab: Ambient panel**.

2.   Click the color chip and select a color from the color picker.
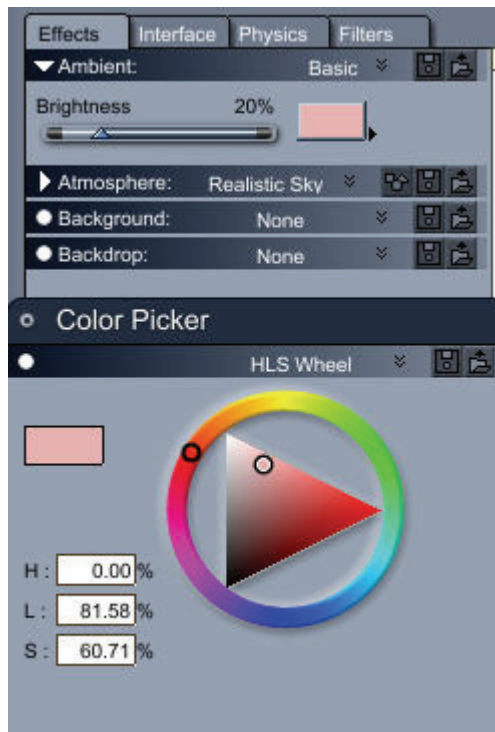
3.   Enter a percentage for the brightness.

4.   To view your changes, use the Test Render tool.

Atmosphere

Atmosphere

Carrara offers two fog effects:Distance Fog and Cloudy Fog:and two different simulations of sky :Sky and Realistic Sky.
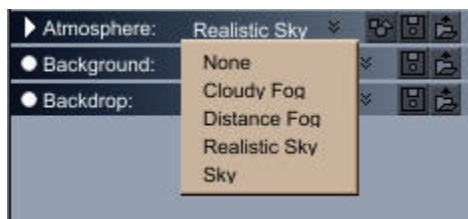
NOTE! Choose None if you do not want your scene to use atmosphere effects. Choose one of the other options to add fog or sky atmospheric effects to your scene.

Cloudy> Fog

Cloudy Fog produces an effect that simulates the irregular clumping of clouds.

To use Cloudy Fog:

1. In the **Properties tray**, choose **Scene** from the Instances tab.

2. In the **Properties tray: Effects tab: Atmosphere panel**, choose **Cloudy Fog**. The available settings appear.
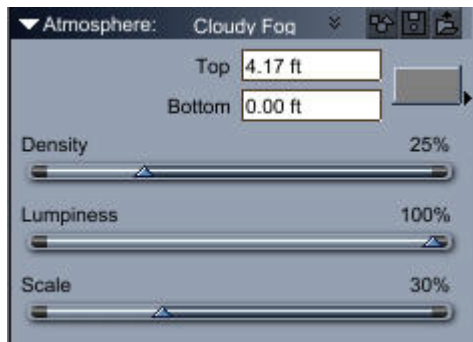
3. Set the Cloudy Fog settings:

   - **Top** and **Bottom** define the vertical range (along the Z axis), where fog appears. When you are setting this option keep in mind that altitude Z=0 runs through the center of the working box when the working box is in its default position.

   - **Density** determines fog thickness.

   - **Lumpiness** determines how much the fog clumps together.

   - **Global Scale** determines the spacing between the wisps of fog.

4. Click the color chip to set its color. For information on Carrara's color picker, refer to "Color Pickers."

Distance Fog

Distance Fog produces an effect like a haze, which is more apparent across greater distances. This fog has uniform density.

To use Distance Fog:

1. In the **Properties tray**, choose Scene from the Instances tab.

2. In the **Properties tray: Effects tab: Atmosphere panel**, choose **Distance Fog**.
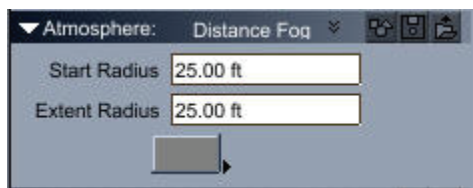
3. Set the Distance Fog settings:

   - **Start Radius** sets the distance from the camera to where the fog begins.

- **Extent Radius** sets the distance beyond the start of the fog where it becomes so thick that visibility becomes zero.

4. Click the color chip to set its color. For information on Carrara's color picker, refer to "Color Pickers."

Realistic Sky

Realistic Sky enhances the simulation offered by Sky, using an even more accurate model to simulate the optical phenomena created by the atmosphere. You have probably noticed that the color of an object changes slightly with its distance from the viewer (e.g. remote mountains appear bluish). Realistic Sky enables you to create this natural effect (called aerial perspective) on your scene. Additionally, the physical model involved is used to render the sky and the clouds, which makes them more realistic than in previous versions of Carrara.

To use Realistic Sky:

1. In the **Properties tray**, choose Scene from the Instances tab.

2. In the **Properties tray: Effects tab: Atmosphere panel**, choose **Realistic Sky**.
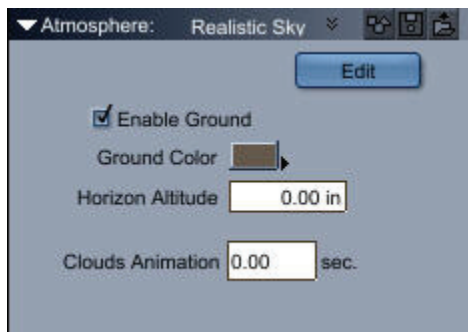
3. Adjust the general settings of your Realistic Sky:

- **Enable Ground**: If you don't yet have a terrain in your scene, enable this checkbox to display an infinite plane as the Ground.

- **Ground Color**: Click the color swatch to set the color of the ground plane.

- **Horizon Altitude**: Set the Horizon Altitude in relation to the working box. Cameras located below the horizon altitude won't see much.

- **Clouds Animation**: Controls the Clouds Animation by giving each keyframe an elapsed time value.

- Click the **Edit** button to open the Atmosphere Editor, described next.

For further information on Realistic Sky, see "Using the Realistic Sky Atmosphere Editor."

Using the Realistic Sky Atmosphere Editor

When you click the Edit button with Realistic Sky selected in the Properties tray: Effects tab: Atmosphere panel, the Atmosphere Editor displays various properties for realistic sky. These settings are similar to those used for the Sky editor discussed in "Using the Sky Atmosphere Editor."
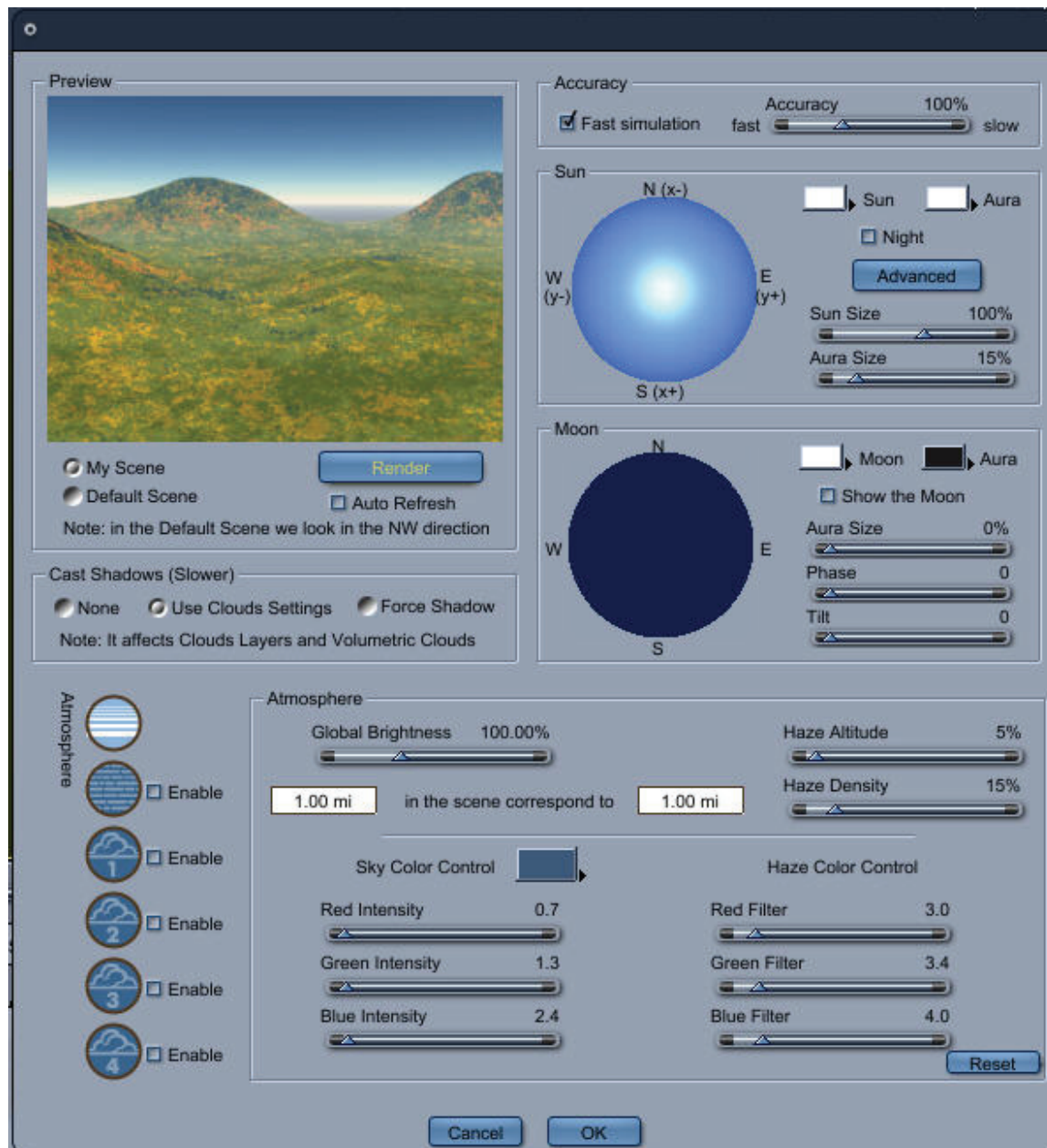
The Atmosphere Editor contains:

- The Preview section in the top left corner.

- The Sun Editor and the Moon Editor in the top right corner.

- The Atmospheric effects section in the bottom half.

The following sections will guide you through the fast and simple process of creating a realistic sky in Carrara. The first step is to configure the Preview Renderer options.

Configuring the Realistic Sky Preview

The Preview renderer is located in the upper-left corner of the Realistic Sky editor.

To render the preview:

1. Select the scene that you want to use in the preview renderer:

    - Choose **My Scene** to use your current scene in the preview renderer.

    - Choose **Default Scene** to use a default grassy plain for your preview.

2. Click the **Render** button, on the right side. The preview shows the Realistic Sky using the current settings from the editor.

3. If you want changes to any parameter to automatically refresh the preview, then check **Auto Refresh**. If not, you will have to click **Render** each time you make a change.
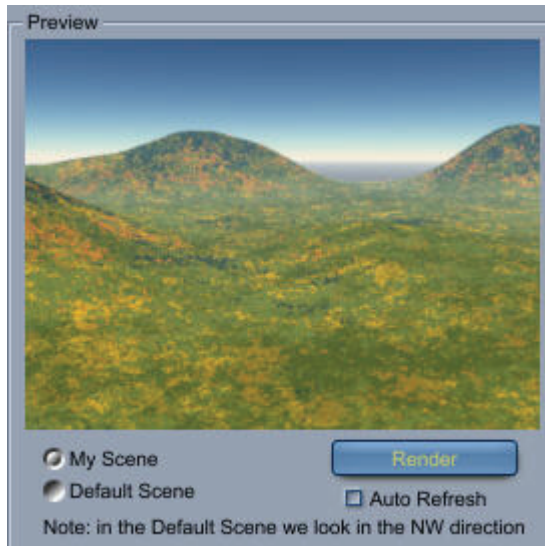
Setting the Aerial Perspective

If you are using your own scene in the Preview renderer, you must first specify the size of your world to get the correct **aerial perspective** effect on your scene

To get the correct aerial perspective:

- Indicate, in the lower Atmosphere section, the scaling factor of your scene. Give an estimated correspondence between the distances in the scene and what they stand for in reality.

- You can tweak these values to get more or less aerial perspective effect on your scene.
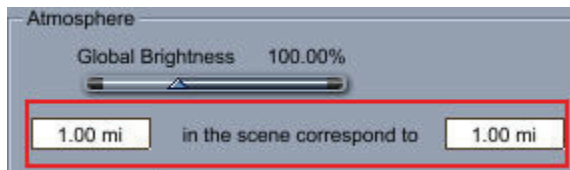
NOTE! This scaling factor also takes into account the cameras. Thus, a big scale can make them go out of the atmosphere.

Putting the Sun in Place

Now that you have visualized the default settings and applied the desired aerial perspective, use the Sun section to adjust the Sun settings:
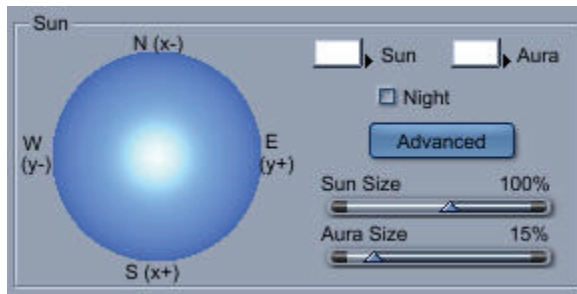
To put the Sun in place:

1. Click the trackball and drag in a direction. The correspondence between the cardinal points and the working box orientation helps you position the sun in your scene.

2. Set the following additional properties as desired:

   - **Sun**: Click the color swatch to set the color of the Sun.

   - **Aura**: Click the color swatch to set the color of the aura.

   - **Night**: Check this option if you want to put the sun behind the Earth.

   - **Sun Size**: Move the slider to set the size of the Sun.

   - **Aura Size**: Move the slider to set the size of the aura.

3. For more accurate settings, click the **Advanced** button to open the Sun properties dialog.
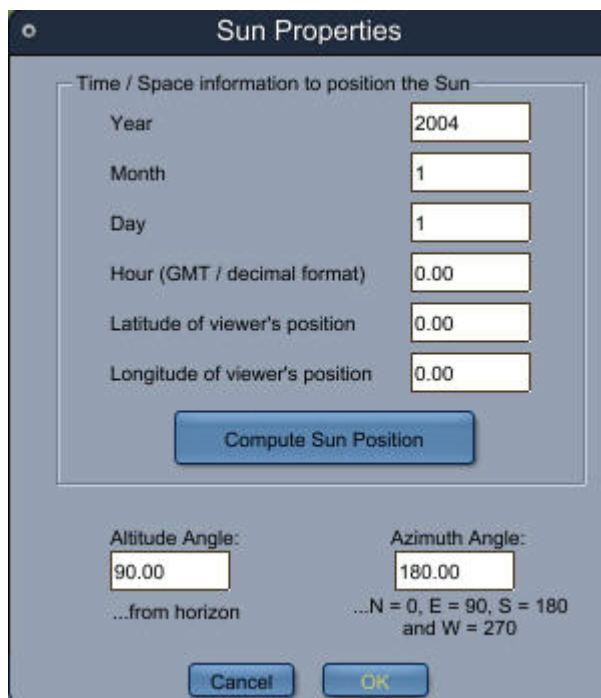
4. Set the following properties in the Sun Properties dialog as desired:

   - **Year**: Enter the desired year that you want to represent.

   - **Month**: Enter the desired month that you want to represent.

   - **Day**: Enter the desired day that you want to represent.

- **Hour (GMT/Decimal Format)**:Enter the desired time, in decimal format, that you want to represent. For example, 6:45 PM would be 18.75. If entering time for a location other than GMT, you'll need to know the number of hours to offset by (also allowing for daylight savings time). For example, California time is 7 hours behind GMT on September 10, so you enter 11.75 to correspond with 18.75 GMT. For further information, see www.greenwichmeantime.com.

- **Latitude of Viewer's Position**: from -90 in the South Pole to 90 in the North Pole.

- **Longitude of Viewer's Position**: from -180 to 180, negative (positive) in the West (East) of Greenwich.

- **Compute Sun Position**: Click this button to calculate the position of the sun based on the above values.

- **Altitude Angle**: The altitude from the horizon based on the computations. You can also enter a value manually.

- **Azimuth Angle**: The azimuth (clockwise movement) of the sun based on the computations. You can also enter a value manually.

Putting the Moon in Place

The Moon editor is located directly beneath the Sun editor, and has similar settings as those found in the basic Sun editor.
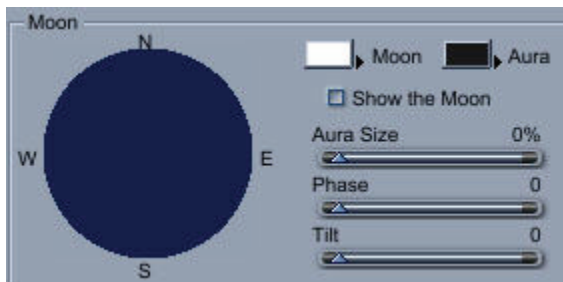
To put the Moon in place:

1. Check the **Show the Moon** option to enable the moon controls.

2. Click the trackball and drag in a direction. The correspondence between the cardinal points and the working box orientation helps you position the moon in your scene.

3. Set the other properties as needed:

   - **Moon**: Click the color swatch to set the color of the moon.

   - **Aura**: Click the color swatch to set the color of the aura.

   - **Aura Size**: Adjust the slider to set the size of the aura.

   - **Phase**: Adjust the slider to adjust the phase of the moon, ranging from new moon to full moon.

   - **Tilt**: Adjust the slider to adjust the tilt of the moon.

Modifying the Atmosphere

You may want to change the visual impact of the atmosphere on your scene. For this purpose, you have several options in the Atmosphere Layer, which is displayed by clicking the Atmosphere icon (the top one) on the left side.
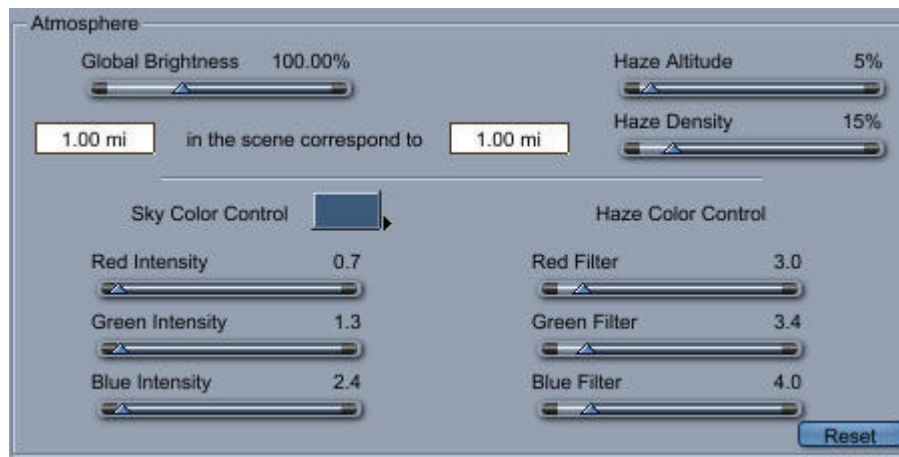
To adjust the luminosity:

- Move the **Global Exposure** slider to adjust the luminosity of the scene.

To modify the sky color

The sky color mainly depends on the effect of the air molecules on the light and the default color is calculated for our planet's air. If you want to modify the sky color, you can either:

- Move the **Red**, **Green** and **Blue Intensities** sliders, or;

- Pick up a color by clicking the **Color** swatch, which will automatically calculate the associated Red, Green and Blue Intensities.

NOTE! The Sky simulation is based on a physical model and the default Red, Green and Blue Intensities stand for measured values of the air properties.

To set the haze:

The haze lies at the bottom of the atmosphere and adds a visual effect that doesn't exist on the top of the atmosphere. Control this effect with the Atmosphere Layer.
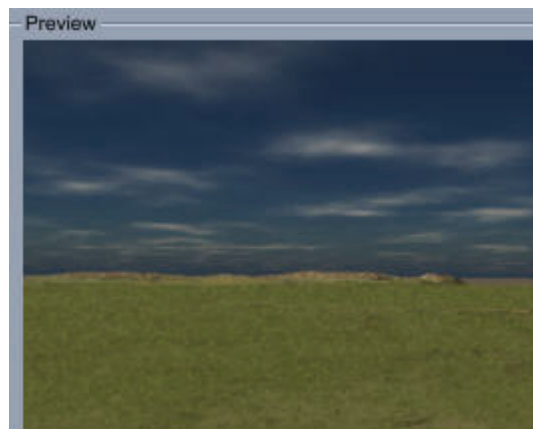
1. If you want the haze more dense where it lies, increase the **Haze Density** slider.

2. If you want the haze to be more present in your sky, increase the **Haze Altitude** slider.

3. You can modify the haze color by using the **Red**, **Green** and **Blue Filters** sliders: the more a color is filtered by the haze, the less it is present in the horizon (but still present up in the sky).

Adding Some Fog

While you can change the color of the sky at the horizon thanks to the haze settings, you may also want to add some fog that affects your objects in the scene. For that purpose, click the Fog icon (the second one) on the left side, which shows the Fog Editor. To be able to use it, check Enable.
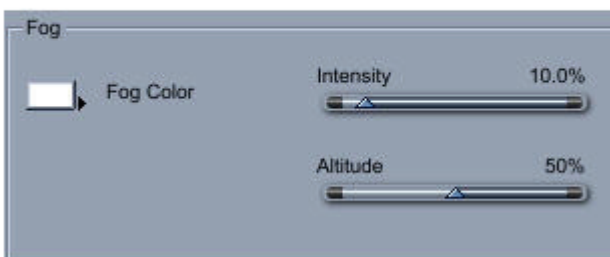
To set the fog:

1. The fog will appear in the region between the horizon altitude (the ground) and an altitude that you control with the **Altitude** slider.

2. If you want your fog to be more opaque, increase the **Intensity** slider.

3. Finally, apply a **Fog Color** by clicking the color swatch.

Adding Some Clouds

Realistic Sky provides you with four independent cloud layers. To use a cloud layer, make sure you have its **Enable** option checked. To modify the settings of a cloud layer, click its icon on the left side, which shows the Cloud Layer Editor.
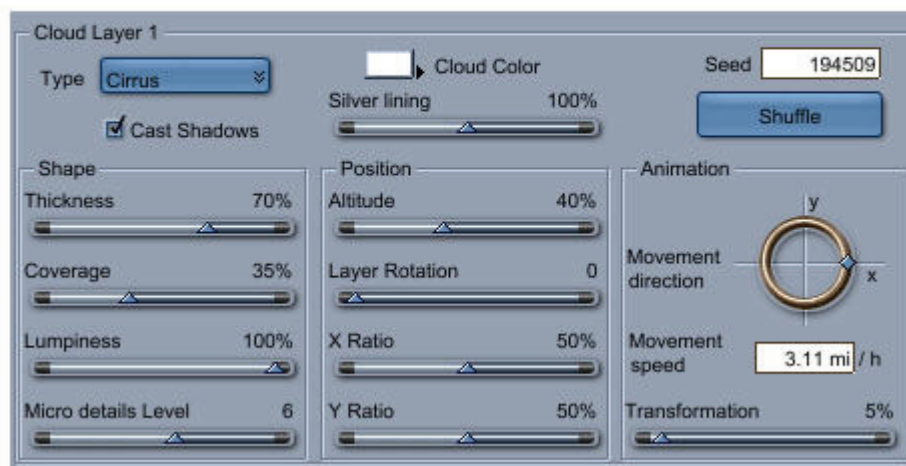
To shape the clouds:

1. If you want big thick clouds, then you can move up the **Thickness** and **Coverage** sliders. If you want thin Cirrus-like clouds, you must have a small **Thickness** value.

2. If you want your clouds to be more uniform, you can decrease the **Lumpiness** value.

3. Control the amount of intricate details of a cloud with the **Micro Details Level** slider.

4. Apply scaling factors to the clouds by moving the **X Ratio** and **Y Ratio** sliders.

5. If you want to change the clouds' distribution, click **Shuffle** to compute a brand new one.

To position the clouds

1. Increase the **Altitude** slider if you want this layer's clouds to appear higher in the sky.

2. To adjust the clouds' position to your scene, you can rotate the layer around the z axis with the **Layer Rotation** slider.

To animate the clouds

1. Set the **Movement Direction** of this layer's clouds by moving the circular slider. If you want them to move in the East (y axis) direction, move the cursor up to 90 degrees from the South (x axis).

2. Set the **Movement Speed** of this layer's clouds by typing in the distance they traverse in one hour.

   NOTE! Type in a distance in a certain unit and Carrara automatically converts it in miles if you prefer Imperial units or kilometers if you prefer metric units.

3. If you want the clouds' shape to evolve quickly over time, move up the **Transformation** slider.

Realistic Sky Tips and Tricks

To speed up your workflow

1. Unless you want to check the effect of Realistic Sky on the objects located under the **Horizon Altitude**, you can enable the **Ground** option to quickly render this region.

2. **Clouds** are mainly responsible for the computation time of Realistic Sky. You may want to start by setting the other features without any cloud layers and then add the clouds.

3. Check **Auto Refresh** if you want to immediately visualize each of your changes.

To get beautiful (realistic) skies

1. In the **Insert** menu, choose a **Sun Light** or a **Moon Light** to lighting your scene accordingly to the Sun's or Moon's position.

2. The **Red**, **Green** and **Blue Filters** sliders for the haze should have about the same values. It comes from the fact that the light scattering (and absorption) processed by the haze is almost independent of the wavelength. Increasing their values makes the haze thicker.

3. The **Red**, **Green** and **Blue Intensities** sliders for the sky color stand for the amount of light scattered by air molecules. Since these are measured physical values, you should not change them too much if you want your sky to keep looking terrestrial. The Reset Button allows you to restore the original settings.

4. If you want a beautiful sunset, set the Sun near the horizon and see how realistic the sky color looks! Add clouds and see how their lighting is nicely affected by the Sun.

TIP! A very hazy sunset has a big bright halo around the Sun. Reproduce this effect by increasing the Haze Density.

5. If you want overcast weather, you can use several cloud layers, with high values for **Thickness** and especially the **Coverage** parameters. Use the **Layer Rotation** and **Altitude** sliders to have the different layers superimposed as you like. Decrease the **Global Exposure** of the Atmosphere Editor.

6. You may observe some artifacts in your sky if you use some unusual settings. To correct them, increase the **Accuracy** slider, on the top right corner of the Atmosphere Editor.

7. Combine Realistic Sky with other environment effects:

    - Volumetric Clouds (refer to "Volumetric Clouds"). (Standard, Pro)

    - Backgrounds and Backdrops: You can load a map as a background and apply Realistic Sky on it. See "Backgrounds and Backdrops."

To create a nice animation

1. Every parameter can be modified from a keyframe to another to create an animation: the Sun's position, the fog intensity, the clouds' color and so on.

2. If you want your clouds to change direction during the animation, create a keyframe with a different value for the **Movement Direction**. You will see the clouds turning in the sky.

You can control the speed of your clouds animation by attributing each keyframe a number of run out seconds. This way, you can choose to simply follow the Time Line or have a time control (and not necessarily linear) dedicated to the clouds.

Sky

Sky simulates the interaction between light and the earth's atmosphere. Based on a physical model of a real atmosphere, Sky creates a realistic surrounding for objects in your scene.

With the Sky effect, you can create various haze, fog, rainbows, cloud, sun and moon effects.

To use Sky:

1. In the **Properties tray**, choose Scene from the Instances tab.

2.   In the **Properties tray: Effects tab: Atmosphere panel**, choose **Sky**.
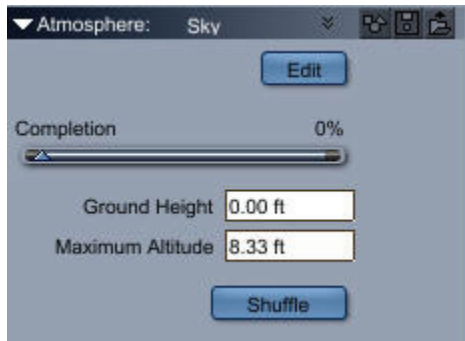
3.   Adjust the Sky settings as appropriate:

- **Ground Height** defines the altitude of the ground in your scene.

- **Maximum Altitude** defines the overall size of the Sky atmosphere. This value scales the atmosphere in every direction. For example, if Maximum Altitude is set to 200", the haze, fog and cloud layers altitudes can be set from 0" (the ground) to 200". Generally, you should set the Maximum Altitude to a value greater than the overall size of your scene.

  For special effects, you can set an Maximum Altitude that is smaller than your scene. This makes cloud layers appear below the highest object in a scene, simulating the effect of an airplane or mountain going through the clouds. The atmosphere always starts at altitude zero (z=0). Objects below z=0 are invisible.

  NOTE! The camera must be at an altitude greater than zero. If your camera is below zero, you'll see the atmosphere from below the ground, which leads to interesting but unpredictable results.

- **Shuffle** resets the random shape of the clouds.

- **Completion** defines when clouds start to move and when they stop. Use this slider in an animation. Typically, you would set the time to 0% at the first frame of your animation and 100% at the last frame.

4.   To quickly and easily set up an atmosphere, choose a setting from the Use Preset pop-up menu. Carrara automatically sets all the parameters for you. Or, to control each effect, click the Edit button. The Atmosphere Editor appears.

For further information about Sky, see "Using the Sky Atmosphere Editor."

Using the Sky Atmosphere Editor

To access the Atmosphere Editor, in the **Sky** workspace, click the Edit button. The Atmosphere Editor appears.
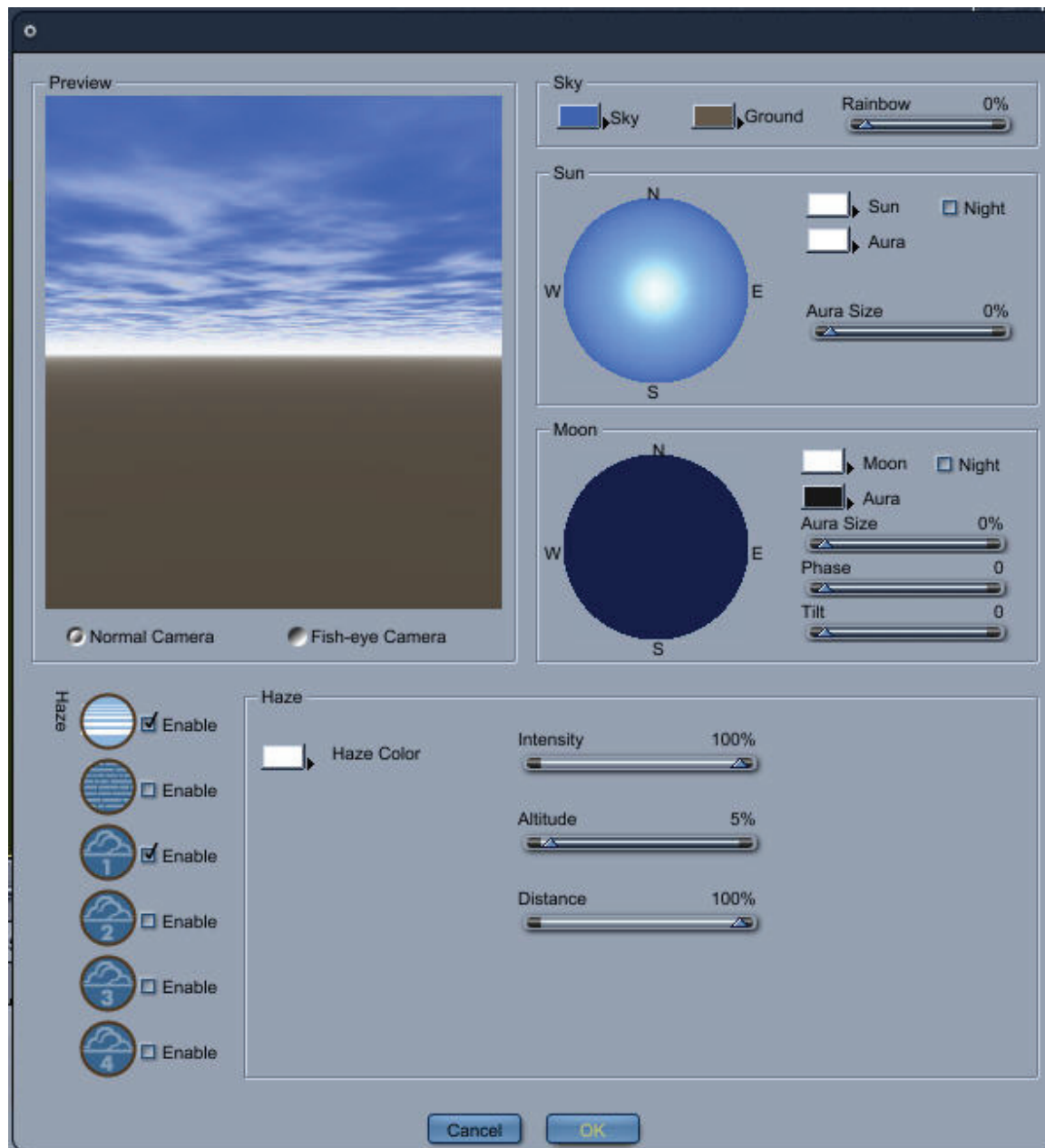
The Atmosphere Editor contains:

- Preview section

- Sky Editor

- Sun Editor

- Moon Editor

- Layer Manager

- Layer Editor

The Preview Section

The Preview section renders the atmosphere based on the current settings. Each time you modify a parameter, the preview updates to show the result. What you see in the Preview is exactly what you get in your scene.

The Preview has two display modes:

- Standard Camera Mode

- Fish-Eye Camera Mode

Standard Camera Mode gives you a view of the atmosphere using a conical camera looking to the West.
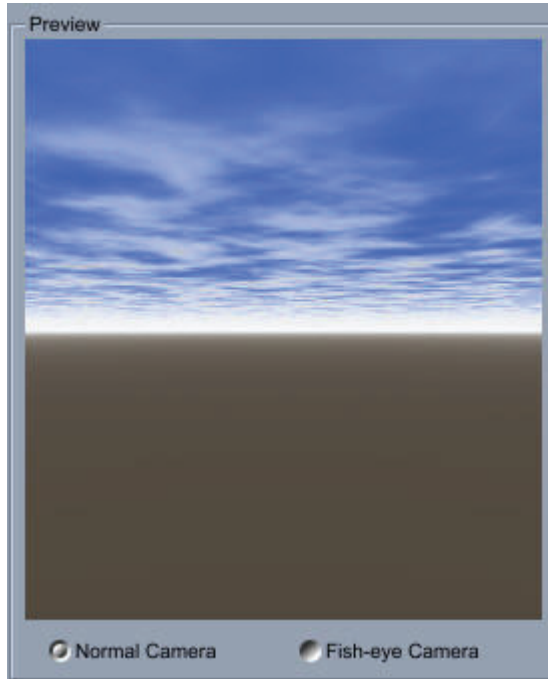
Standard Camera Mode only renders a small area of the atmosphere but it is perfect for setting up a sunrise or a sunset.

To simulate Standard Camera Mode in your final rendering:

1. Set the rendering camera to Front position.

2. In the Camera Palette, set the lens type to Conical.

3. Raise the camera (Z=8 in).

Fish-Eye Camera Mode renders the whole atmosphere as viewed from the ground through a fish-eye lens. You can use this mode to preview the entire sky, not just the western horizon.
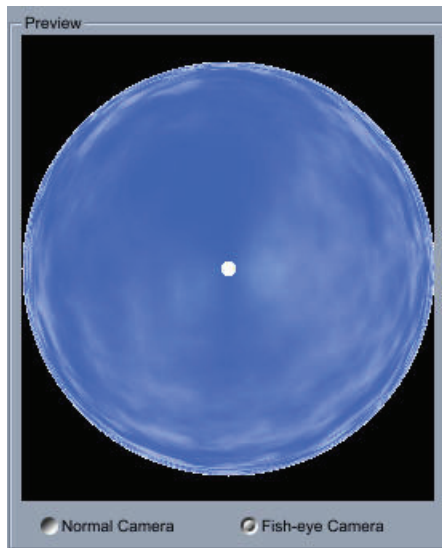
The Sky Editor

The Sky Editor lets you change the color of the sky and the ground. You can also add or remove rainbows.
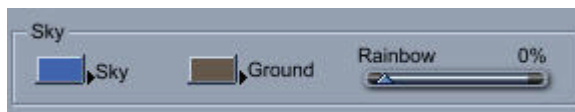
To set the sky or ground color:

1. Click the Sky or Ground color Swatch. You can also click the arrow next to the swatch.

2. Choose the desired color.

3. The Rainbow slider controls the intensity of the rainbow in your atmosphere. By default, the rainbow intensity is 0% (invisible). Dragging the Rainbow slider to the right to change the intensity of the rainbow.

The rainbow rendering is made of two parts: the primary rainbow and the secondary rainbow. The intensity of the primary rainbow is about ten times greater that the intensity of the secondary one. By adjusting the intensity setting you can make the secondary and primary rainbows appear or disappear.

 NOTE! The Sky rainbow reacts just like a real one. To see a rainbow, you must carefully move the sun behind the camera.

The Sun Editor

The Sun Editor lets you change the position, color, aura (or halo) color and size of the sun.

The virtual trackball at the center of the Sun editor lets you set the position of the sun.
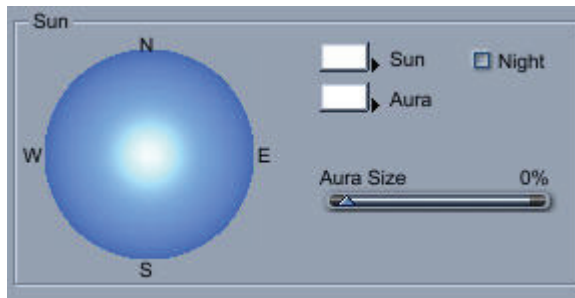
To set the sun's position:

1. Click anywhere on the trackball and drag in the direction you want to move the sun.

2. When the sun is in the desired position, release the mouse button. The sun's position and sky color are updated automatically.

Sky uses the convention illustrated below to map geographic directions to the Carrara perspective window. To predict where the sun will appear, keep the convention in mind.

TIP! To see the sun in the Dynamic Preview, drag it to the West.

To switch between day and night:

*   Check the Night check box in the upper right corner of the editor.

This moves the sun from one side of the Earth to the other. When the Night check box is selected, the Sun is not visible in the atmosphere (since it is on the other side of the Earth) and the sky darkens.

To set the sun's color:

1. Click the arrow next to the Sun color swatch to open a color palette. Or, click the swatch to open a Color Picker.

2. Select the desired color.

To adjust the sun's aura size:

1. Use the Aura control to adjust the size of the sun's aura.

2. Drag the Aura control to the right to increase the aura or halo size, or to the left to decrease it.

To select a color for the sun's aura:

1. Click the arrow next to the Aura color swatch to open a color palette. Or, click the swatch to open a Color Picker.

2. Select the desired color.

The Moon Editor

The Moon Editor lets you change the position, color, phase, aura (or halo color), and size of the moon.
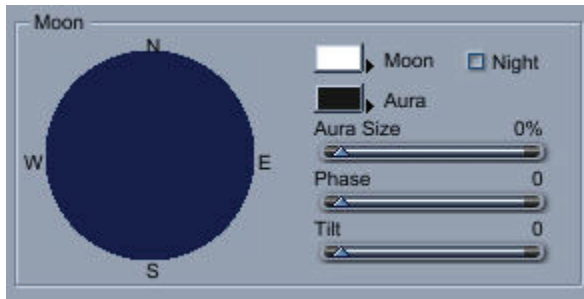
The virtual track ball at the center of the Moon editor lets you set the position of the moon.

To set the moon's position:

1.   Click anywhere on the trackball and drag in the direction you want to move the moon.

2.   When the moon is in the desired position, release the mouse button.

The moon's position and sky color are updated automatically.

Sky uses the convention illustrated below to map geographic directions to the Carrara perspective window. To predict where the moon is going to appear, keep the orientation of the directions in mind.

TIP! To see the moon in the Dynamic Preview, drag it to the West.

To switch between day and night:

•   Click the Night check box in the upper right corner of the editor.

The Night check box moves the moon from one side of the Earth to the other.

To set the moon's color:

1.   Click the arrow next to the Moon color swatch to open a color palette. Or, click the swatch to open a Color Picker.

2.   Select the desired color.

Use the Aura control to adjust the size of the moon's aura or halo.

To adjust the moon's aura size:

•   Drag the Aura control to the right to increase the halo size, or to the left to decrease it.

To select a color for the moon's aura:

1.   Click the arrow next to the Aura color swatch to open a color palette. Or, click the swatch to open a Color Picker.

2.   Select the desired color.

To change the moon's tilt and phase:

•   Click the Moon Phase and Tilt selectors and drag each one in the direction you want to tilt the moon.

The Layer Manager

The Layer Manager lets you edit the layers in your atmosphere and display or hide them.
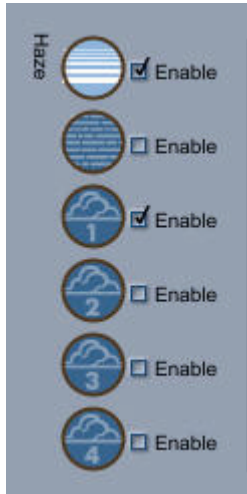
To display or hide a layer:

- Click the Enable check box next to the layer. When the layer is displayed, there is a check in the Enable check box.

To edit a layer:

- Click the layer's icon. The settings for this layer are displayed in the Layer Editor.

NOTE! When you edit a layer, you must display it to see the result in the dynamic preview.

The Layer Editor

The Layer Editor lets you edit the various types of layers you can add to an atmosphere. Its appearance varies, depending on which layer is selected for editing.

The Layer Editor types are:

- Haze layer
- Fog layer
- Cloud layer

Haze Layer Editor

Haze is a light fog that appears due to the humidity found in the atmosphere.
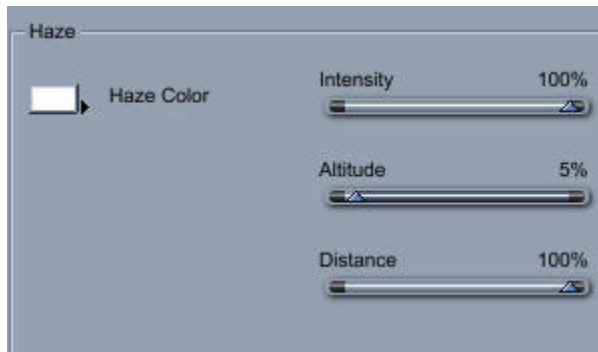
To edit the haze color:

1. Click the arrow next to the color swatch in the upper left corner of the Layer editor to open a color palette, then choose the desired color. Or, click the color swatch to open the Color Picker.

2. Adjust the sliders to the appropriate settings.

   - **Intensity** sets the intensity of the haze.

   - **Altitude** sets the altitude of the haze. The value set in this slider is the height of the ground.

   - **Distance** sets the distance at which the haze begins to be visible.

The altitude and distance percentages are based on the Maximum Altitude. For example, if you set the Maximum Altitude to 200", the haze layer altitude can be set from 0" (0%) to 200" (100%).

Fog Layer Editor

The Fog Layer editor lets you set the color, intensity, altitude, and distance of fog in your atmosphere.
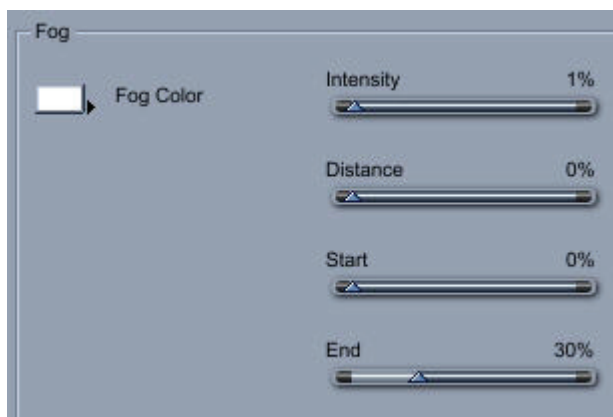
To edit the fog color:

1. Click the arrow next to the color swatch in the upper left corner of the Layer editor to open a color palette, then choose the desired color. Or, click the color swatch to open the Color Picker.

2. Adjust the sliders to the appropriate settings.

   - **Intensity** sets the intensity of the fog.

   - **Start** and **End** define the altitudes where the fog starts and stops.

   - **Distance** defines the distance at which the fog begins to be visible.

The distance, start, and end sliders use percentages based on the Maximum Altitude. For example, if you set the Maximum Altitude to 200", the fog layer altitudes can be set from 0" (0%) to 200" (100%).

Cloud Layer Editor

Your atmosphere can have up to four layers of clouds. The Cloud Layer editor lets you set the color, size, altitude, and density for each cloud layer. When adjusting these settings, keep in mind how the cloud layers you use will interact. You can see this interaction in the Preview.

To edit the cloud color:

1. Click the arrow next to the Clouds color swatch to open a color palette. Or, click the color swatch to open the Color Picker.

2. Choose the desired color.

The Cloud Layer editor provides seven graphic cloud controls for adjusting the density and altitude of clouds in the layer.
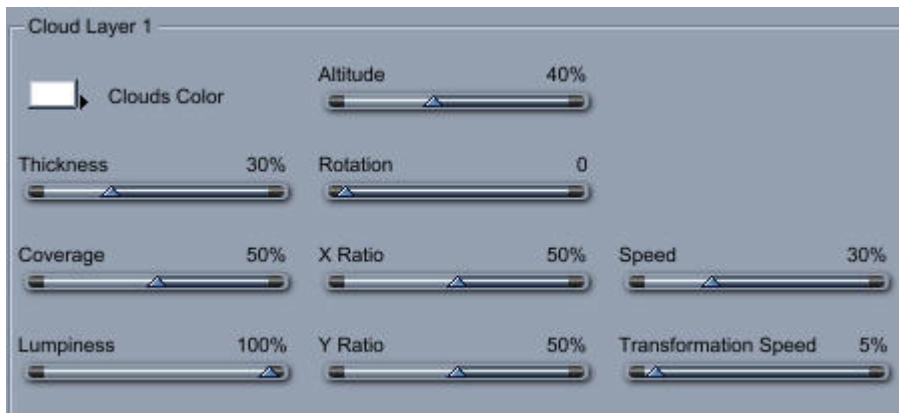
To adjust a cloud control setting:

Use the three first sliders to set the cloud's contrast, thickness and coverage. Together these settings determine the quality of the weather.

- The Thickness control lets you set the cloud thickness from almost transparent clouds to thick clouds in a heavy storm.

  TIP! The cloud thickness setting is the most important of the settings because it controls the density of the clouds and this affects how light interacts with them. Like in the real world, thin clouds scatter light and thick clouds absorb light. To create beautiful sunsets, drag the sun near the horizon and add several layers of thin clouds.

- The Coverage control determines the cloud coverage. Use this control to choose between a blue sky and a completely overcast sky.

- The Lumpiness control sets the cloud's contrast.

The next four sliders are used to change geometry of the cloud layer.

- The Rotation slider affects not only the appearance of clouds, but the direction they move in an animation as well.

- The X and Y ratio controls control the size of the clouds in the layer.

- The Altitude control controls the altitude of the cloud layer.

NOTE! By default, Cloud Layers are arranged with Layer 1 set to the lowest altitude and Layer 4 at the highest.

Cloud Animation Sliders

The two sliders on the right let you control the animation of the cloud layer. Use the first slider to set the speed of the clouds. If you do not want the clouds to move, set the slider to 0%.

Use the second slider to control the transformation of the clouds over time. At 0% the shape of the clouds remains the same.

Rendering the Sky

After you're satisfied with the settings in the Atmosphere Editor, click OK. Render the image as usual. You'll soon see the atmosphere surrounding your objects.

Animating Sky

The Sky effects can be animated. Just by dragging the sun, you can create a photo-realistic sunset. Carrara handles everything for you, including the reddening of the sun as it approaches the horizon, moving clouds, darkening skies, and scattering light through the clouds. You can even modify the atmosphere parameters to render alien planets.

To use Sky in an animation:

1. Create your animation as usual.

2. With the Current Time Bar at 0:00:00, select Scene from the Properties tray.

3. Select **Properties tray: Effects tab: Atmosphere panel: Sky**.

4. Edit the Sky parameters.

5. Set the Completion slider to 0%.

6. Jump to the end of your animation.

7. Set the Completion slider to 100%.

8. Render your animation as usual.

You can also use various Sky parameters over the course of an animation. To do this, apply different parameters to different key frames. For example, you could make the sky seem to become overcast over time by setting the Sky parameters in the first frame to create a clear day, and then creating a key frame and adjusting the Cloud density parameters to create a cloudy day. When animated, the sky appears to become overcast. You can animate any of the Sky parameters in this way.

Backgrounds and Backdrops

When you render your scene, regions where there are no objects are considered background areas.

Carrara offers two features for filling in the space between objects:background and backdrop.

- **Backgrounds**: The background gives you control over the scene environment. The background is sometimes called an environment map. During rendering, the background is projected on the inside surface of a giant sphere that surrounds your scene. Any region of a rendering not occupied by an object will show the background. The background is also the environment reflected by reflective objects. There are several types of backgrounds available, which are described in the following sections.

![Carrara logo] **NOTE!** An isometric camera cannot record a Background. If you are using an isometric camera for your final rendering, use a Backdrop instead.

- **Backdrops:** A backdrop is a 2D plane placed behind your scene, and that stays perpendicular to the camera. The backdrop appears in regions of the rendering where there are no objects. The backdrop does not appear in reflections or interact with the lighting.

Tip You may want to use both a backdrop and a background. In this case, Carrara uses the background for the reflection environment and places the backdrop behind the objects.

There are several options for backgrounds and backdrops, covered in the sections that follow.

None

By default, or by selecting None, these areas appear black in the renderings.

Bi Gradient Controls

A Bigradient Background or Backdrop creates a paired gradation for the background. This is an easy way to create a sky and horizon.

To use a bigradient for the background or backdrop:

1. In either the **Properties tray: Effects tab: Background** or **Backdrop panel**, choose **Bigradient**.

2. Set the Bigradient settings as you like.

3. Select colors for Start Sky, End Sky, Start Ground, and End Ground.

4. Use the Horizon slider to set the height of the horizon.

Color Controls

Color sets a solid color as the background.

To use color background or backdrop:

1. Choose either the **Properties tray: Effects tab: Background or Backdrop panel: Color.**

2. Adjust the color in the color chip.

Formula Controls

The Formula control creates a color pattern from a formula. You can devise your own formula to create a new pattern.

The formulas for a Background or Backdrop differ. The Background is projected onto a sphere, so it uses a 3D system. The Backdrop is rectangular, so it uses a 2D system.

To select a formula for the background or backdrop:

1. In either the **Background panel: Properties tray: Effects tab: Background or Backdrop panel**, choose **Formula**.

2. Double-click the object. The Formula controls appear.
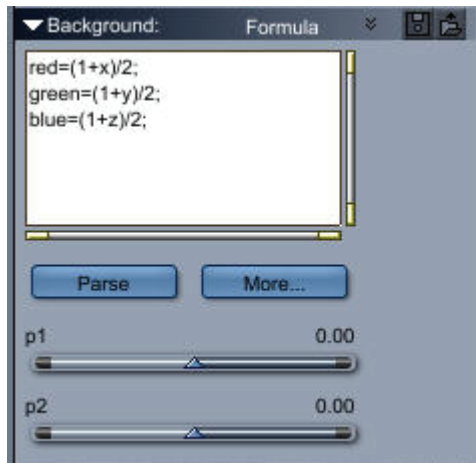
3. Enter a formula into the editing panel on the top, or edit the existing one.

4. Set the Formula controls to create an object of the shape you want:

    *   **Parse** tests and compiles the formula. If there are any errors, Carrara gives you an error report.

    *   **Animatable Parameters** increase and decrease the value of parameters.

- **More** displays the Formula Editor.
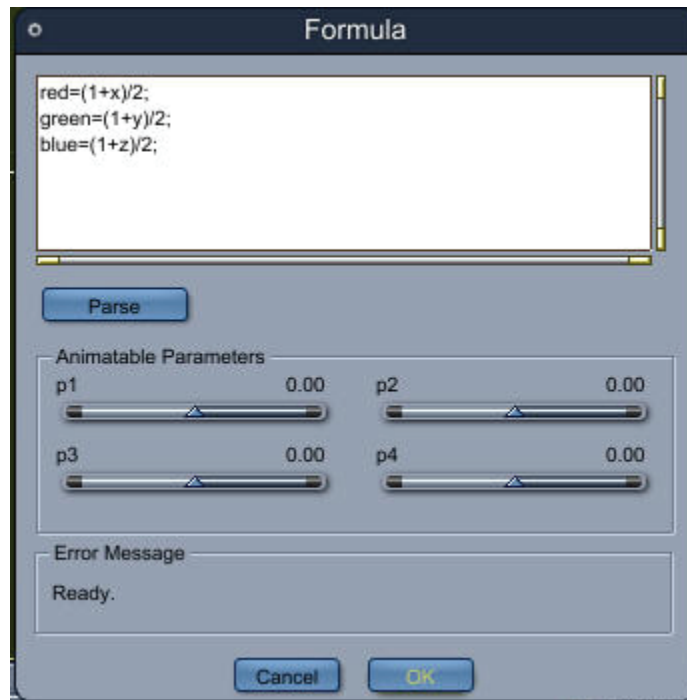
For more information on creating Formula objects, refer to "Using Formulas in Carrara."

HDRI Controls

The HDRI background uses a High Dynamic Range Image (HDRI) as a background (Note that there is no HDRI backdrop). Using the HDRI background is especially useful if you want to create realistic reflections on your objects or simply create complex lighting (using the sky light option of the renderer).

What is HDRI? A HDR image (.hdr file) is a special file format that stores an image with a lot more precision than a regular bitmap format (such a JPEG). In the real world, the intensity of the light varies a lot more than on a regular photo. You can see this when taking pictures of scenes lit by the sun: the areas in the shadows appear black whereas the areas under the sun appear almost white. An HDR image will contain details in both the bright areas and in the shadows.

What type of map is supported? There are several types of HDRI maps (cube maps, sphere maps....). At this point, only one type is supported: simple spherical maps. They are easy to recognize: if you look at the image the contains the map, it should completely fill the rectangle of the image.
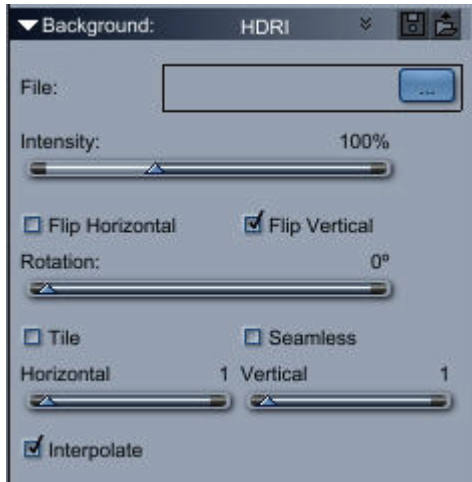
To use a HDR image for the background:

1. In the **Properties tray: Effects tab: Background panel**, choose HDRI.

2. Click on the button to open the file dialog and select a.hdr file to use as your background.

3. Adjust the following settings:

   - **Intensity**: Increase or decrease the intensity of the map.

   - Flip Horizontally or Vertically

   - **Rotation**: Rotates the map

   - **Tile**: Repeats the image. (You can use the Horizontal and Vertical sliders to select the number of repetitions).

   - **Seamless**: Creates tiling where the map is flipped so that the side of each tile is continuous.

   - **Interpolation**: Use to create a smoother map

Map Controls

The tools and methods for using a map are the same for the background or backdrop.

Map uses a bit-mapped image file as the background. This is the best choice when you want specific images in reflections.

Ideally, images for the background should have an aspect ratio of 2:1 (twice as wide as they are tall). Images for the backdrop should have the same aspect ratio as the final rendering and be equal or greater in resolution.

To select a map for the background or backdrop:

1. In either the **Properties tray: Effects tab: Background or Backdrop panel**, choose **Map**.

2. Click the folder icon to locate and open a bitmapped image file. When you've opened an image, a preview appears in the right panel.
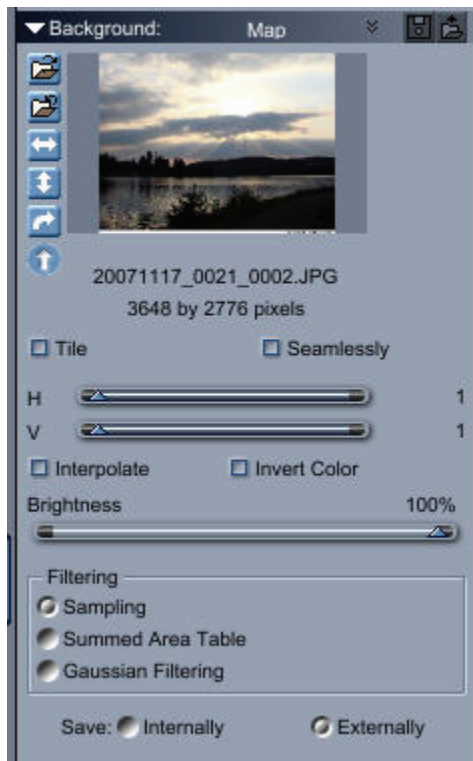
3. Set the Map controls:

    -   The directional buttons under the folder icon change the image's orientation.

    -   **Tile** causes the image to repeat. The **Horizontally** and **Vertically** sliders set the number of tiles in each direction.

    -   **Seamlessly** tiles the image without visible seams between tiles. Carrara flips adjacent tiles vertically and/or horizontally as necessary to avoid seams.

    -   **Invert Color** inverts the image's colors.

    -   **Interpolate** gets cleaner results in your rendering.

    -   **Brightness** slider increases and decreases the image's brightness.

    -   **Filtering** determines the quality of the render:

        -   **Sampling** produces the fastest but lowest quality render.

        -   **Summed Area Table** produces a better quality render, but uses more memory. You may want to avoid it for large texture maps.

        -   **Gaussian Filtering** produces the best quality render.

            TIP! You can apply a filter to the map. For more information, refer to "Rendering Effects."

4. **Save Internally** - TBD

5. **Save Externally** - TBD

Movie Controls

You can use a sequence of images in the background of your scene. Use movies only when you create an animation.
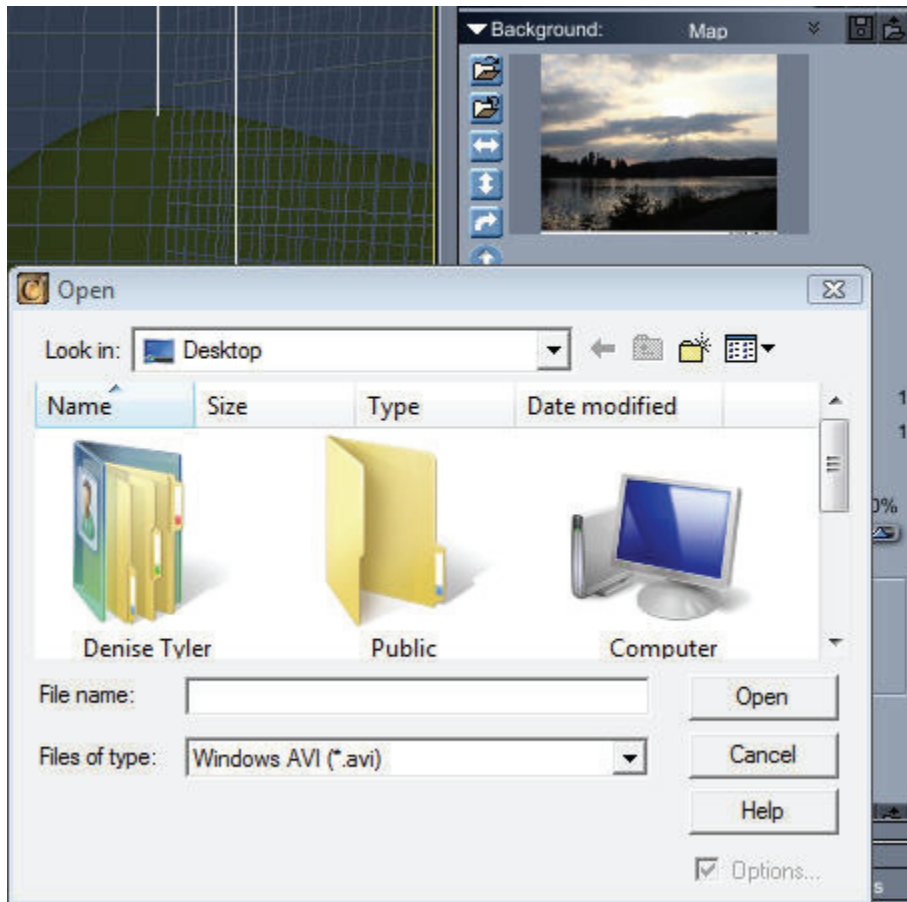
To select a movie for the background or backdrop:

1. In either the **Properties tray: Effects tab: Background or Backdrop panel**, choose **Map**.

2. Click the folder icon to locate and open an AVI or QuickTime movie file. When you open a move file, a preview appears, with a preview player below it

3. Set the Movie controls as you like:

    -   The directional buttons change the image's orientation.

    -   **Tile** causes the image to repeat. The **Horizontally** and **Vertically** sliders set the number of tiles in each direction.

    -   **Seamlessly** tiles the image without visible seams between tiles. Carrara flips adjacent tiles vertically and/or horizontally as necessary to avoid seams.

    -   **Invert Color** inverts the image's colors.

    -   **Interpolate** gets cleaner results in your rendering.

    -   **Brightness** slider increases and decreases the image's brightness.

    -   **Filtering** determines the quality of the render:

- **Sampling** produces the fastest but lowest quality render.

- **Summed Area Table** produces a better quality render, but uses more memory. You may want to avoid it for large texture maps.

- **Gaussian Filtering** produces the best quality render.

Flash Controls

TBD

Casting Shadows on Background Objects

The Shadow Catcher feature allows you to generate true shadows for 3D objects, so they appear on background imagery in a realistic way.

The surface you apply to the Shadow Catcher object provides the color and transparency level of the shadow. A Shadow Catcher object is rendered only in the areas where it is in shadow. All other parts of the object display the background color or image. Objects behind a Shadow Catcher object are not visible.

NOTE! Shadow Catcher objects work with Ray Tracing or Hybrid Ray Tracing rendering only. In all other rendering modes, they appear as normal objects.

To cast a shadow on a background:

1. Create a simple model of the background object. Use the background image to align the objects and the camera's point of view. In the following example, we want to cast the shadow of the saucer on a cityscape background image.
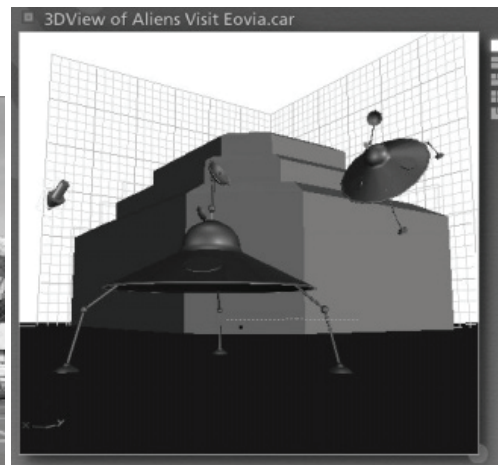
2. Select the object or objects to which you want to apply the Shadow Catcher.

3. Position the object that is casting the shadow over the Shadow Catcher object.

4. Go to the texture room and change the top shader to **Shadow Catcher** (in the Lighting Models category).

5. Disable the Cast Shadows checkbox. Otherwise, Shadow Catcher objects that cast shadows on other objects can create an undesirable effect.

6. Render the scene. During rendering, the background becomes visible, as though the image itself is shadowed.

Adding Sound to Your Scene

Carrara provides tools for adding a sound track to animations you create. The sound track can have any number of individual channels, each with sounds placed at times you choose. When Carrara renders an animation, it saves the sound track to the final movie file.

Sound can be integrated in your workflow in two ways. First, you can synchronize sound effects to events in your animation. For example, if you have an explosion in your animation, you can add an explosion sound effect and place it in the sound track at the appropriate time.

Second, you can create a sound track for your animation in an audio production package, and then add the sound track to your animation so that you can synchronize the events in the animation to the sound track.

Creating Animations With Sound

When you create a new file or open a file saved with an older version of Carrara, the Sound track is empty and the document has no sounds loaded. You can add sound to an animation by first loading sound files as master sounds, and then placing instances of them in the sound track. Currently, Carrara supports sound files in uncompressed Microsoft WAV and Apple AIFF formats. See "Animation Features of the Sequencer Tray" for more information about using the Sequencer tray.

Adding Soundtrack Channels

When you create a new document, or open an existing document without sounds, the soundtrack has a single, empty channel. The soundtrack can have any number of simultaneous channels, each with multiple sounds. Soundtracks appear at the bottom of the hierarchy in the Sequencer Tray.
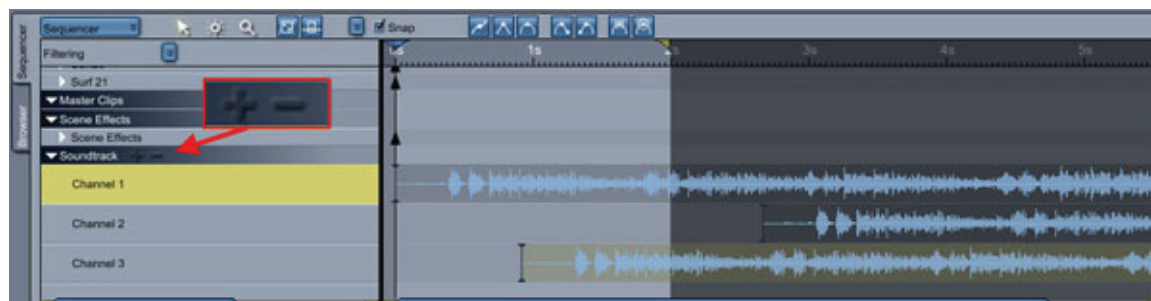
To add channels to the soundtrack:

- Click the + button next to the word **Soundtrack** in the Sequencer tray.

To remove channels from the soundtrack:

1. Select the channel to be removed by clicking on the channel's name.

2. Do one of the following:

    - Click on the - button next to the word **Soundtrack** in the Sequencer tray.

    - Right-click the sound channel and choose **Delete** from the menu that appears.

NOTE! When you remove a channel from the soundtrack, all sound instances in that channel will be deleted with it.

Adding a Master Sound to Your Project

Before you add sounds to your soundtrack you have to add a master sound to the project. Use the Properties tray: Sounds tab to add one or more sounds to your project. When a master sound is loaded, a small preview icon appears in the Sounds tab.

To load a master sound into the document:

1. Click on the **Sounds** tab in the Properties tray. Alternatively, choose the **Insert > Insert Sound** command.

2. Click the **Add** button.

3. Select the sound file to load, click OK.
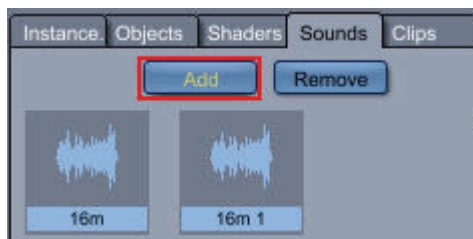
Removing Master Sounds

When you no longer want to have the master sound in your document you can remove it. For example, when you want to replace it with another sound. Removing a master sound removes all instances of that sound in the soundtrack.

To remove a master sound from the document:

1. Click on the **Sounds** tab in the **Properties** tray.

2. Select the master sound icon to be removed.

3.   Click the **Remove** button.
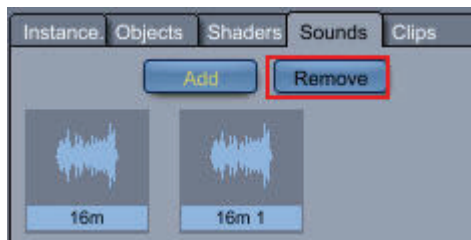
Adding Sounds to the Soundtrack

After you add a master sound in the Sounds tab of the Properties tray, you can drag and drop the icon from the Sounds tab into the soundtrack to create an instance of the sound.

To add sounds to the soundtrack:

Drag a master sound preview icon into the sound track and release it.
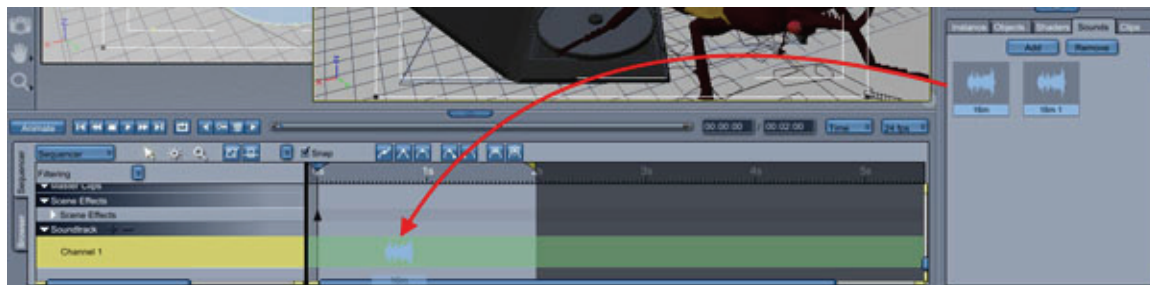
A preview of the sound will appear, beginning at the cursor location when you released the sound. Sound instances can be positioned within their channel by clicking and dragging them.
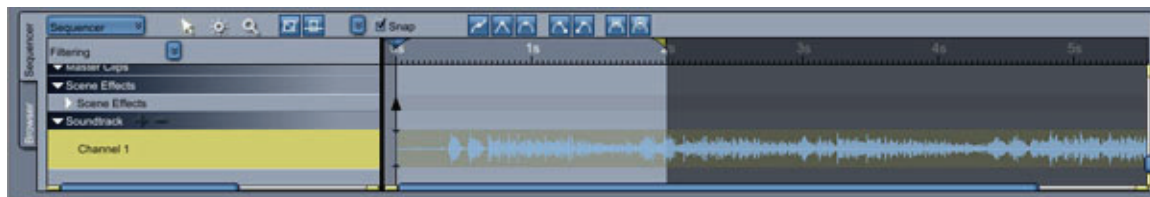
Rendering Animations With Sound

When rendering animations with sounds in the sound track, the sounds are automatically written to the resulting movie file after all frames have been rendered. If you abort a render in progress, sounds will be written to the animation up to when the render aborted.

NOTE! Not all of Carrara's output file formats support sound writing. A warning message is displayed if you try to render an animation with sound to a format where the sound track would be lost.

Setting Cameras

Setting Cameras

Just as a live action photographer strategically places cameras for different views of a scene, you can set up cameras for viewing and rendering your scene. This chapter covers how to create and use cameras. It includes descriptions of the different camera types and settings and how to manipulate them to achieve dramatic viewing angles.

Cameras for Viewing and Rendering

In the Assemble and Storyboard rooms, cameras provide your viewpoints of the scene. For instance, you can think of the scene window's 2D presets as views through fixed cameras that point straight at different sides of the working box. As you build your scene, you can add and position new cameras and shift your view to any of them to get the best view for working.

Once you are finished building and lighting your scene and are ready to render, you select a camera as the viewpoint for your scene. Using this camera, you set the Production Frame of the view that you want to render, just like looking through a viewfinder. The Production Frame, described in the next section, shows you exactly how much of your scene will be rendered.

NOTE! Carrara also lets you create camera effects like lens flare, filters, and depth of field. For information on these and other rendering effects, refer to "Rendering."

Creating and Setting Cameras

A new camera appears in your scene as an object that faces downward. Cameras are objects with projections, just like lights and modeled objects. You can move the camera as you would any other object. You'll learn about special features for moving and orienting cameras later in "Positioning and Aiming Cameras."

Creating a New Camera

Cameras are objects with projections, just like lights and modeled objects. You can move the camera as you would any other object. You'll learn about special features for moving and orienting cameras later in "Positioning and Aiming Cameras."

In the Hierarchy, cameras are listed as numbered camera objects. You can rename a camera just like any object. If you set up several cameras, you might want to name them according to their viewpoints. This makes it easier to select the view you want. For more information on naming objects in the Hierarchy, refer to "Naming Objects."
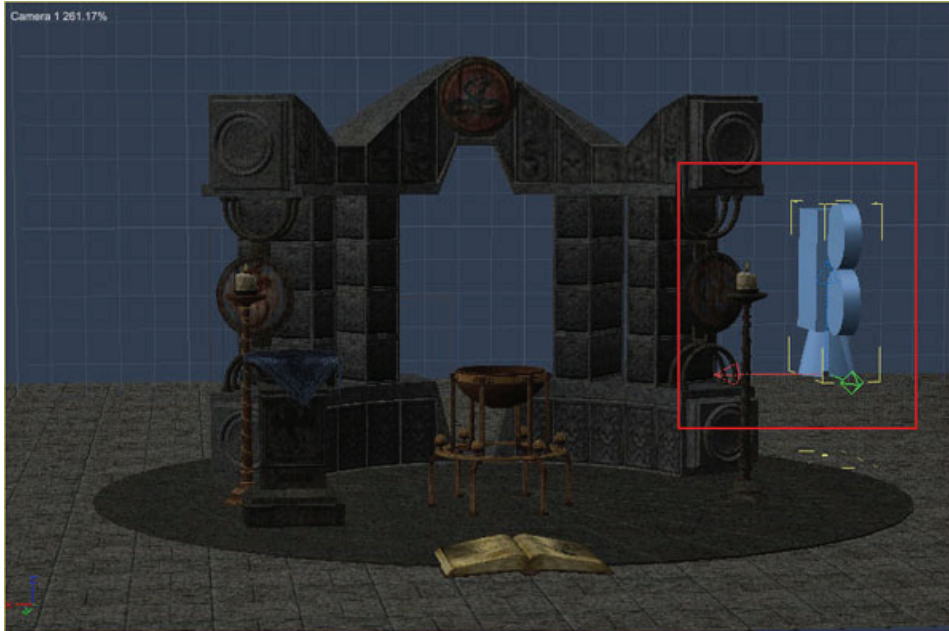
The current camera is not visible in the scene window. Because you are viewing the scene through it, it is not visible within the scene. You create specific lights in the Assemble or Storyboard room. All the light tools come off a single pop-up menu.

To create a new camera:

1. Choose the type of camera you want from the Create Camera tool pop-up menu. Choices are **Conical**, **Isometric**, and **Spherical**.
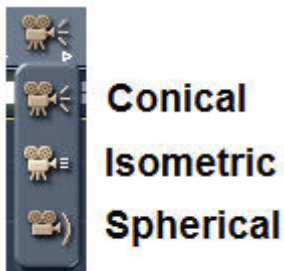
2. Drag the camera into the scene window. The new camera appears in your scene as an object that faces downward.

To create a new camera from the Insert menu:

1. From the Assembly or Storyboard room, choose one of the following:

    *   Choose **Insert >Conical** to create a conical camera.

    *   Choose **Insert >Isometric** to create an isometric camera.

    *   Choose **Insert >Spherical** to create a spherical camera.

2. The camera appears in the center of your scene. Position the camera to the desired location.

TIP! To make a camera easier to see (or see around), you can change its size with the scale control in the Properties tray: Motion tab: Transform panel just as you can with any other object. This does not affect your view through the camera.

Setting Camera Properties

You can change the properties of the current camera using the Camera controls on the **Properties tray: General tab: Camera panel**.

NOTE! For more information on camera properties, refer to "Carrara's Cameras."

To change a camera's properties:

1. Select the camera you want to change.

2. Display the **Properties tray: General tab: Camera panel**.

3. Expand the Camera list and choose the type of camera you want:

   - **Conical**: This camera has four settings: **Wide**, **Normal**, and **Telephoto** cameras have preset focal lengths. When you choose **Zoom**, you can set a focal length between 6 and 500 mm. The **Distance to Near Plane** setting TBD.

   - **Isometric:** This camera provides a view in which object size is not related to distance from the camera (that is, there is no vanishing point). There are two settings for this camera: **Zoom** (which increases or decreases the zoom of the camera view), and **Distance to Near Plane**, (which TBD).
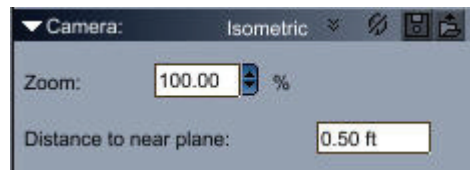
   With the isometric camera, use a Backdrop, not a Background. For more information on using a Backdrop, refer to "Backgrounds and Backdrops."

   - **Spherical**: This camera renders all items in your scene, including objects that are behind it. There are two settings for this camera: **Zoom** (, which increases or decreases the zoom of the camera view) and **Distance to Near Plane** (which TBD)
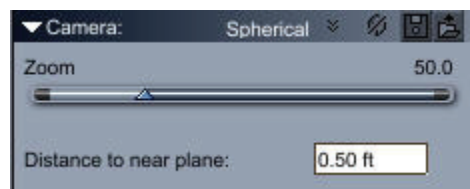
Carrara's Cameras

Carrara provides two types of cameras: conical and isometric. This section explains the basic functions of these cameras and the best usage of them. If you are familiar with basic photography concepts you may skip this section. If you want more detailed information, the techniques given in most reference books on still and live action photography may be applied to viewing and photographing scenes in Carrara.

The Conical Camera

A conical camera views a scene or object much as the human eye does: from a single point that widens as the distance from the eye increases. This creates perspective and the illusion of distant objects being smaller than close objects.

A conical camera is most useful for visualizing how this perspective affects your scene or model. It is generally the type of camera you use for rendering, since it most accurately reproduces how our eyes perceive distance and depth.
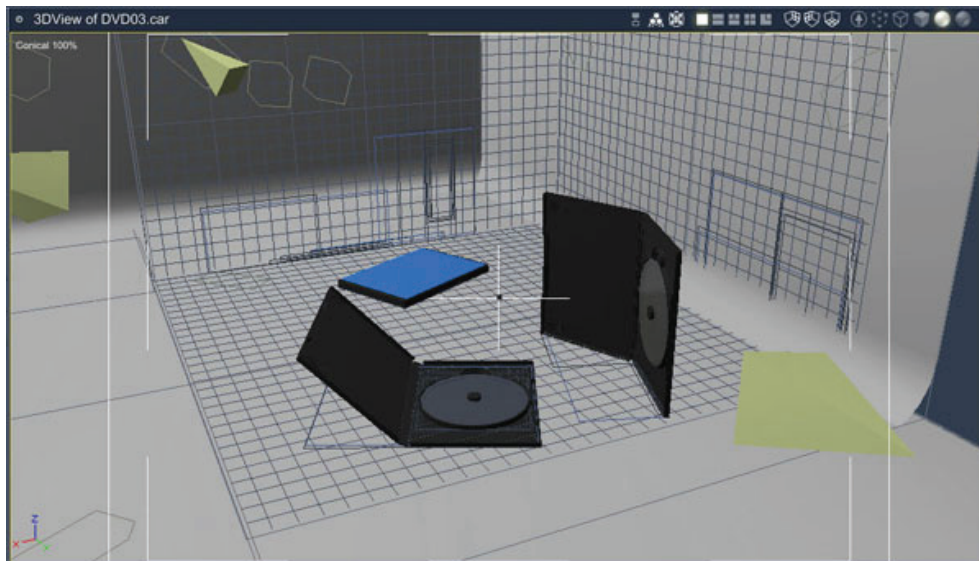
Conical Camera Lenses

With the conical camera, you can choose a lens setting of Wide, Normal, Telephoto, or Zoom. These correspond to lenses used on a real-world 35 mm camera.

- **Wide** has a focal length of 25 mm. This provides a wider field of view and a smaller image of your subject.

- **Normal** has a focal length of 50 mm and most closely approximates the human field of vision. It is the default lens for the Reference view camera.

- **Telephoto** has a focal length of 200 mm. This provides a larger image of your subject but a narrower field of view.

- **Zoom** lets you set any focal length between 6 and 500 mm. The shorter the focal length, the more of your scene you see and the smaller your subject appears.

Distortions of the Conical Camera

In real life, a conical camera creates certain distortions that do not equal what the human eye perceives. Some of these distortions are present in the Carrara conical camera while others are not.

The conical camera's view appears to bend in at the sides while shrinking objects in the center of the frame and enlarging objects at the sides.

When you move a conical camera away from the subject and then zoom in, there is less distortion.

To achieve exaggerated perspective, move the camera closer and set the zoom to a lower number.
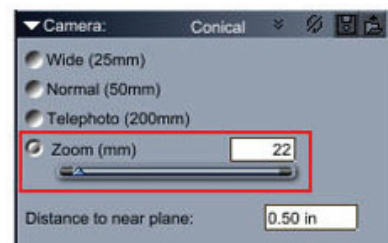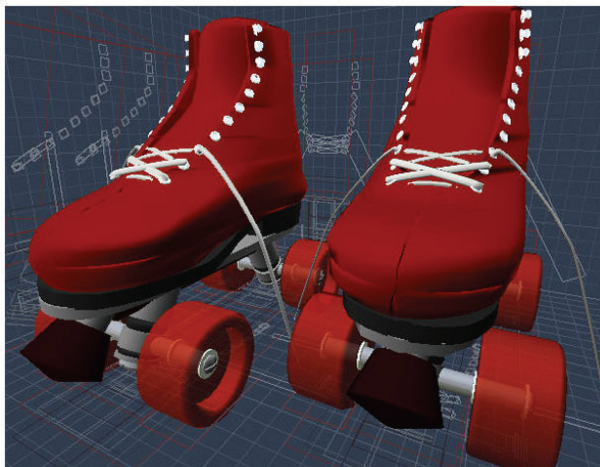
One distortion that is not captured by Carrara's conical camera is background blur; with a real-life camera, this occurs when you use a long focal length.

You can simulate this background blur using the Depth of Field controls. For more information, refer to "Using the Depth of Field Effect."

The Isometric Camera

While the conical camera views a scene from a single point, the isometric camera views a scene from a plane. In an isometric view, object size is not related to distance from the camera, every edge appears the size it actually is, and parallel lines appear parallel instead of tapering as they would in a perspective view. In contrast, from a conical camera, the scale of objects diminishes as their distance from the camera increases.

The isometric cameras view the scene without perspective. These views are similar to architectural and engineering drawings, where placement and absolute relations between objects are important.

NOTE! that six isometric cameras can't be repositioned: Top, Bottom, Front, Back, Left and Right. They are attached to the working box and move with it.

The most effective way to work is to model and align objects using an isometric camera, then examine and render your work with a conical camera. The 2D presets are views through isometric cameras.

Zooming with the isometric camera simply scales the view, making objects appear larger or smaller.

TIP! Use an isometric camera to render logos without perspective or distortion.

The Spherical Camera

When doing a final rendering with the spherical camera, the output image shows all objects that can be seen from the camera's position, including objects behind it!

This is useful when creating an immersive view of a scene. Certain software products enable you to take this image rendered with the spherical camera in Carrara and save it as a QuickTime VR file. It can then be used in a web browser, giving you an interactive, 360° view of the original scene.

In the assemble room, the spherical camera behaves like the conical camera and renders the portion of the scene in front of it from a single point.
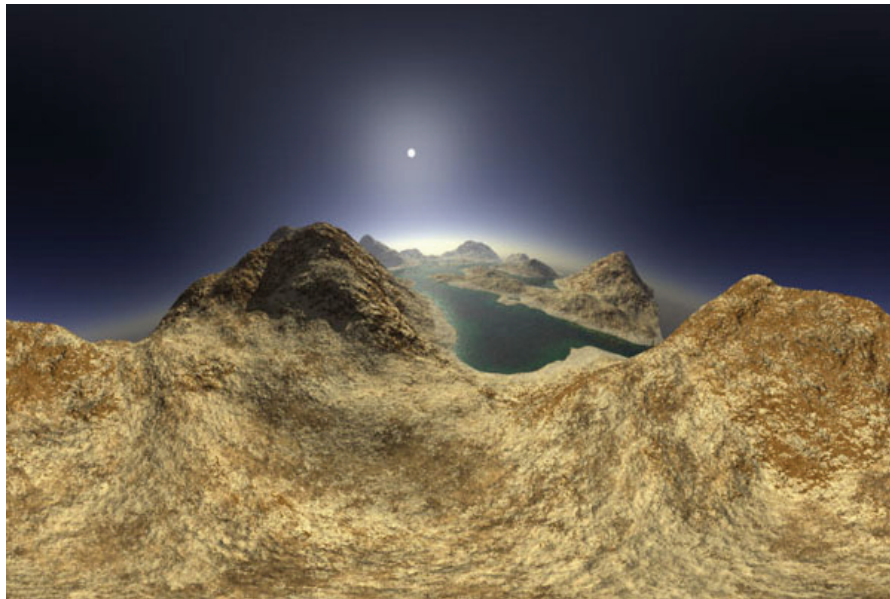
Working with Multiple Cameras

Multiple cameras help you arrange your objects by providing different views of your scene. You can switch between these views or use them simultaneously by using a multi-pane layout of your scene window and looking through a different camera in each pane.

When you are ready to render, you select one of the cameras for the view to render. For details, refer to "Cameras and Rendering."

Changing Views

All views are shown through cameras; you can animate many of them, render through them, and use modifiers on them, and so on.

Click the view controls to select how many views are displayed. These controls are discussed in "Displaying Multiple Camera Views." After you display your views, you can select the camera to display in each view.

Selecting a Camera

The default view of your scene is through the default camera. You can send the camera to many default positions and orient it using the Camera tools. You can also add additional cameras to the scene.

A camera selection menu appears in the upper-left corner of the document window. You can use this menu to easily switch cameras so that you can see one or more different views of your scene.

To choose a camera to view your scene:

- Choose the camera you want to use from the camera menu in the document window.

- If you have multiple views displayed, you can specify a camera for each view.

    o **Conical cameras** view the scene as the human eye would: objects far away appear smaller, and parallel lines converge at a vanishing point.

    o Choose one of the isometric camera views such as **Left, Right, Top, Bottom, Front,** or **Back** to use one of Carrara's preset cameras.

    o The **Director's Camera** is a perspective, non-animatable camera that allows you to see a perspective view of your scene. You can reposition the Director's camera as needed. It is useful for navigating around in the scene without accidentally animating a camera.

    o If you ever "get lost" or aim a camera so you can't see where you are oriented, change to the Reference position to re-orient yourself and re-aim the camera. For further information, see "Saving and Using Camera Positions."

Positioning and Aiming Cameras

Cameras (other than the current camera) appear in the scene window as objects. You can move them anywhere in the 3D workspace and change their angle with any of the standard positioning and orientation features.

Camera Navigation

Carrara lets you directly manipulate the current camera by using the camera controls to track, dolly, pan, and bank the camera.

Track YZ

Track XY

Track XZ

Dolly

Pan

Bank

TIP! You can also select the current camera and use the Position (Center and Hot Point) and Rotation controls in the Properties tray: Motion tab: Transform panel to move and aim the camera numerically.

To work faster, you can move and orientate the camera with a 3-button mouse using the following shortcuts:

- **Option/Alt + Left Mouse Button** will rotate the camera (Dolly).

- **Option/Alt + Right Mouse Button** will Track the camera horizontally and vertically.

- **Option/Alt + Middle Mouse Button** will move the camera back and forth.

- If your mouse is equipped with a **Mouse Wheel** you can use it to move your camera back and forth.

NOTE! In isometric views it is not the camera that is moved when using those shortcuts but the zoom and pan that are modified instead.

Tracking

You can track the current camera in the following ways:

- Vertical and depth track

- Horizontal and vertical track

- Horizontal and depth track

- Depth track only

Tracking affects the camera's position without affecting its orientation. The objects in your field of view appear to change position in relation to each other as you view your scene from different vantage points.

The camera tracks in screen space, not on the axes of the universe or working box.

To perform vertical and depth tracking:

- Click the YZ Track control and drag in the scene up, down, left, and right. You can also drag on the Track control itself.

- Drag left and right to perform depth tracking.

- Drag up and down to perform vertical tracking.

- The YZ Track control moves the camera up, down, in, and out in screen space. This changes your field of view and perspective if you are using a conical camera.

To perform horizontal and vertical tracking:

- Click the XY Track control and drag in the scene. You can also drag on the Track control itself.

- Drag up, down, left, and right.

- The XY Track control moves the camera up, down, left, and right in screen space.

To perform horizontal and depth tracking:

- Click the XZ Track control and drag in the scene. You can also drag on the Track control itself.

- Drag left and right to perform horizontal tracking.

- Drag up and down to perform depth tracking.

- The XZ Track control moves the camera left, right, in, and out in screen space. This changes your field of view and perspective if you are using a conical camera.

To perform depth tracking only:

- When using any of the camera controls, hold down the Command/Ctrl key and drag upward to move the camera forward.

- Hold down the Command/Ctrl key and drag downward to move the camera back from your subject.

Dollying

Dollying changes the camera's position and orientation while keeping it pointed at the same spot in the scene. The visual effect is similar to rotating an object, except in this case, it's the camera that moves.

To dolly the camera:

1. Select the object around which you want to rotate the camera. If you deselect all objects, the current camera rotates around the origin of the universe.

2. Click on the arrow at the bottom right corner of the camera orientation control.

3. A choice of controls will be displayed. Select the Dolly tool.

4. Drag in the scene window to dolly the camera. You can also drag on the camera orientation control itself.

Panning

Panning only changes the camera's orientation. To visualize panning, imagine standing in place and looking through your camera's viewfinder. Turn side to side and tilt up and down. The objects in your field of view do not change position in relation to each other.

To pan the camera:

1. Click on the arrow at the bottom right corner of the camera orientation control.

2. A choice of controls will be displayed Select the Pan tool.

3. Drag in the scene window to pan the camera. You can also drag on the camera orientation control itself.

Banking

Banking changes the camera's orientation only, rotating the camera on the Z axis. This creates the effect of a tilted horizon, which is useful for creating tilted cockpit views.

To bank the camera:

1. Click on the arrow at the bottom right corner of the camera orientation control.

2. A choice of controls will be displayed.

3. Drag left and right to bank left and right. You can also drag on the camera orientation control itself.

Pointing a Camera at an Object

In addition to the camera controls, Carrara offers several methods of aiming cameras, including the Point At command, the Point At modifier, and the Direct Manipulation controls.

TIP! To direct a camera to a particular area in your scene, create a temporary object for the camera to point at, then delete the object. Remember that the camera points at the hot point of the selected object.

The Point At Command

The Point At command is usually the easiest way to aim a camera in the general direction of an object. This command points the camera at the hot point of the object. Then, you can make precise adjustments to the camera's position and aim using the camera orientation controls.

The Point At command does not link the camera to the object selected. That is, if you move either the object or the camera, the camera no longer points at the object. To make a camera follow an object, use the Point At modifier, described in the next section.

NOTE! You cannot point a camera at a light.

To point a camera at an object:

1. Select the camera that you want to aim.

2. Select the object or objects you want to point at. You'll probably find it is easiest to select them from the Hierarchy in the Sequencer tray or in the Properties tray.

3.   Choose **Edit > Point At**.
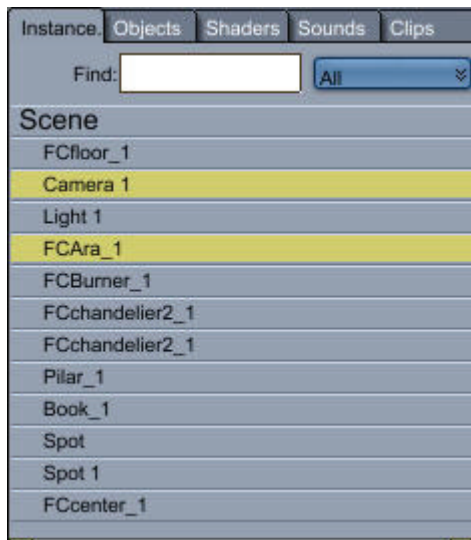
The Point At Modifier

If you want the camera to follow an object, apply the Point At modifier. If either the object or the camera moves, the camera automatically reorients itself to point at the object in its new position.

To have a camera follow an object:

1.   Select the camera you want to aim.

2.   On the **Properties tray: Modifiers tab**, click the + sign to expand the list and choose **Behaviors > Point At**.
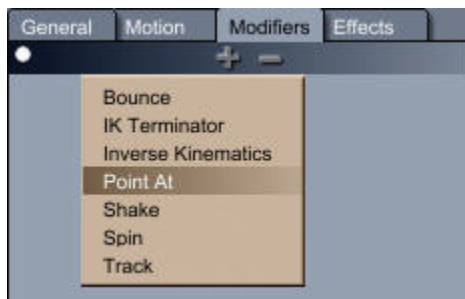
3.   The Point At panel appears in the Modifiers tab. Click the Towards button to select one or more objects from the Point At dialog.
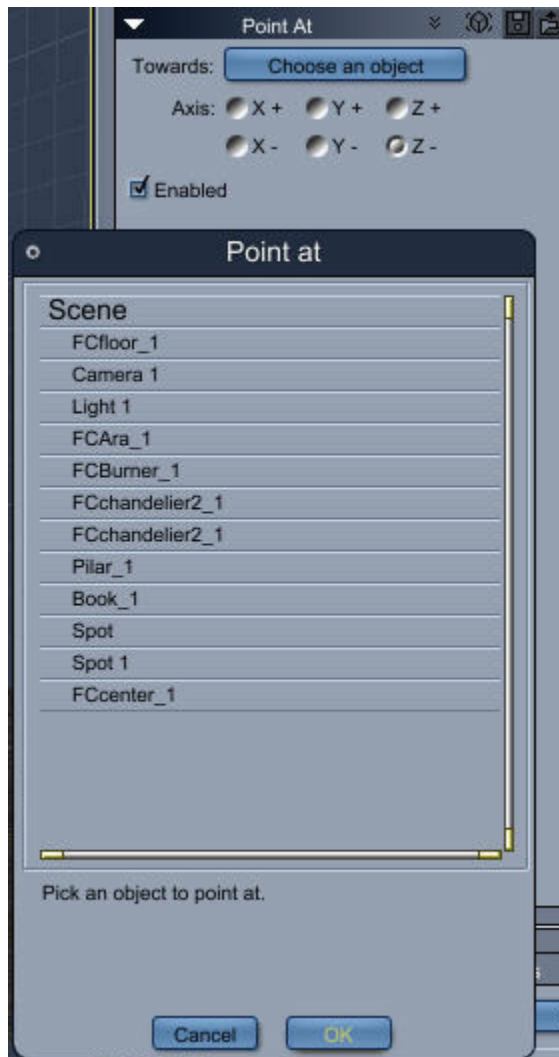
4. The **Axis** option sets which axis of the object points at the target. In most cases, you can leave this option as is.

5. Check the **Enabled** option to turn the Point At modifier on.

6. After you apply the Point At modifier, the camera reorients to point at the object when you move either the camera or the object that it points toward.

Controlling a Camera Directly

The Direct Manipulation controls for a camera provide an on-screen description of the camera's aim, zoom level, and position. You can drag control handles to change these properties.

NOTE! These features are not available for the current camera. You must view the camera as an object to control it directly.

To display a camera's Direct Manipulation controls:

1. Select a camera in your scene window. Note that you will not be able to select the camera from the viewport that looks through the selected camera.

2. Display the **Properties tray: General tab: Camera panel**.

3. Click the Direct Manipulation button to display the controls on the selected camera.

The controls appear as a set of wires with handles at certain points. Each handle represents a control you can drag to set properties.
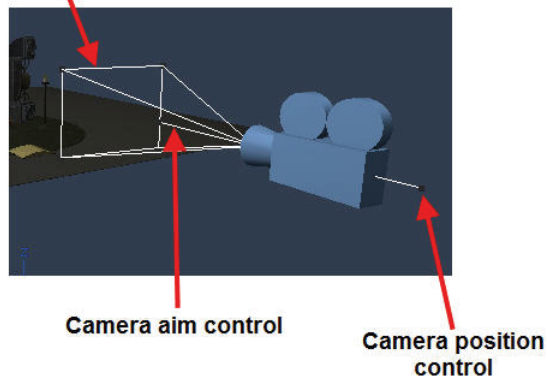
4. Drag the wireframe and projection handles to aim the camera and set its properties.

TIP! You may want to aim the camera with the Direct Manipulation controls while looking through that camera. You can do this by looking through the camera in a different pane.

To aim the camera:

The cursor turns into a crosshair when placed over the camera aim control, which appears in front of the camera. When you move the camera aim control, the camera rotates to aim where you place the handle.

- Drag the camera aim control to move parallel to the active plane.

- Click the camera aim control then hold down the Option/Alt key to drag the target handle parallel or perpendicular to the active plane.

- You can drag the aim control out, extending the wire all the way to the point you want to aim at. The wire shows the direction of the camera's aim.
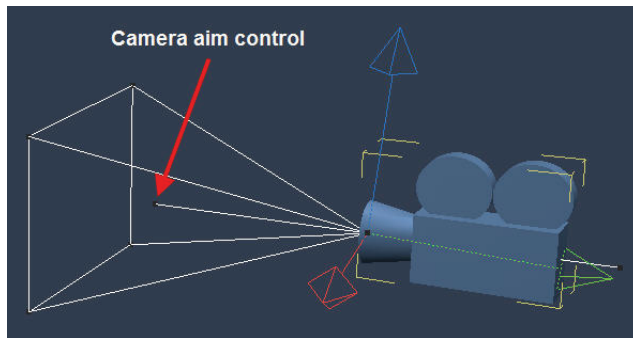
As you drag the Camera Aim control, the camera rotates around the Camera Position control.

To move the camera:

The cursor turns into a crosshair when placed over the camera position control, which appears behind the camera. When you move the camera position control, the camera moves to the new position and orients itself to point at the target.

- Drag the handle behind the camera preview.
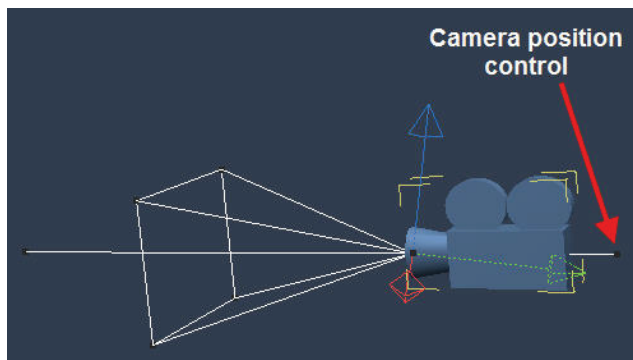
As you drag the Camera Position control, the camera rotates around the Camera Aim control.

To change the camera's zoom:

The rectangle in front of the camera represents the production frame.

- Drag a corner of the rectangle in front of the camera to change the zoom.
- Drag toward the target object to zoom in.
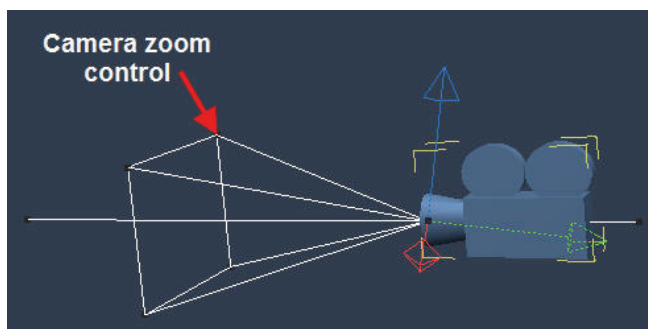
Saving and Using Camera Positions

You can save any camera position you set. Saved positions are available from the camera list in the upper left corner of the scene window.

To save a position as a preset:

1. Select the camera with the position and orientation you like.

2. Expand the view menu in the upper-left corner of the document window, and choose **Set Position to >Save Position**. The Save Camera Position dialog appears.
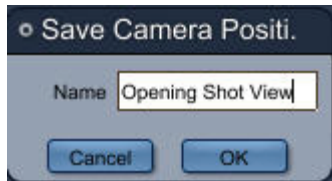
3. Type a name for your camera position.

4. Click OK. Your new camera position now appears as a choice in the view menu.

To remove a preset from the list:

1. Expand the view menu in the upper-left corner of the document window, and choose **Set Position to > Delete Position**. The Remove Position dialog appears.

2. Select the position you want to remove and click OK.

To move a camera to a preset position:

You can use the camera view menu to move the camera to any preset position. When you jump into an object or group using any of the default preset positions, the camera goes with you:that is, it establishes the preset position and orientation relative to the local universe you've jumped into. A custom position, on the other hand, is kept as is when you jump in. This could result in an empty window when you jump into a group. Choose a default preset or select an object and Point At it to see the contents of the group.

NOTE! There is a difference between the Left, Right, Top, Bottom, Front, and Back presets on the top level of the camera list and those within the Set Position To menu. On the top level, these settings refer to isometric cameras that you cannot manipulate. The settings in the Set Position To menu refer to physical locations to which you move the current camera.

1. Select the camera you want to look through from the camera list. Choose one of your own cameras or the Director's Camera. Since the Carrara preset isometric cameras are fixed, you cannot use the Set Position To option for them.

2. From the camera list, choose **Set Position To**, then choose a position from the list:

   - **Left, Right, Top, Bottom, Front**, and **Back:** These presets move the camera to the position of the isometric preset cameras.

   - **Director's Camera**: This preset moves the camera to the current position of the Director's Camera.

   - **Reference:** This preset moves the camera to the original view of the scene window.

   - **User-created**: Any preset positions you have created and saved appear here.

Cameras and Rendering

What you see in the scene window does not necessarily equal what will be rendered. For example, you might adjust your view with the 2D Pan and 2D Zoom tools, but these adjustments are not updated to the settings of the current camera. The Production Frame and the camera settings are the key elements to determining the size and framing of the rendered scene.

The Camera's Field of View

The inner, dashed-line rectangle is called the Title Safe Margin. This is an "out of bounds warning" for objects that approach the edge of the frame. Use this reference when developing video output. Because the edges of the video frame are often cut off, the Title Safe Margin can keep objects fully within the visible area when the animation is displayed on video.

Production Frame   Title Safe Margin
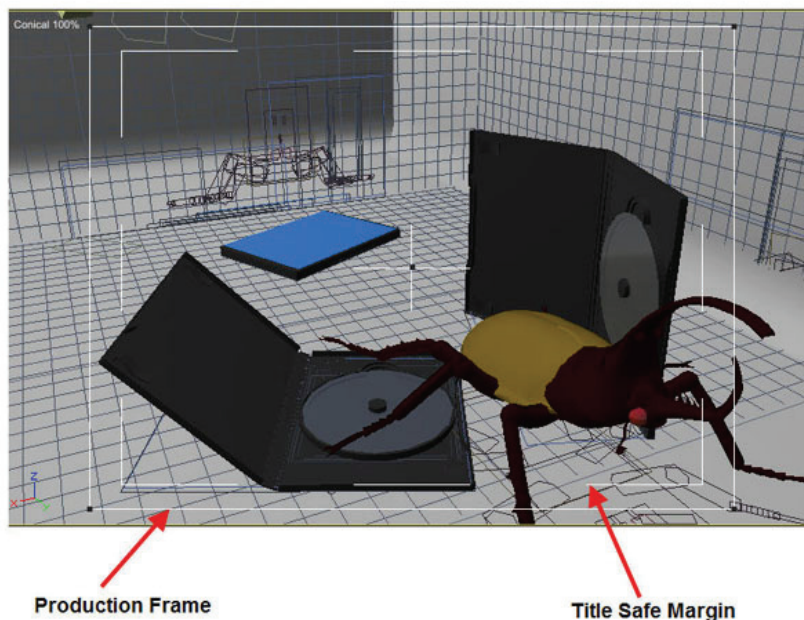
For more information on the Production Frame and Render Settings, refer to "Viewing the Production Frame" and "Rendering."

Viewing the Production Frame

The Production Frame shows you exactly what portion of your scene will be rendered, just as looking through a camera's viewfinder shows you what portion of a real-life scene will actually be captured on film. By default, the Production Frame appears as a rectangle on top of your view.

To view the Production Frame:

• Choose **View > Show Production Frame**. The Production Frame appears as a colored rectangle in the Universe.

To resize the Production Frame:

1. Choose the Selection tool.

2. Click the outer rectangle to select it. Handles appear on each corner

3. Drag a corner handle to resize in two dimensions. Hold down the Shift key while you drag to constrain proportions. The camera position does not change when you move or adjust the Production Frame. Only the camera's view is affected.

Setting the Production Frame Safe Area

When you create animations and graphics for video or broadcast media, allowances must be made to keep the most important visual elements within a safe viewable area. If you don't make allowances for overscanning, you may find that some elements are not seen on the screen. This is known as the **production frame safe area** (also known as the **title safe area**).

Carrara provides a visual guide that helps you keep the important content in your scene within this safe area. An inner, dashed-line rectangle serves as an "out of bounds warning" for objects that approach the edge of the frame.

You may need to experiment with the percentage setting to find the setting that's right for your video output system.

To adjust the Production Frame Safe Area:

1. Choose **View > Production Frame.** The Production Frame Setting dialog appears **.**

2. Adjust the Production Frame Safe Area slider, or enter the percentage you need.

3. Click OK. The inner dashed line updates to reflect the setting you choose.

Using the Depth of Field Effect (Standard, Pro)

Normally, every object appears in focus regardless of its distance from the rendering camera. The Depth Of Field effect simulates the lens of a real life camera. It blurs the objects according to their distance from the rendering camera.
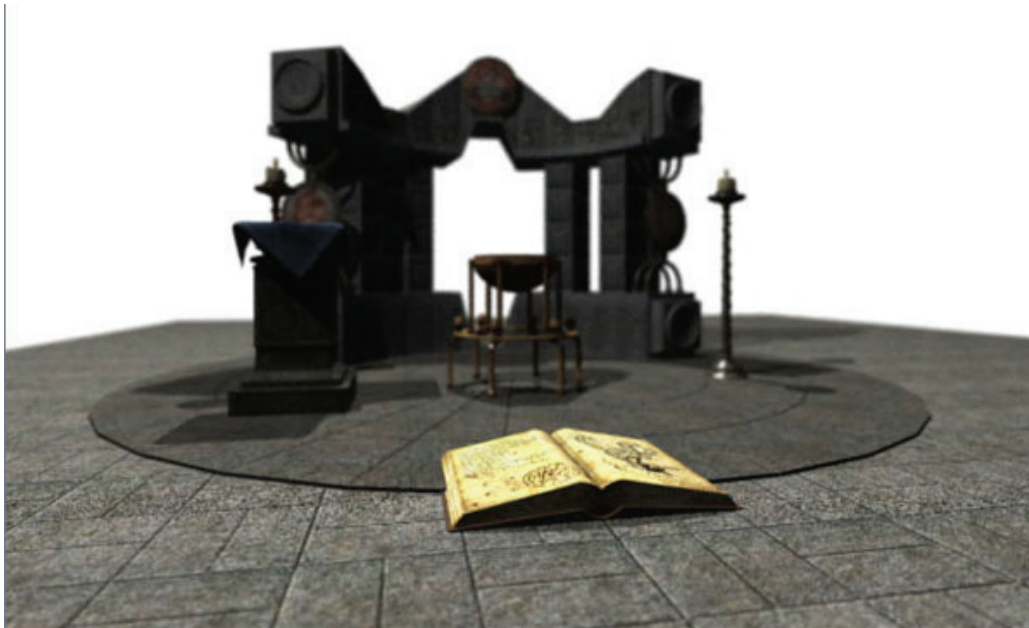
To set Depth of Field options:

1. Select the camera that you want to use for the Depth of Field effect. Usually it is the camera that you will use for your render.

2. In the **Properties tray: Effects tab: Depth of Field panel**, check the **Enable** box and click the **Edit** button.
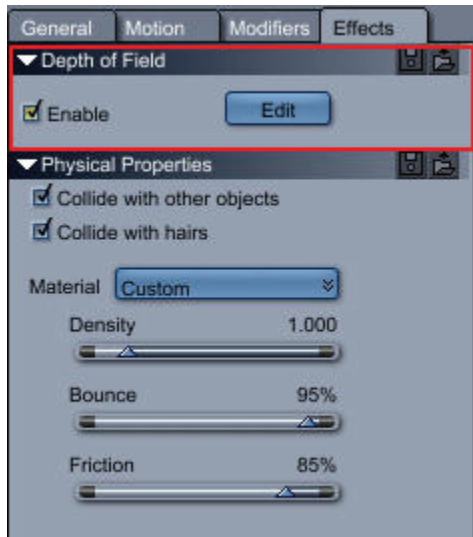
The Depth of Field dialog comes up.
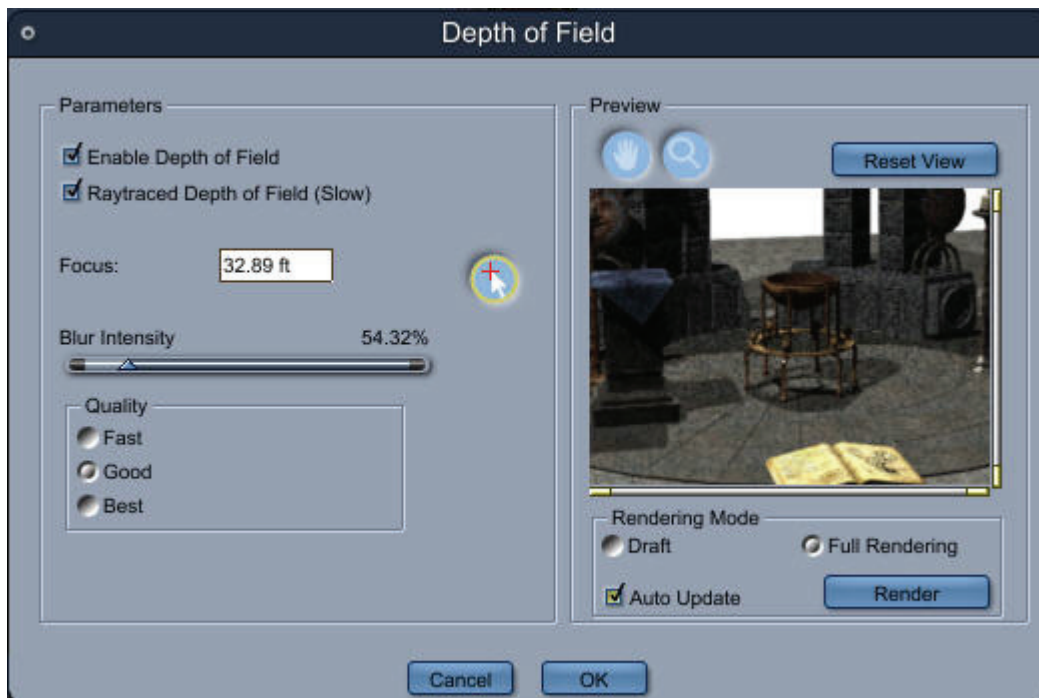
3. In the **Parameters** section, set the following options in the Depth of Field dialog:

- **Enable Depth of Field**: Check or uncheck this option to enable or disable depth of field.

- **Raytraced Depth of Field(Slow)**: Check this option to get a better quality depth of field result.

- **Focus**: The focus is the distance from the camera to the objects that appear sharp in the image (the object is in focus). To set the focus, simply type in the distance or use the Focus tool. The Focus tool allows you to pick the distance directly in the preview. When you click in the preview, a red arrow appears and the focus distance will change to match

the distance of the closest object at the center of the arrow.

- **Blur Intensity**: Determines how much blur will be applied. When set at zero, you get a perfectly sharp image.
- **Quality**: Choose **Fast**, **Good**, or **Best** quality for your depth of field render. Higher settings consume more resources.

4. Use the **Preview** section to fine tune the image as follows:

- **Zoom and Pan controls**: Use these controls (above the preview image) to get a better view of the details in the preview image.
- **Reset View**: Click this button to return the preview image to its default size and position.

5. Use controls in the **Rendering Mode** section as follows:

- Rendering Mode: Choose **Draft** or **Full Rendering** to select the quality of the preview image. Full Rendering will take longer but will use the same settings as your final render.
- **Auto Update**: Check this option to update the preview automatically while you change depth of field settings.
- **Render**: Click this button to manually render the preview image.

6. Click OK to accept the settings and close the Depth of Field dialog.

Setting Lights

How objects appear in a Carrara scene is influenced greatly by the light in which they are viewed. Lighting contributes to effects such as shadows and colors. Lighting also influences the shaders applied to your objects. Sometimes you can adjust the lighting instead of editing a shader to create a specific result.

This chapter covers how to create and use lights. It includes descriptions of the different types of lighting and how you set the options for each.

Carrara supports several different types of lights. You may add as many lights as you like, but as the number increases, so does the rendering time. Most scenes can be lit with one, two, or at most three well-placed lights.

Tip Carrara also lets you create visible light spheres and cones as rendering effects. The visible light cone effect is like a searchlight cutting through the fog. For more information, refer to "Rendering Effects."

Lighting Concepts

This section explains basic lighting concepts. If you are already familiar with the subject, you may skip this section. If you want more detailed information, the techniques given in most standard reference books on lighting and photography can be applied in Carrara.

A good set of lighting conditions is an important step toward creating high-quality artwork. The same scene rendered under varying light can provide strikingly different results. For example, rendering with all lighting at zero brightness is like taking a photograph:without a flash:in the bottom of a coal mine. Conversely, too much lighting washes out subtle effects.

Three-Point Lighting

Three-point lighting is the classic way to place lights. It provides illumination while preserving shadows and the illusion of depth.

Place two lights at 45° angles to the front of your scene. Make one of these a **key light**. This is the main source of illumination and strong shadows. Make the other a **fill light**, which softens the shadows and supplies the mid-tones.
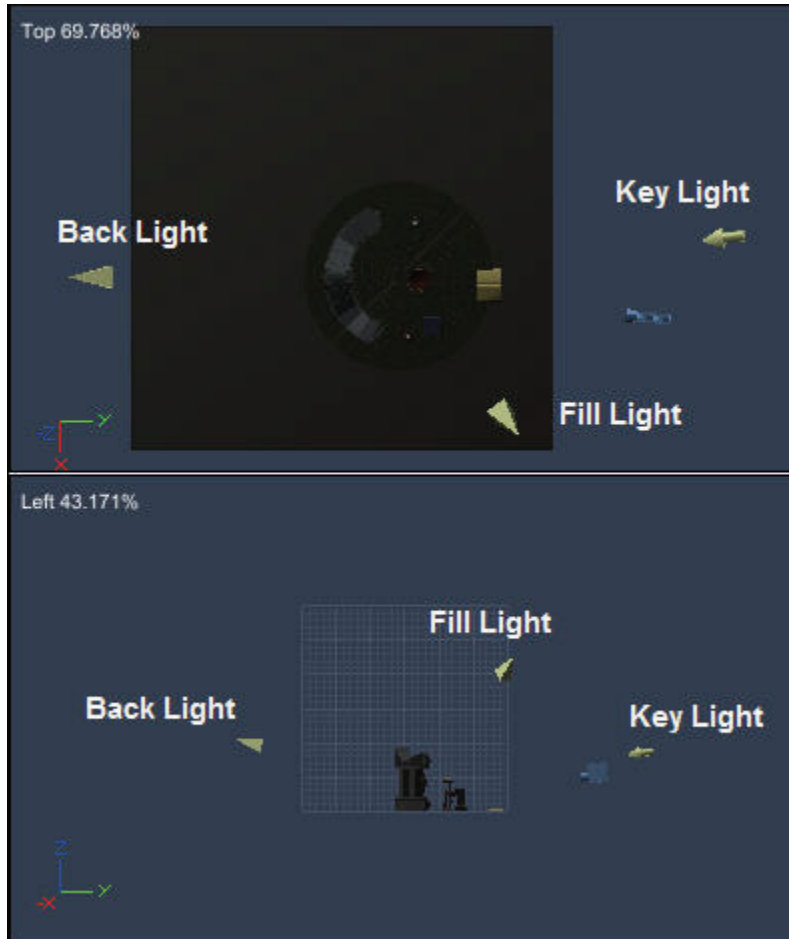
It is also a good idea to color one of these lights warm and the other cool:which one you color warm and which cool depends on the mood and effect you want to achieve. To simulate outdoor light, choose a cool color for the key light; to simulate indoor light, choose a warm color for the key light.

Set a **backlight** behind your subject to separate it from the background, adding a feeling of depth. If you want to create the effect of light creeping over the subject's features, elevate the light slightly.

Other Lighting Arrangements

Here are some of the effects of different light placements.

- **Frontlighting** is placing a light source in front of your subject. This allows clear rendering of details and colors, but reduces shadows and depth.

- **Sidelighting** creates dramatic light-and-shadow effects. If you don't want some objects to be completely obscured by shadows, you can place a softer fill light in the scene on the opposite side of the subject from the sidelight.
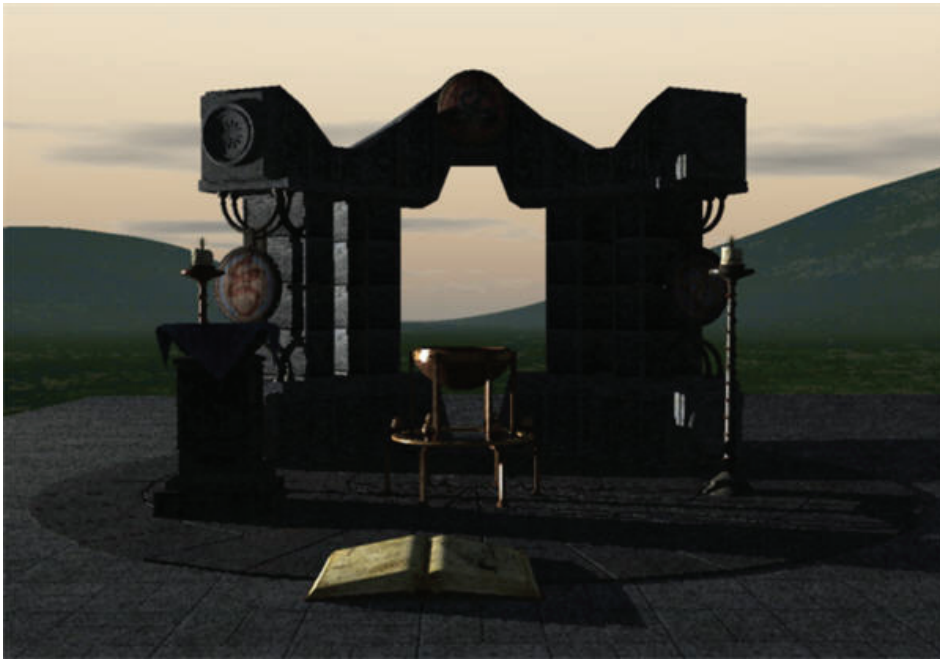
- **Backlighting** is placing a light source behind your subject. This creates silhouettes. You can combine backlighting with spots set at the side or front of the subject to cast shadows from behind while brightly illuminating a small area, such as a face.

- **Bottomlighting** is placing a light source below your subject. Like backlighting, this also creates silhouettes. It can be very effective if you are lighting a transparent subject. However, if you want to show more detail, add a fill light above your subject.

Working with Lights

A new light appears in the scene as a colored object that points toward the center of the universe.
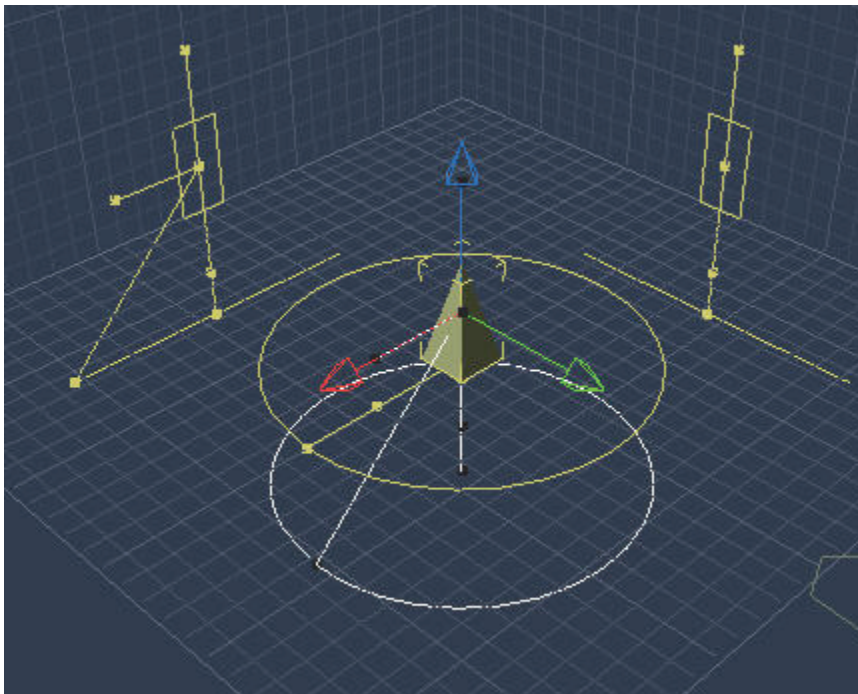
Lights are objects with projections, just like cameras and modeled objects. You can move the light as you would any other object. You'll learn about special features for moving and orienting cameras later in "Shadow Options."

In the Hierarchy, lights are listed as numbered light objects. You can rename a light just as you can any object. If you set up several lights, you can name them according to their position and type to make it easier to select the light you want. For information on naming objects, refer to "Using the Sequencer Tray."

You create specific lights in the Assemble or Storyboard room. All the light tools appear off a single pop-up menu.

To create a new light source:

1. Choose the type of light you want from the Create Light tool pop-up menu.

2. Click in the scene window where you want to place the light.

You can also choose **Insert > (Type of) Light** to add a light at the center of the universe and then drag it to any position you like.

If you want to create a light with the same settings as an existing light, select the existing light, then choose **Edit > Copy and Paste**, or **Edit > Duplicate**.

After creating a light, you can change its properties.

TIP! To make a light easier to see (or see around), you can change its scale using the Scale control in **Properties tray: Motion tab: Transform panel** as you would any other object. This does not affect the brightness of a light or any other light characteristics.

Setting Light Properties

Once you have created a new light, you can specify the type of light you want to use, and then set its options in the Properties tray. You can use the light controls at any time to adjust the settings of a selected light.

NOTE! You can also use these controls to adjust the default distant light.

To set a light's properties:

1. Select the light that you want to modify.

2. Display the **Properties tray: General tab: Light panel**.

3. Choose the type of light you want. When you change the light type, the panel displays the parameters appropriate to that type.

Setting Common Light Characteristics

The characteristics of color, brightness, and shadow control are common to distant, bulb, and spot lights.

To set common light characteristics:

1. Display the **Properties tray: General tab: Light panel**.

2. Adjust the controls as appropriate:

    -   **Brightness** sets the light's brightness. The color chip launches the Carrara color picker, which you can use to set a color for the light.

    -   **Casts Shadows** makes the light cast shadows.

    -   **Shadow Intensity** sets the intensity of shadows. A lower setting creates weaker shadows. By default, all lights except ambient light cast shadows. Shadows appear only in renderings. Use the Test Render tool or render the image to view the changes.

For more control over shadows, refer to "Shadow Options."

Viewing Lighting Effects

You can quickly view your lighting effects using the Test Render tool to preview lighting and shading effects.

To view the entire image, use the Render room. For details, refer to "Rendering."

Carrara's Light Types

Carrara offers several different types of lights that you can add to your scenes and landscapes.

Spot Lights

A spot light emanates from a single point and casts light in a specific direction. It works the same way as a stage spotlight by making what appears in the light the center of attention.

Spot lights appear in the scene window. You can move them anywhere in the 3D workspace and change their direction with any of the positioning and orientation features, including the Selection tool, the Rotate tool, and the **Properties tray: Motion tab: Transform panel** controls.

Setting Spot Light Characteristics

To insert a spot light and set its characteristics:

1. To insert a spot light, do one of the following:

   • Choose **Insert > Spot**.

   • Drag the Spot Light tool into your scene.

2. With the light selected, display the **Properties tray: General tab: Light panel**.
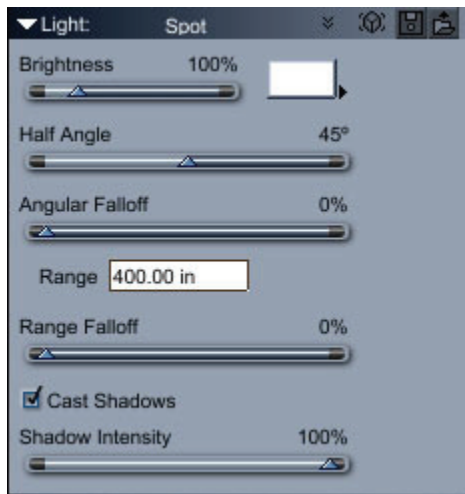
3. Set the following properties as needed:

- **Half-angle** is the angle of the radius of the cone. A narrow angle creates a beam like that of a spot light. A wide angle creates a beam like that of a floodlight.
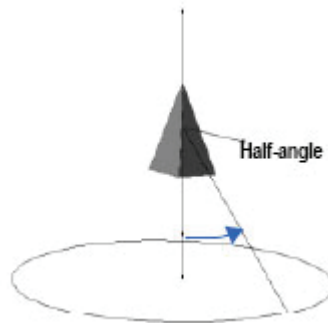
- **Brightness**: Adjust the Brightness setting to increase or decrease the strength of the distant light.

- **Color chip**: Click to set the color of the light.

- **Angular Falloff**: Drag the slider to set how the brightness of the light diminishes toward the edge of the light cone. A falloff of 10% means that the light has full intensity from the center to 90% of the radius of the light cone, then decreases linearly to the edge of the cone.
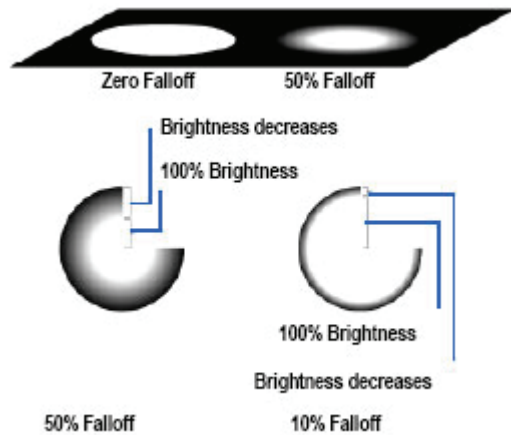
- **Range** : Enter a value for the distance from the light itself to the point where the light has no effect. The units of measure are those currently set in **File > Preferences> General Scene Settings**.
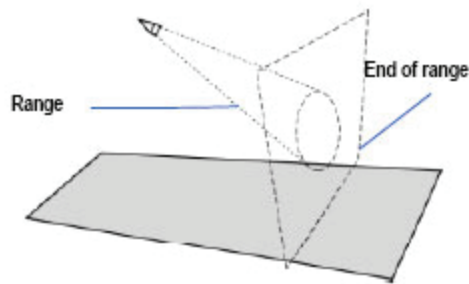
- **Range Falloff** : Drag the slider to determine how the brightness of the light diminishes toward the edge of its range. A falloff of 10% means that the light has full intensity from the source through 90% of its range, then decreases linearly to the end of the range.
- **Cast Shadows**: Check this option to generate shadows with the light. Uncheck the option if you do not want the light to cast shadows.
- **Shadow Intensity**: Adjust the slider to increase or decrease the strength of the shadow.

Directly Controlling Spot Lights

You can use the Direct Manipulation controls to change all the characteristics of a spot light except color.

To aim the spot light:

- Drag the control handle in front of the light cone.
    - The handle moves parallel to the active plane.
    - Hold down the Option/Alt key to drag the handle perpendicular to the active plane. You can drag the handle out, extending it all the way to the point you want to aim at. The wire shows the direction of the light beam.
    - You can also drag the target handle on one of the light's projections to aim the light on that plane.

      ○   The light rotates to aim where you place the handle.
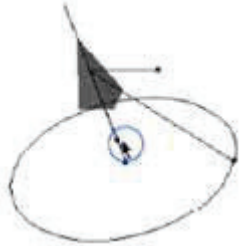
To move the position of the spot light:

- Drag the handle behind the light cone.

You can also drag the handle on one of the light's projections to position the light on that plane.

The light moves to the new position and orients itself to point at the target.

To check the light cone diameter at a given point:

- Drag the handle on the direction wire, centered in the ring.

The ring around the direction wire describes the cone diameter at that point. Notice that as you drag the handle closer to the target, the ring expands.

NOTE! The Cross-Section control is intended to help you visualize the diameter of the light cone at different ranges. It does not change any of the light's properties.

To change the spot light's half angle:

- Drag the handle on the ring perimeter.

TIP! If you have the **Properties tray: General tab: Light panel** displayed, you can see the effect of your changes in the preview and settings.

To change the Angular falloff:

- Hold down the Cmd/Ctrl key while dragging in toward the center of the ring.
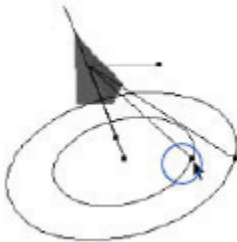
TIP! If you have the **Properties tray: General tab: Light panel** displayed, you can see the effect of your changes in the preview and settings.

To change the brightness:

- Drag the handle on the wire that extends from the light perpendicular to the direction wire.

- Drag away from the light to increase the brightness, or towards the light to decrease brightness.

Bulb Lights

A **Bulb Light** emanates from a single point and casts light evenly in all directions, like a naked bulb in a lamp.

You can move bulb lights anywhere in the workspace with any of the positioning features, including the Selection tool, Rotate tool, and The positioning controls in the **Properties tray: Motion tab: Transform panel**. Because bulb lights shine equally in all directions, you do not need to aim them.

Bulb light controls appear in the **Properties tray: General tab: Light panel**.
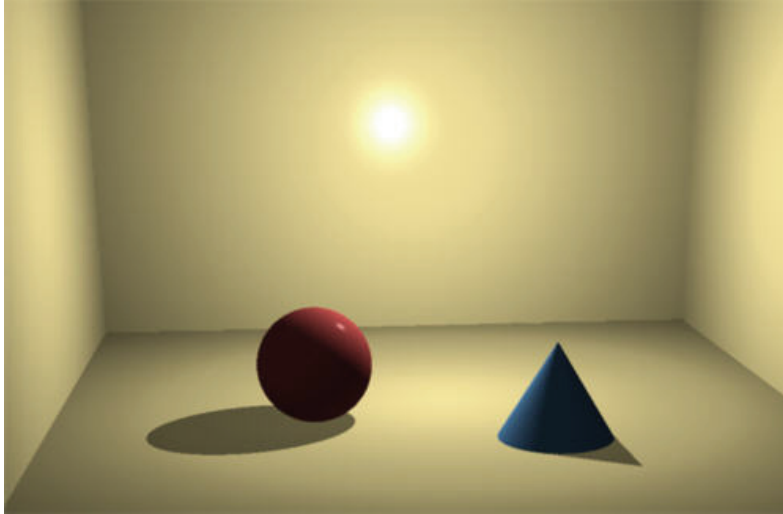
Setting Bulb Light Characteristics

To insert a bulb light and set its characteristics:

1. To insert a bulb light, do one of the following:

    • Choose **Insert > Bulb**.

    • Drag the Bulb Light tool into your scene.

2. With the light selected, display the **Properties tray: General tab: Light panel**.
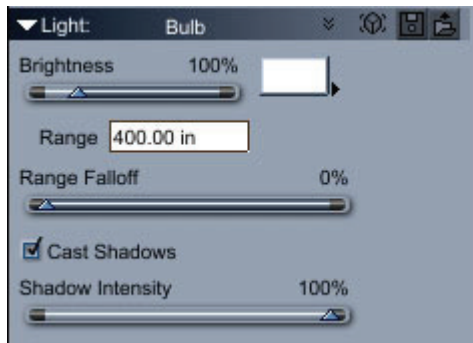
3. All the parameters interact with each other. For example, using a soft-colored light with a low brightness setting may generate a result similar to a darker-colored light with a higher brightness setting. However, different results are produced when you animate the individual parameters. Set the following properties as needed:

    • **Brightness**: Adjust the Brightness setting to increase or decrease the strength of the bulb light.

    • **Color chip**: Click to set the color of the light.

    • **Range**: The distance from the light itself to the point where the light disappears. The units of measure are those currently set in file preferences.

- **Range Falloff**: Drag the slider to set a percentage. This setting determines how the brightness of the light diminishes toward the edge of its range. A falloff of 10% means that the light has full intensity from the source through 90% of its range, then decreases linearly to the end of the range.

- **Cast Shadows**: Check this option to generate shadows with the light. Uncheck the option if you do not want the light to cast shadows.

- **Shadow Intensity**: Adjust the slider to increase or decrease the strength of the shadow.

Controlling a Bulb Light Directly

On a bulb light, brightness is the only control available with Direct Manipulation.

To change a bulb light's brightness:

1. Select the bulb light that you want to change.

2. Click the Direct Manipulation icon in the Properties tray: General tab: Bulb Light panel.
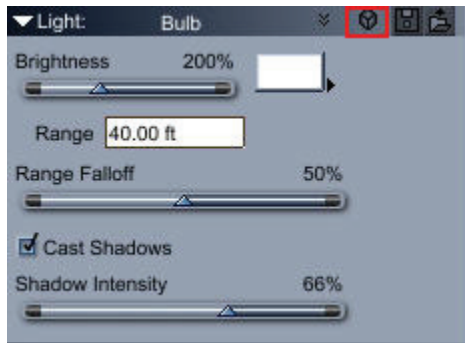
3. Drag the handle on the wire that extends sideways from the bulb light. Drag inward to decrease brightness or outward to increase brightness.
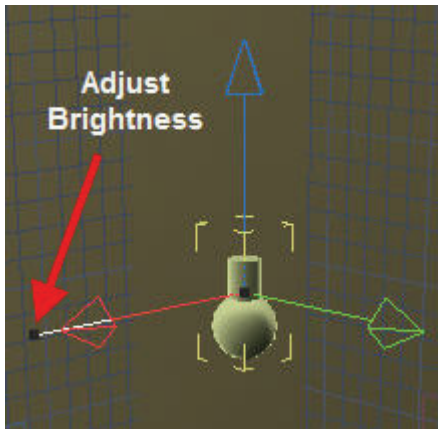
 TIP! If you have the **Properties tray: General tab: Light panel** displayed, you can see the effect of your changes in the preview and settings.

Distant Lights

A **Distant Light** source's position is conceptually outside of the scene's universe, although you may choose to place the shape representing its direction inside the working box. The rays are parallel as they enter your scene and they affect all objects. An example of distant light is the way the sun lights the earth.
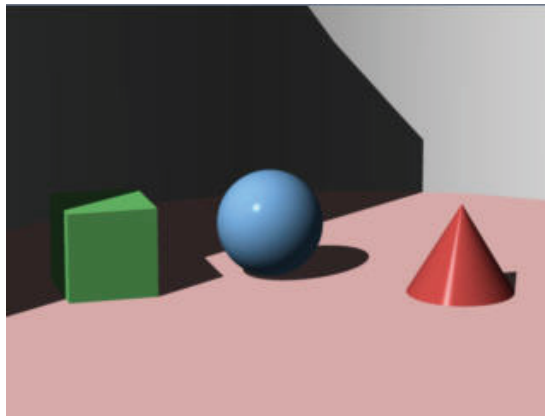
Setting Distant Light Characteristics

To insert a distant light and set its characteristics:

1. To insert a distant light, do one of the following:

    - Choose **Insert > Distant**.

    - Drag the Distant Light tool into your scene.

2. With the light selected, display the **Properties tray: General tab: Light panel**.
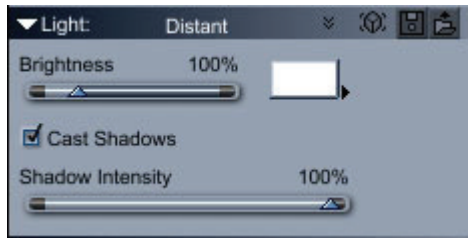
3. Set the following properties as needed:

- **Brightness**: Adjust the Brightness setting to increase or decrease the strength of the distant light.

- **Color chip**: Click to set the color of the light.

- **Cast Shadows**: Check this option to generate shadows with the light. Uncheck the option if you do not want the light to cast shadows.

- **Shadow Intensity**: Adjust the slider to increase or decrease the strength of the shadow.

Setting Distant Light Direction

No matter where you position a distant light in your scene, it behaves as if the source is a distant object, sending parallel rays into your scene. To change the direction of a distant light, change the orientation of the light.

To set the direction of a distant light:

1. Select the distant light arrow.
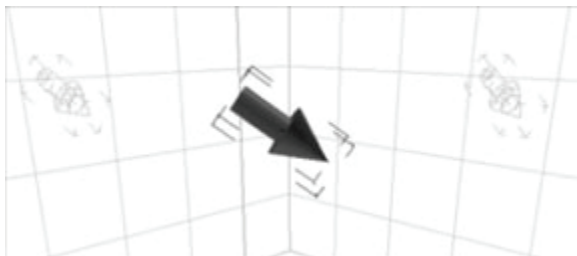
2. Use the Rotate tool to point the arrow in the direction in which you want the rays to shine.

TIP! As with ambient light, you can use a distant light to soften the shadows cast by other light sources. Position a distant light above your scene, then experiment with light settings to diffuse the shadows from the other lights.

Moon Light

Moon light is used with Sky or Realistic Sky.

To set moon light characteristics:

1. To insert a moon light, do one of the following:

   - Choose **Insert > Moon Light**.

   - Drag the Moon Light tool into your scene.

   ![Moon Light tool icon]

2. With the light selected, display the **Properties tray: General tab: Light panel**.

   ![Moon Light panel showing Brightness 50% with Note: This light is to be used with the Sky or Realistic Sky atmosphere.]

3. Adjust the slider to increase or decrease the brightness of the moon light.

Sun Light

Sun light is used with Sky or Realistic Sky.

To set sun light characteristics:

1. To insert a sun light, do one of the following:

   - Choose **Insert > Sun Light**.

   - Drag the Sun Light tool into your scene.

   ![Sun Light tool icon]

2. With the light selected, display the **Properties tray: General tab: Light panel**.

   ![Sun Light panel showing Brightness 100% with Note: This light is to be used with the Sky or Realistic Sky atmosphere.]

3. Adjust the slider to increase or decrease the brightness of the sun light.

Tube Lights

To insert a tube light and set its characteristics:

1. To insert a tube light, do one of the following:

   - Choose **Insert > Tube Light**.

   - Drag the Tube Light tool into your scene.

   ![Tube Light tool icon]

2. With the light selected, display the **Properties tray: General tab: Light panel**.
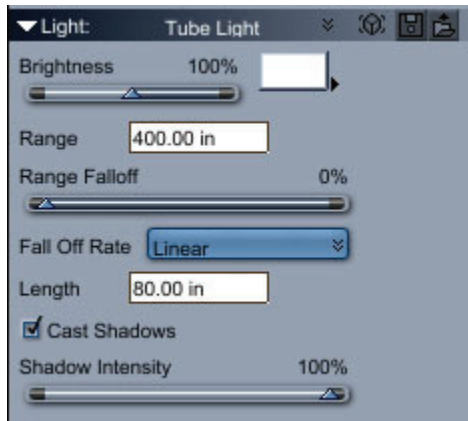
3. Set the following properties as needed:

- **Brightness**: Adjust the Brightness setting to increase or decrease the strength of the tube light.

- **Color chip**: Click to set the color of the light.

- **Range**: The distance from the light itself to the point where the light disappears. The units of measure are those currently set in file preferences.

- **Range Falloff**: Drag the slider to set a percentage. This setting determines how the brightness of the light diminishes toward the edge of its range. A falloff of 10% means that the light has full intensity from the source through 90% of its range, then decreases linearly to the end of the range.

- **Fall Off Rate**: Select how the light goes from full brightness to zero: **Linear**, **Distance Squared**, or **Logrithmic**.

- **Length**: Enter the length of the tube light.

- **Cast Shadows**: Check this option to generate shadows with the light. Uncheck the option if you do not want the light to cast shadows.

- **Shadow Intensity**: Adjust the slider to increase or decrease the strength of the shadow.

Shape Lights

To insert a shape light and set its characteristics:

1. To insert a Shape light, do one of the following:

2. Choose **Insert > Shape Light**.

3. Drag the Shape Light tool into your scene.

4. With the light selected, display the **Properties tray: General tab: Light panel**.
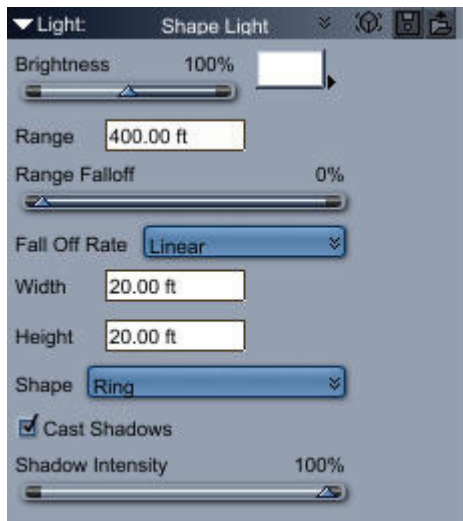
5. All the parameters interact with each other. For example, using a soft-colored light with a low brightness setting may generate a result similar to a darker-colored light with a higher brightness setting. However, different results are produced when you animate the individual parameters. Set the following properties as needed:

    -   **Brightness**: Adjust the Brightness setting to increase or decrease the strength of the shape light.

    -   **Color chip**: Click to set the color of the light.

    -   **Range**: The distance from the light itself to the point where the light disappears. The units of measure are those currently set in file preferences.

    -   **Range Falloff**: Drag the slider to set a percentage. This setting determines how the brightness of the light diminishes toward the edge of its range. A falloff of 10% means that the light has full intensity from the source through 90% of its range, then decreases linearly to the end of the range.

    -   **Fall Off Rate**: Select how the light goes from full brightness to zero: **Linear, Distance Squared, or Logrithmic.**

    -   **Width: Sets the width of the shape light.**

    -   **Height: Sets the height of the shape light.**

    -   **Shape: Select the shape that you want to use for the shape light. Choices are Ellipse, Rectangle, and Ring.**

    -   **Cast Shadows: Check this option to generate shadows with the light. Uncheck the option if you do not want the light to cast shadows.**

    -   **Shadow Intensity: Adjust the slider to increase or decrease the strength of the shadow.**

Anything Glows

Anything Glows is a light that allows you to use any object as a source of light. Anything Glows helps you create glowing objects that really cast light, realistic lighting effects, and it allows you to control many lights at the same time using Master Lights.
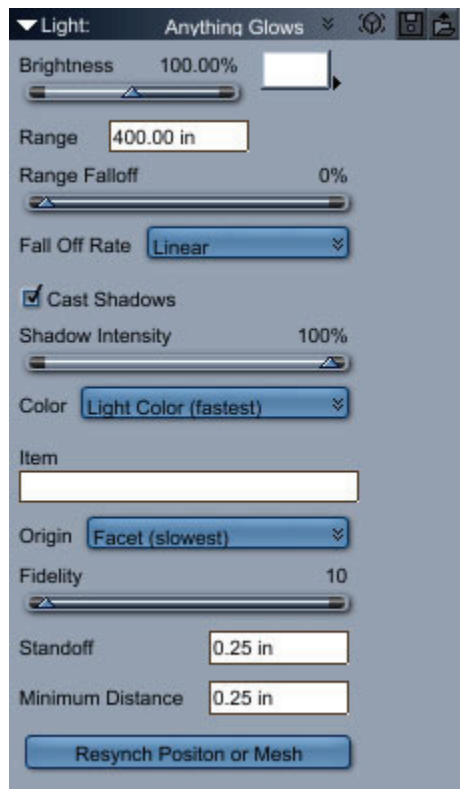
To make any object glow:

1. Create a scene with the desired number of objects. The glow from Anything Glows affects surrounding objects.

2. Do one of the following:

   - Choose **Insert > Anything Glows**.

   - Drag the Anything Glows tool into your scene.

3. Set Anything Glows properties as follows:

   - **Brightness**: Controls the intensity of the light.

   - **Color chip**: Controls the color of the light when the Color option is set to Light Color.

   - **Range**: Controls how far away the light illuminates.

   - **Range Falloff**: Controls how the brightness decreases at the end of its range, Fall Off Rate controls how it goes from full brightness to zero.

   - **Fall Off Rate**: Options are **Linear**, **Distance Squared**, and **Logarithmic**. The image below shows the three falloff rates with Range at 10 feet and Range Falloff at 70.

Linear   Distance Squared   Logrithmic
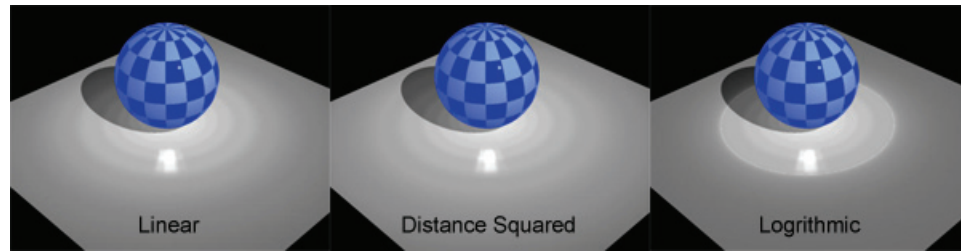
- **Cast Shadows**: Turns shadows on and off. Anything Glows 2 doesn't require that the Cast Shadows option be turned off for Anything Glows Items, but some scenes look better with it that way.

- **Shadow Intensity**: Sets how dark the shadows are.

- **Color**: Determines what drives the light's color. Options are **Light Color (fastest)**, **Object Color**, or **Object Glow**. If you are far from the object Anything Glows uses, set the Color to Light Color instead of Object Color or Object Glow. The image below shows color settings from left to right as Light Color (when set to white), Object Color (the checkerboard shader), and Object Glow (a blue gradient).

Light Color   Object Color   Object Glow
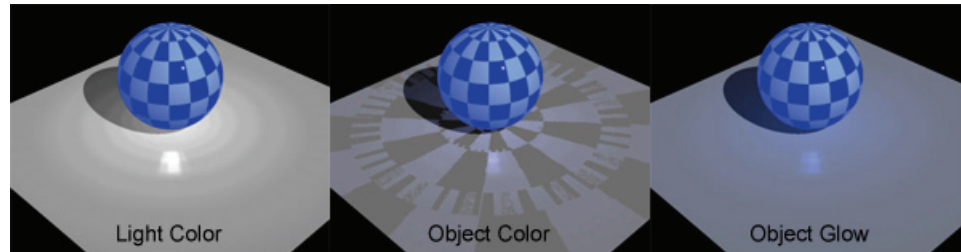
- **Item**: Enter the name of the object that you want to use as the light source. If the name of the object is not unique in the scene you may get unpredictable results. If Anything Glows cannot match the name it will not emit any light.

- **Origin**: Controls the starting point of the light rays. Options are **Bounding Box (fastest)**, **Vertex**, and **Facet (slowest)**. If you are far away from the object Anything Glows is using, turn Origin to Bounding Box or Vertex. Turn it back to Facet when you get close enough to see the details. On rough surfaces (heavy bump) or surfaces with lots of color changes, you often won't be able to tell the difference between an Origin of Facet or Vertex. If your light only illuminates those kinds of surfaces, use Vertex, it's much faster.

- **Fidelity**: Controls the amount of detail that will be used when Origin is set on Vertex or Facet. Higher Fidelity settings allow for greater mesh detail. Set the Fidelity to the lowest setting that give the results you need. The lower the Fidelity, the faster Anything Glows renders.

- **Standoff:** Specifies how far away from the mesh the light ray starts. This keeps the Anything Glows light ray from intersecting the rendered mesh and causing unwanted shadows.

- **Minimum Distance**: Controls how far away from the mesh a point must be to be illuminated. This keeps Anything Glows from illuminating its own mesh.

- **Resynch Position or Mesh**: Tells Anything Glows you have moved the object or changed the geometry. If you render, make a change, and then want to render again, press this button to tell Anything Glows to synch up with your object.

Tips and Tricks

- Gels are a great way to control Anything Glows output without changing the shader's color or glow. Putting a slightly blurred version of the objects glow map in Anything Glows' gel is a great way to simulate monitors or TV screens.

- Use Light Smart Sampling on your render settings. Anything Glows is about three times faster with Light Smart Sampling on and most of the time you won't be able to tell the difference in the resulting image.

Setting up Environmental Lighting

Environment Lighting works by breaking up the mesh into little chunks and assigning an Anything Glows light to each chunk. The plane primitive, which only has two facets regardless of the Fidelity, makes a poor emitter. To create a flat emitter, build a grid in the Vertex modeler.

To activate Environment Lighting:

1. Select an object in the 3DView.

2. Click **Edit > Environment Lighting**.

3. Set the following options:

    - **Mesh Fidelity** controls how detailed a mesh Anything Glows will use when Origin is set on Vertex or Facet. The higher the Mesh Fidelity, the more detailed the mesh.

    - **Quality** specifies how many lights will be used. At 1 Anything Glows will use 1 light for every 10 facets, at 6 Anything Glows will use 1 light for each facet.

    - **Color** determines what drives the light's color.

4. Click OK to apply the settings.

Setting Master Lights Characteristics

To set Master Lighting options:

1. Choose **Edit > Master Light**. The Master Light dialog appears.
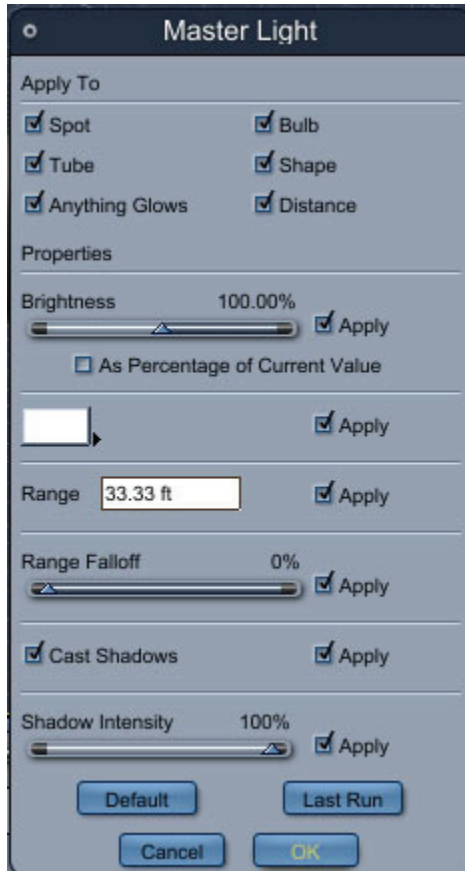
2. Use the **Apply To** section to control the types of lights that will be changed. Check or uncheck Spot, Bulb, Tube, Shape, Anything Glows, or Distance as needed.

3. In the **Brightness** section:

   - **Apply:** Enables the brightness controls.

   - **Brightness**: Use the slider to control the intensity of the light.

   - **As Percentage of Current Value:** Check this option to multiply the light's current value times the Brightness setting to get the light's new value.

4. To control the color of the light, check the **Apply** option near the color chip. Then click the color chip to select the color of the light .

5. To control how far away the light illuminates, check the **Apply** option in the **Range** section and enter the desired distance in the Range field.

6. To control how the brightness decreases at the end of the range, check the **Apply** button in the **Range Falloff** section, and adjust the slider to obtain the desired amount of falloff.

7. Check the **Cast Shadows** option to turn shadows on, or uncheck to turn shadows off. The **Apply** option enables or disables this feature.

8. Use the **Shadow Intensity** slider to control the darkness of the shadow. Check the **Apply** button to enable the feature.

9. Click the Default button to restore the Master Light options to the default values.

10. Click the **Last Run** button to restore the settings of the Master Light to the values stored the last time you clicked OK.

Shadow Options

Objects block light and, therefore, cast shadows on other surfaces. For each light, you can choose between two methods to calculate shadows: Raytraced shadows and Shadow buffer shadows. Each of those methods allows you to create two types of shadows:

- **Hard shadows** have sharp edges. The transition from complete shadow to full light is immediate.
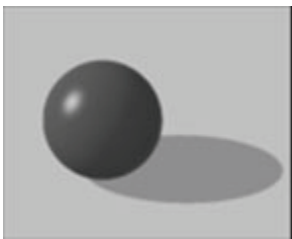
- **Soft shadows** create an obscurity at the shadow edge. The shadow has a soft transition between complete shadow and full light.
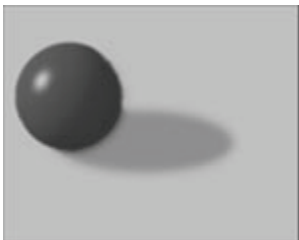
To set shadow options for a light:

1. Select the light you want to set.

2. On the **Properties tray: Effects tab: Shadow panel**, choose the method you want to calculate your shadows.
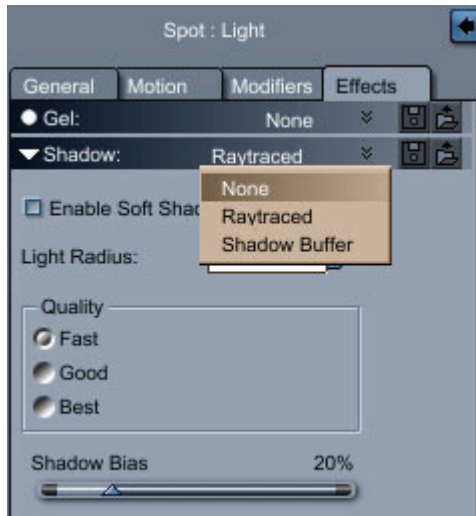
Enabling and Disabling Shadows on Objects

By default, all objects that are not transparent cast shadows. There may be an instance where you don't want an object to cast shadows. Carrara allows you to turn Shadow Casting on and off for individual objects.

To set shadow casting for the selected object:

1. In the Assemble room, select the object on which you want to change the shadow properties.

2. In the **Properties tray: General tab: General panel**:

    - Check the **Casts Shadows** option to turn shadow casting on for the selected object.

    - Uncheck the **Casts Shadows** option to turn shadow casting off for the selected object.

Raytraced Shadows

The Raytraced shadow is the default. It is the most accurate method for calculating shadows. It will create very nice soft shadows but tends to be slow with this type of shadow.

You can adjust the following settings:

- **Enable Soft Shadows** will create soft shadows instead of the simple hard shadows (Note that soft shadows take a quite a bit more time to render).

- **Light Radius** only affects soft shadows. It determines how big the light source is and thus how soft the shadows will be. Note that you need to take into account the distance of the light to the scene. A light that is far away will only create soft shadows if the radius is very big (this is especially true for distant lights that are always far from the scene).

- **Quality** only affects soft shadows. It determines how many rays are used to evaluate the shadows. It recommended to use the lowest quality setting that works with your scene and to only increase it if there is a visible artifact.

Shadow Buffers

Shadow buffers should be mostly used to create soft shadows. They are usually much faster than raytraced soft shadows, especially for complex scenes. However they are not as accurate as Raytraced Soft Shadows and do not let light go through objects (even when the Light through transparency option of the renderer is checked).

To set the Shadow buffers options:

1. Select a light.

2. On the **Properties tray: Effects tab: Shadow panel**, choose Shadow Buffer.

3. Set shadow buffers controls.

   - **Resolution** determines the quality of the shadow effect. A higher resolution uses more memory.

   - **Blur Size** increases the area of softness. You need a higher setting here if you have set a high Resolution.

   - **Bias** controls the placement of the shadow. Increase this setting if your objects are self-shadowed. Decrease this setting if shadows appear too far away from the objects themselves.

Positioning and Aiming Lights

You can position lights as you would any object in your scene:by dragging a light or its projection, or numerically by using the controls in **Properties tray: Motion tab: Transform panel**.

Note You can also use the Direct Manipulation controls to move and aim lights. Refer to "Controlling Lights Directly."

For information on positioning and orienting lights numerically, refer to "Arranging Objects."

Positioning Lights by Dragging

You can drag lights as you would any other object in your scene.

To move a light:

1. Select the light.

2. Select the Selection tool.

3. Drag the light to where you want it.

   - The light moves parallel to the active plane.

   - To move the light perpendicular to the active plane, hold the Option/Alt key while you drag.

- You can also drag one of the light's projections to position the light on that plane.
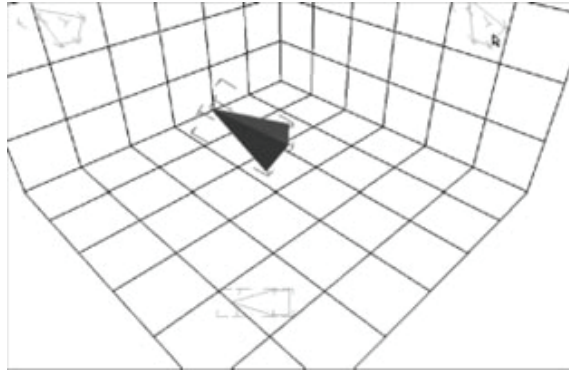
Pointing a Light at an Object

Carrara provides other methods of aiming lights, including the Point At command, the Point At modifier, and the Direct Manipulation controls.

TIP! To direct a light to a particular area in your scene, create a temporary object for the light to point at, then delete the object. Remember that the light points at the hot point of the selected object.

The Point At Command

The Point At command lets you point a light directly at an object.

To point a light at an object:

1. Hold down the Shift key and select both the light and the object you want to point it at. You may select multiple lights, but only one object.

2. Choose **Edit > Point At**. Carrara reorients the light to point at the hot point of the selected object.

The Point At command does not link the light to the object. That is, if you move either element, the light no longer points at the object. If you want to maintain the distance between a light and the object it points at, group them. For more information, refer to "Grouping Objects." If you want the light to follow the object, apply the Point At modifier, described in the next section.

The Point At Modifier

The Point At modifier aims a light at an object. If the object moves, the light automatically reorients itself to point at the object in its new position.

To have a light follow an object:

1. Select the light you want to use.

2. On the **Properties tray: Modifiers tab**, click the + sign to expand the list and choose Point At. The Point At controls appear.
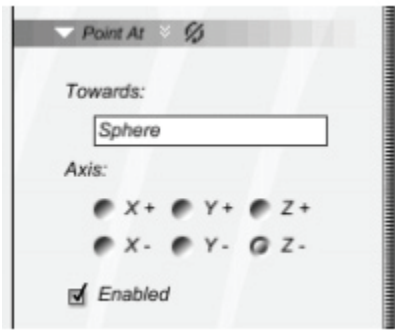
3. Adjust the Point At controls as appropriate:

- **Towards** Enter the name of the target object:the one you want the light to follow. Enter the name exactly as it appears in the Hierarchy area of the Sequencer. (The text field is case-sensitive.)

- **Axis** Leave this option as is. The axis option sets which axis of the object should point at the target. For a light, you only want to use the axis that emits the light.

- **Enabled** Check this box to turn on the Point At modifier.

4. When you move the target object, the light reorients to point at it.

Controlling Lights Directly

The Direct Manipulation controls provide an on-screen description of a light's aim and properties. You can drag a light's control handles to change light properties.

To display the Direct Manipulation controls on a light:

1. Select a light.

2. Display the **Properties tray: General tab: Light panel**.

3. Click the Direct Manipulation button to display the control handles on the selected light.

The controls appear as a set of wires with handles at certain points. Each handle represents a control you can drag to set properties.

The Direct Manipulation controls for a spot light are shown below. The appearance and number of these controls depend on the type of light you have selected.
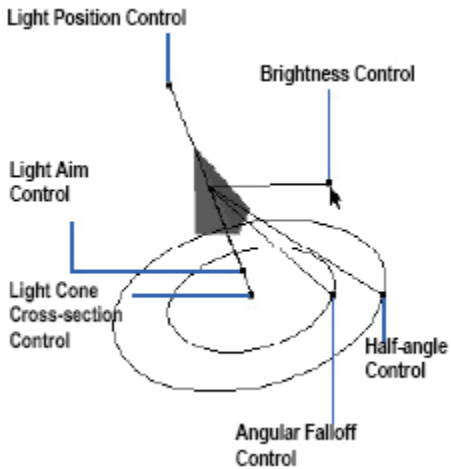
Using Gels

You can place an image as a mask or transparency in front of a light to project complex patterns and images on your scene. The image you place in front of the light is called a **gel**. Gels are typically used on spot lights.

If the gel is a 1-bit image (see image below), it becomes a mask. White regions of the mask transmit the light, and black regions block it. Bitmap gels can create intricate effects, such as the shadow of a chain-link fence, or the dappled shade under a tree.

Grayscale or color images create transparencies when used as gels. Their image is projected into your scene, just as a slide projector sends an image across the room. With a color image, you can achieve many effects, such as sunlight filtering through a stained glass window.
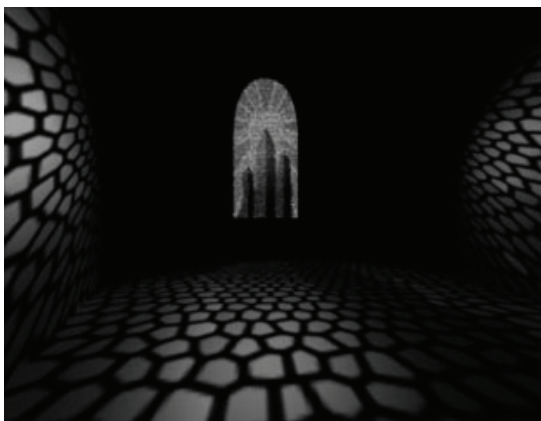
Blinds, Gradient, and Blinds, Gradient, and Formula are built-in gels. You may also load a texture map or movie to use as a gel. An 8-bit gel will probably provide all the color you need.

As you adjust the settings for these gels, a preview displays at the right. Specific blend controls are described later in this chapter.

NOTE! Gels have no effect on distant lights.

To place a gel on a light:

1. Select a light. Gels are generally used on spot or bulb lights. Your choice depends on the effect you want.

2. On the **Properties tray: Effects tab: Gel panel**, choose a type of gel:

   - **None** is the default. The light has no gel.

   - Blinds Options

   - Formula Options

   - Gradient Options

   - Map Options

Blinds Options

The **Blinds** option creates horizontal or vertical stripes such as venetian blinds or prison bars.
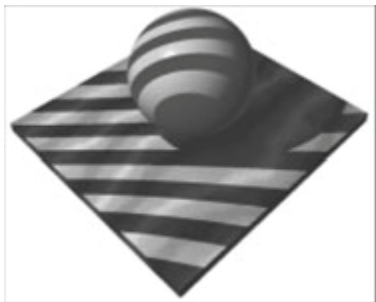
To set blinds options:

1. On the **Properties tray: Effects tab: Gel panel**, choose Blinds.
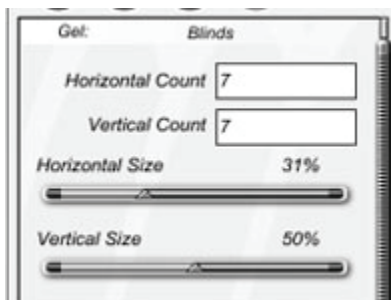
2. Set the Blinds controls:

   - **Horizontal Count** sets the number of horizontal slats.

   - **Vertical Count** sets the number of vertical slats.

- **Horizontal Size** sets the width of the vertical slats. The width is expressed as a percentage of the gel frame.

- **Vertical Size** sets the height of the vertical slats. The height is expressed as a percentage of the gel frame.

Formula Options

The **Formula** option uses a mathematical formula to create colors and patterns in the gel. You can devise your own formula to create a new pattern.
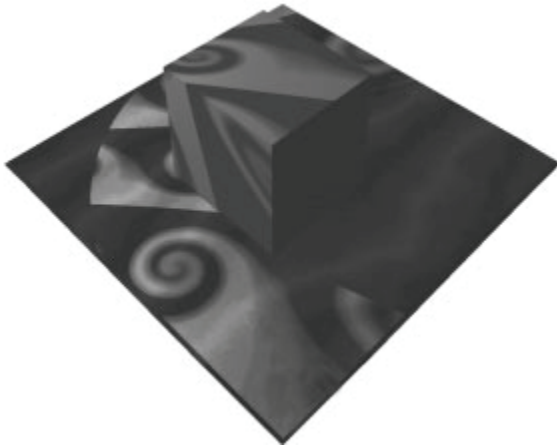
To select a formula for the gel:

1. On the **Properties tray: Effects tab: Gel panel**, choose Formula.
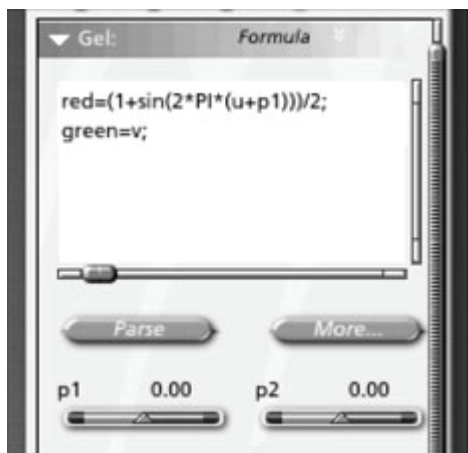
2. If the current formula uses Parameter values, you can adjust them to change the formula result.

3. If you want to edit the formula or create your own, click More. The formula editor appears.

Carrara uses the Formula Editor in several places. Use of the editor is common, but the type of formula you're creating determines the valid input and output variables. Refer to "Using Formulas in Carrara" for details on using the Formula Editor.

Gradient Options

The **Gradient** option uses a blend of two colors as a gel. A gradient gel can be vertical or circular.
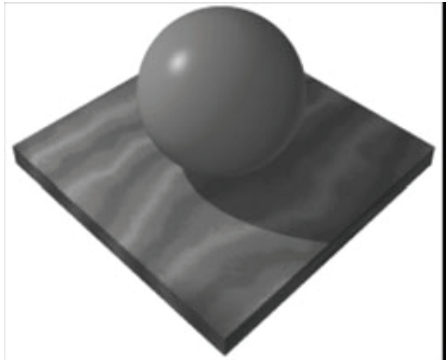
To set gradient options:

1. On the **Properties tray: Effects tab: Gel panel**, choose Gradient.
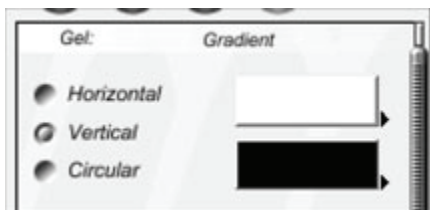
2. Set the Gradient controls: **Horizontal**, **Vertical**, or **Circular** sets the gradient pattern. Click the top color chip to select a start color, and the bottom color chip to select an end color.

    *   For a **Horizontal** gradient pattern, the start color appears on the top and the end color on the bottom.

    *   For a **Vertical** gradient pattern, the start color appears on the left and the end color on the right.

    *   For a **Circular** gradient pattern, the start color appears on the inside and the end color on the outside.

Map Options

The **Map** option uses a bit-mapped image file as the gel. This is the best choice when you want specific images in the gel.
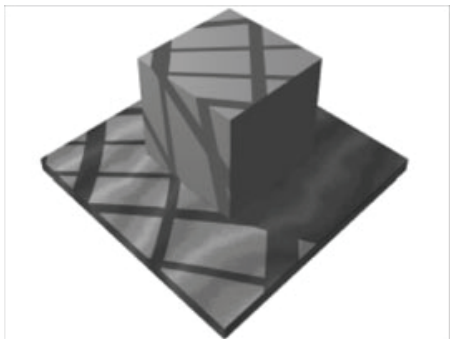
You can use any 2D graphics program to create an image for the gel. You might scan a photograph and use filters to accentuate or nominalize some aspects of the image. The gel does not need to be high-resolution. If you intend to use this gel on a bulb light, create the image at an aspect ratio of 2:1. If you are going to use this gel on a spot or distant light, use an aspect ratio of 1:1.

Save the image file in a format that Carrara supports and in a convenient folder.

For a list of the image file types that Carrara supports, refer to "Importing 3D Objects."

To load a map image as a gel:

1. On the **Properties tray: Effects tab: Gel panel**, choose Map.
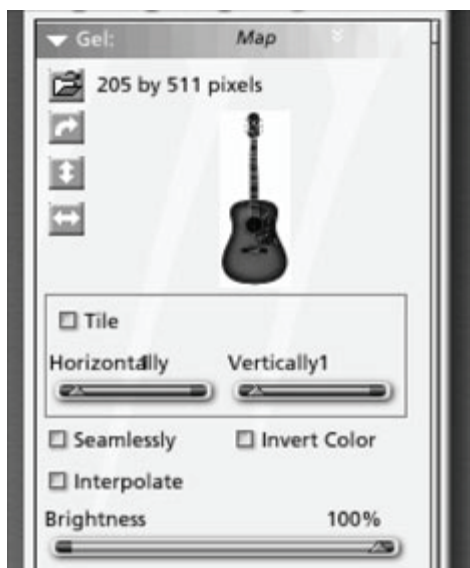
2. Click the Folder icon to display the Open dialog.

3. Use the Open dialog to locate and open the map image you saved. A preview appears on the Gel panel with the image's dimensions given in pixels.

To set map controls:

1. Click the directional buttons next to the Folder icon to change the image's orientation.

2. Choose among the following options:

   - **Tile** makes the image repeat. Use the Horizontally and Vertically sliders to set the number of tiles in each direction.

   - **Seamlessly** smoothes the transitions between tiles.

   - **Invert Color** inverts the image's colors.

   - **Interpolate** smoothes out textures for zooming. This option works best with low resolution textures, and may slow the rendering.

   - **Brightness** sets the image's brightness.

NOTE! You can use Map options to place movies. To do this, load movie images as a gel, just as you would load map images.

Light Effects (Standard, Pro)

You can add effects to your scene that simulate the results obtained from photographic cameras and special lighting. You apply light effects on a light-by-light basis.

NOTE! Light effects are post-render filters. Although they simulate adjustments to your lights and cameras, they do not actually change any of your light or camera settings.

Using the Preview

All the light effects dialogs have a rendered preview that helps you to adjust the settings of a specific effect.

You can use the **Zoom** and **Pan** tools at the bottom of the preview to get a better view of the details of the image.

Clicking on the **Render** button after zooming in will increase the quality of that part of the image you are viewing. To display the full image again, click on the **Reset View** button.

When you check the **Auto Update** box, the preview is re-rendered each time you adjust a control, otherwise you will need to click the **Render** button after changing one or more controls.

CrossScreen

The CrossScreen effect adds a glow and star-like rays to every visible light source.

The light source must be visible from the camera to produce an effect. It must be within the production frame and not hidden behind or inside an object.

To set CrossScreen options:

1. Select the light to which you want to apply an effect.

2. In the **Properties tray: Effects tab: Light Effects panel**, check the Enable CrossScreen box, then click Edit. The CrossScreen dialog appears.
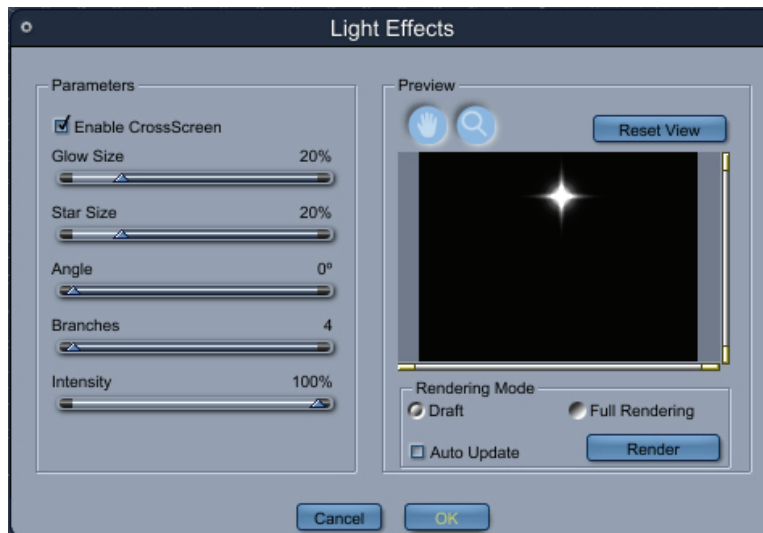
3. Set the CrossScreen options:

- **Glow Size** sets the diameter of the glow.
- **Star Size** sets the star radius.
- **Angle** rotates the stars around the light source center.
- **Branches** sets the number of rays.
- **Intensity** sets the effect intensity.

Glow (Lights)

Glow simulates a photographic filter by adding a glow around every visible light source. The effect of the glow depends on the light's position, color, and intensity.
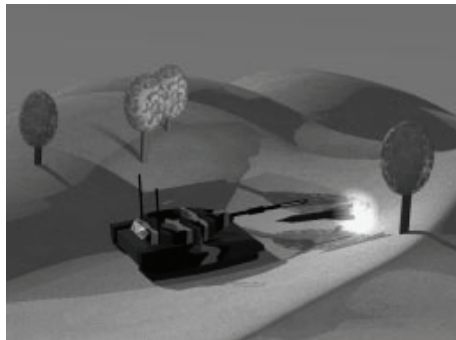
The light source must be visible from the camera to produce a glow. It must be within the production frame and not hidden behind or inside an object.

To set Glow options:

1. Select the light to which you want to apply an effect.

2. In the **Properties tray: Effects tab: Light Effects panel**, check the Enable Glow box, then click Edit. The Glow dialog appears.
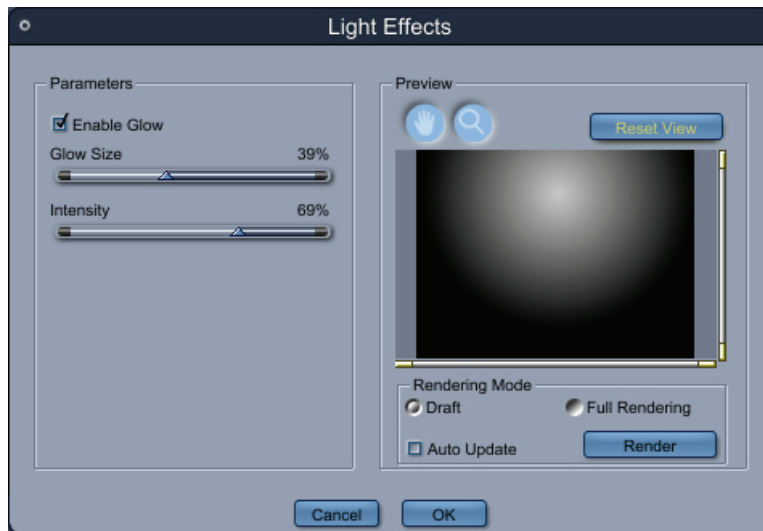
3. Set the Glow options:

    *   **Glow Size** sets the diameter of the glow.

    *   **Intensity** sets the strength of the glow.

Nebula

The Nebula effect adds multicolored streaks around every visible light source.
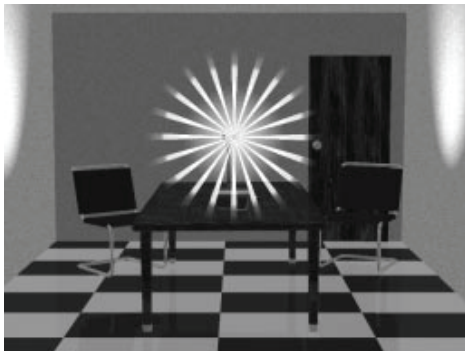
The light source must be visible from the camera to produce a nebula effect. It must be within the production frame and not hidden behind or inside an object.

NOTE! Render on a dark background to see the Nebula effect.

To set Nebula options:

1. Select the light to which you want to apply an effect.

2. In the **Properties tray: Effects tab: Light Effects panel**, check the Enable Nebula box, then click Edit. The Nebula dialog appears.
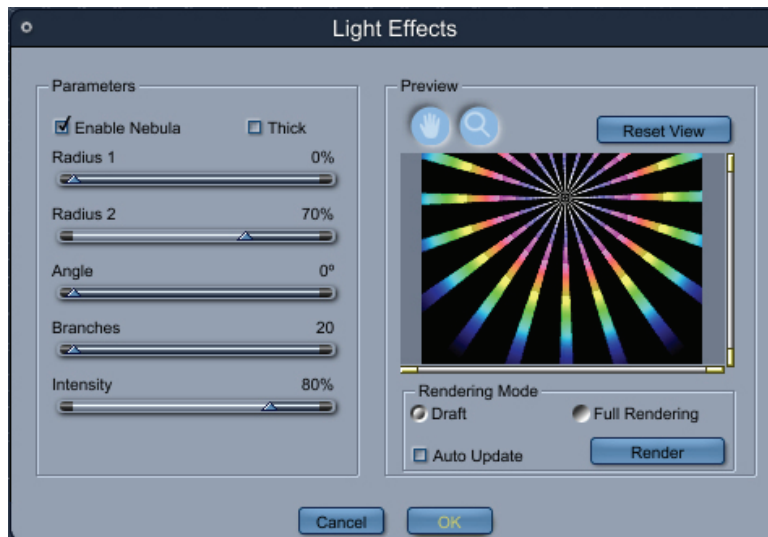
3. Set the Nebula options:

- **Radius 1** sets the streaks' starting radius.

- **Radius 2** sets the streaks' ending radius.

- **Angle** rotates the streaks around the light source center.

- **Branches** sets the number of streaks.

- **Thick** makes the streaks thicker.

- **Intensity** sets the streak intensity.

Pulsator

Pulsator adds dotted streaks around every visible light source.

The light source must be visible from the camera to produce a pulsator effect. It must be within the production frame and not hidden behind or inside an object.

To set Pulsator options:

1. Select the light to which you want to apply an effect.

2. In the **Properties tray: Effects tab: Light Effects panel**, check the Enable Pulsator box, then click Edit. The Pulsator dialog appears.
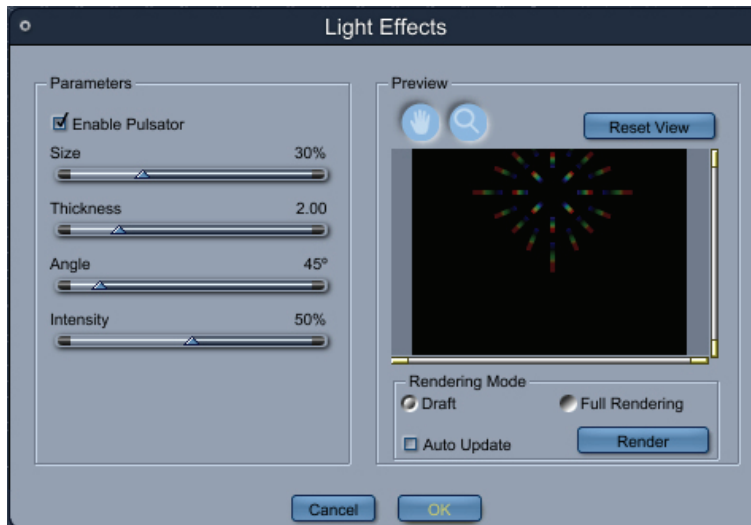
3. Set the Pulsator options:
4. **Size** sets the streak radius.

    o   **Thickness** sets the streaks' thickness.

    o   **Angle** rotates the streaks around the light source center.

    o   **Intensity** sets the streak intensity.

Stars

Stars adds a star around every visible light source.
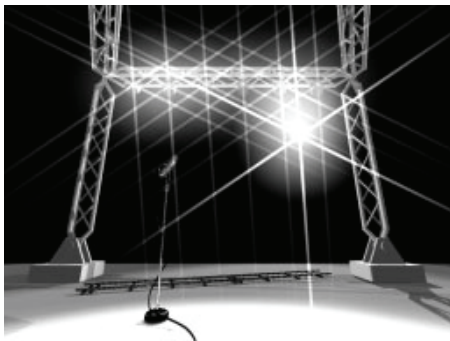
The light source must be visible from the camera to produce a star effect. It must be within the production frame and not hidden behind or inside an object.

To set Stars options:

1. Select the light to which you want to apply an effect.

2. In the **Properties tray: Effects tab: Light Effects panel**, check the Enable Stars box, then click Edit. The Stars dialog appears.
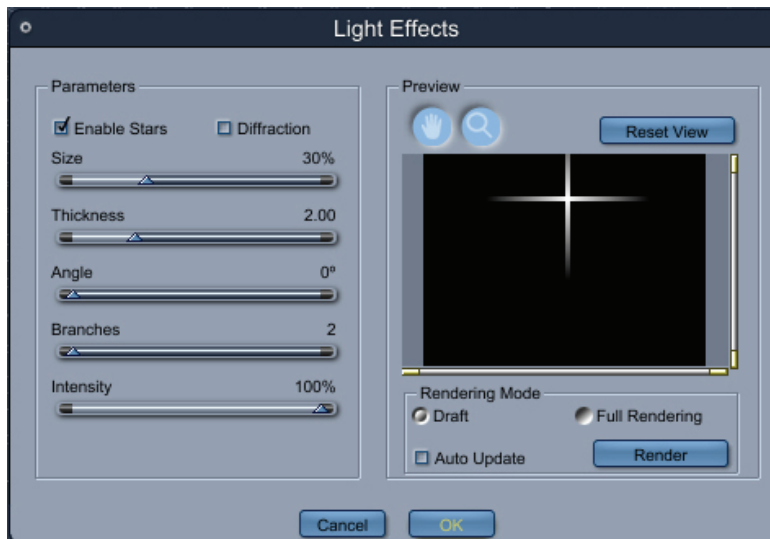
3. Set the Stars options:

- **Size** sets the stars' radius.

- **Thickness** sets the stars' thickness.

- **Angle** rotates the stars around the light source center.

- **Branches** sets the number of rays.

- **Diffraction** splits the stars into rainbow colors.

- **Intensity** sets the stars' intensity.

VarioCross

The VarioCross effect adds two streaks around every visible light source. Use the VarioCross effect to simulate the light streaks produced by distant bright lights.

The light source must be visible from the camera to produce a VarioCross effect. It must be within the production frame and not hidden behind or inside an object.

NOTE! Render on a dark background to see the VarioCross effect.

To set VarioCross options:

1. Select the light to which you want to apply an effect.

2. In the **Properties tray: Effects tab: Light Effects panel**, check the Enable VarioCross box, then click Edit. The VarioCross dialog appears.
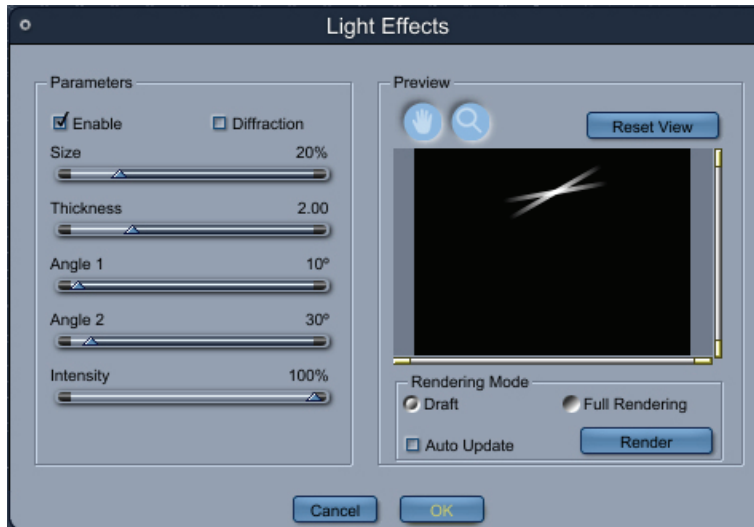
3. Set the VarioCross options:

    - **Size** sets the streak radius.

    - **Thickness** sets the streaks' thickness.

    - **Angle 1** rotates the first streak around the light source center.

    - **Angle 2** rotates the second streak around the light source center.

    - **Diffraction** splits the streaks into rainbow colors.

    - **Intensity** sets the streaks' intensity.

Light Cone

In the real world, light rays are visible when the atmosphere contains small particles of dust or vapor that diffuse light. The Light Cone effect simulates the interaction between light from a spot light and smoke, fog, and dust.

The Light Cone effect post-processes the rendered picture by adding visible light beams from spot lights. The visible light beams accurately describe the half angle of each spot light in the scene.

To set Light Cone options:

1. Select the spot light to which you want to apply an effect.

2. In the **Properties tray: Effects tab: Light Cone panel**, check the Enable box, then click Edit. The Light Cone dialog appears.
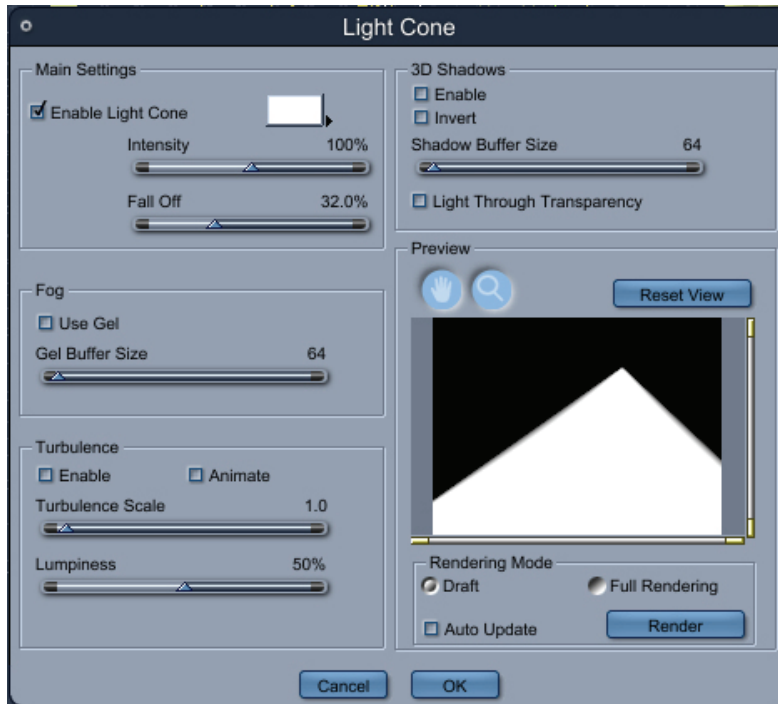
3. Set the Main Settings:

   - **Color** - To change the color of the fog, click on the color chip. Carrara opens the system color picker so you can choose a color. The color of the visible light cone results from the interaction of the light's color and the fog color. Yellow light in a white fog produces a yellowish light cone. In a blue fog, yellow light produces a green cone.

   - **Intensity** sets the strength of the light cone effect.

   - **Fall Off** - This setting determines how the brightness of the light diminishes toward the edge of its range.

4. Set the Fog options:

   - **Use Gel** allows you to use any gel on the lights in the light cone effect. Enable this option only if you've applied gels to your spot lights.

   - **Gel Buffer Size** sets the quality of the gel effect in the light cone. Drag to the right to increase quality of the gel in the light cone.

   NOTE! To render textured light rays, the filter pre-processes and stores buffers for each spot light in your scene. High buffer values produce better results, but are costly in rendering time and memory requirements. Don't use values higher than 200 unless you have a powerful computer and are working on high-resolution pictures. Always start with a small value.

5. Set the Turbulence options:

- **Enable** produces swirls in the fog medium.

- **Animate** makes the fog medium change over time. This option slows down the rendering. Enable it only when creating animations.

- **Turbulence Scale** sets the mean size, in inches, of the wreath of smoke. This slider ranges from 0.01 inches to 489 inches.

- **Lumpiness** sets the contrast level in the wreath of smoke.

6. Set the 3D Shadows options:

- **Enable** makes objects in the light cone cast shadows in the fog.

NOTE! 3D Shadows is a powerful but costly effect. The filter must pre-process and store buffers for each spot light in your scene. High values are costly in rendering time and memory requirements. Do not use values higher than 200 unless you have a powerful computer and you are working on high-resolution pictures. Always start with a small value.

- **Shadow Buffer Size** sets the quality of the 3D shadow effect in the light cone.

Light Sphere

The Light Sphere simulates the interaction between light from a bulb light and smoke, fog, and dust.
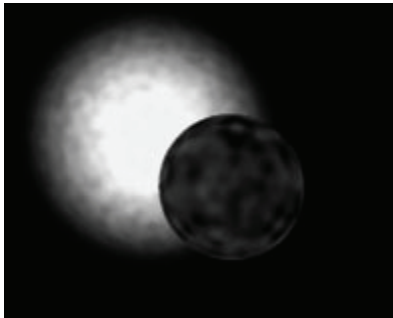
The Light Sphere effect post-processes the rendered picture by adding a light sphere around each bulb light.

To set Light Sphere options:

1. Select the bulb light to which you want to apply an effect.

2. In the **Properties tray: Effects tab: Light Sphere panel**, check the Enable box, then click Edit. The Light Sphere dialog appears.
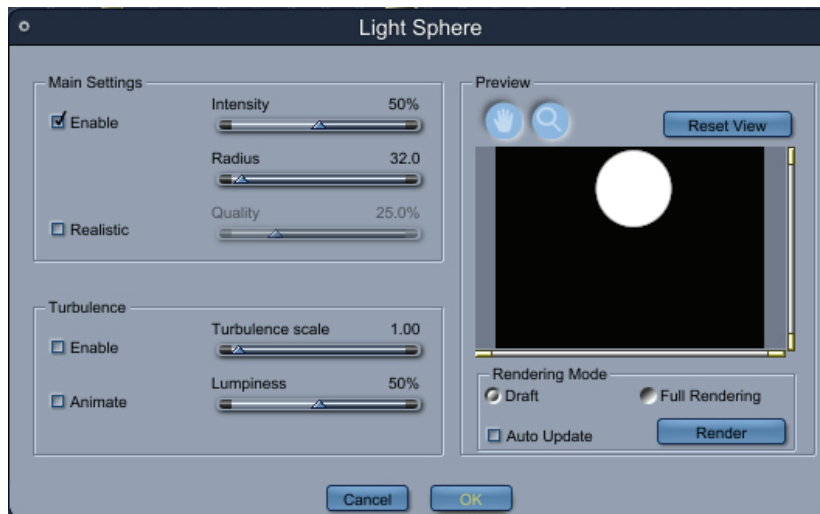
3. Set the Main Light Sphere options:

    - **Intensity** changes the strength of the light sphere effect.

    - **Radius** sets the size of the light sphere

    - **Realistic** TBD

    - **Quality** TBD

4. Set the Turbulence options:

    - **Enable** produces swirls in the fog medium.

    - **Animate** makes the fog medium change over time. This option slows down the rendering. Enable it only when doing animations.

    - **Turbulence scale** sets the mean size, in inches, of the wreath of fog.

    - **Lumpiness** sets the contrast level in the wreath of fog.

Lens Flare

Lens flares are reflections of a strong light source on the various components of the lens. Flare color and size depend on the kind of glass and shape of each lens component.

The Lens Flare effect adds lens flares to one or more light sources in your final rendered image, according to their positions, colors, and intensities.

To get a lens flare effect, a light source must be visible to the rendering camera. The light source must be within the production frame and not hidden behind or inside another object:even if it is a transparent

object. A spot light must be aiming toward the rendering camera to produce a lens flare. The camera must be within the light cone.

The Lens Flare effect uses three parameters to render the effect: Light source color, Light source intensity, and Lens Flare intensity.

To get brighter lens flares, use brighter light sources. It also helps to lower the ambient light and use darker backdrops.

NOTE! If you are rendering with a white background, you won't see the result of the lens flare effect. Select a dark color for the background.

To set Lens Flare options:

1. Select the light to which you want to apply an effect.

2. In the **Properties tray: Effects tab: Lens Flare panel**, click Edit. The Lens Flare dialog appears.
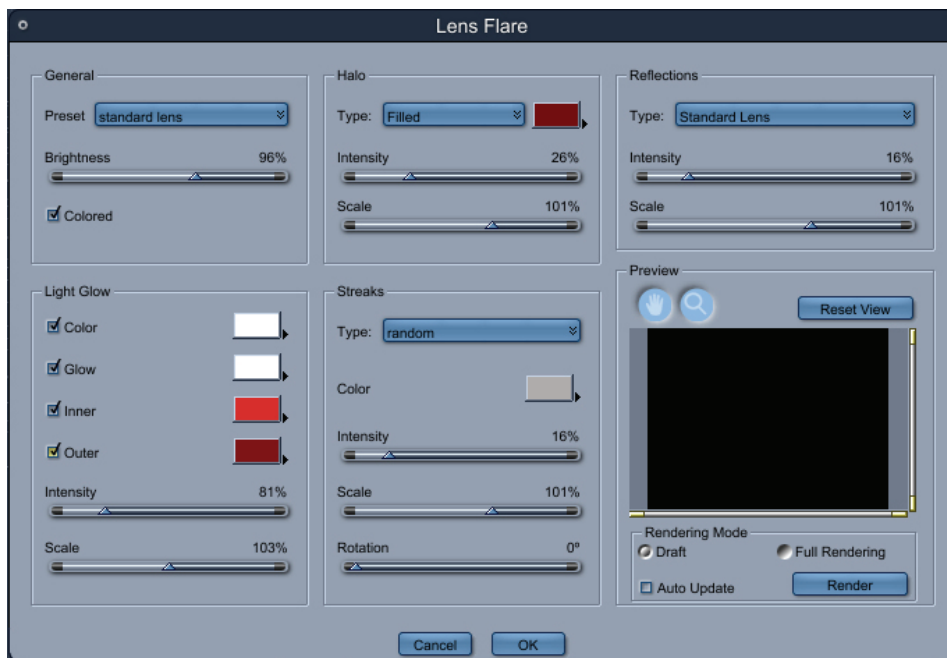
3. Set the General controls:

    - **Settings** sets the type of lens flare. Choose a Carrara preset and edit it as needed. If you choose None, there is no lens flare.

    - **Brightness** controls the overall intensity of the entire flare.

    - **Colored** renders the lens flare in color. When disabled, the lens flare is rendered in grayscale.

4. Set >The Light Glow controls. Enable the option you want to apply, then click the color chip to choose a color:

    - **Color** enables and sets the color of the light itself.

    - **Glow** enables and sets the color of the glow immediately surrounding the light.

    - **Inner** enables and sets the color of the inner glow gradation.

    - **Outer** enables and sets the color of the outer glow gradation.

- **Intensity** sets the strength of the light glow effect.

- **Scale** sets the size of the light glow.

5. Adjust the Streaks controls. This section sets the appearance of the streaks you see when looking at a light (such as a car headlight or street lamp). You can animate all the attributes of a halo, but you cannot change the type of streaks over time in an animation.

- **Type** sets the streak type. Choose a Carrara preset or choose None.

- **Intensity** sets the strength of the streak effect.

- **Scale** sets the size of the streaks.

- **Rotation** rotates the streaks for animated effects such as a photon torpedo, which spins as it moves.

- **Color** sets the color of the streaks.

6. Set the Halo controls. This section sets the appearance of the ring you see when looking directly at a light source. You can animate all the attributes of a halo, but you cannot change the type of halo over time in an animation.

- **Type** Choose Filled for a solid color ring, or Lenticular for a ring in the full spectrum of the rainbow that gradually fades in intensity. Choose a color for a Filled halo, or a starting color for a Lenticular halo.

- **Intensity** sets the strength of the halo effect.

- **Scale** sets the size of the halo.

7. Set the Reflection controls. This section sets the appearance of the set of circles that emanates from the light on a line toward the camera lens. You can animate all the attributes of a halo, but you cannot change the type of reflections over time in an animation.

- **Type** sets the reflection type. Choose a Carrara preset or choose None.

- **Grayscale** turns off color for the reflections.

- **Intensity** sets the strength of the reflection effect.

- **Scale** sets the size of the reflection.

8. Click OK when you are done.

Working with Shaders

Shading

You use Shaders to define an object's appearance. They define not only the texture and color of an object, but how it responds to light. Carefully selected and properly applied shaders enhance the realistic appearance of your scene.

This chapter introduces shaders:what they are and which types of shaders are available:and it provides background information you must know when you create your own shaders. It also introduces the Texture room, including the windows, trays, palettes, and tools you use when applying or creating shaders.

To learn how to apply shaders to your objects, refer to "Shading Objects." Refer to "Creating and Editing Shader Trees" to learn how to create shaders.

Thinking about Textures as Shaders

When you learn to think in 3D, you must analyze surfaces around you to determine how to recreate the same effects in Carrara. As you start to use shaders, experiment to learn which shaders you can use to recreate different types of effects.
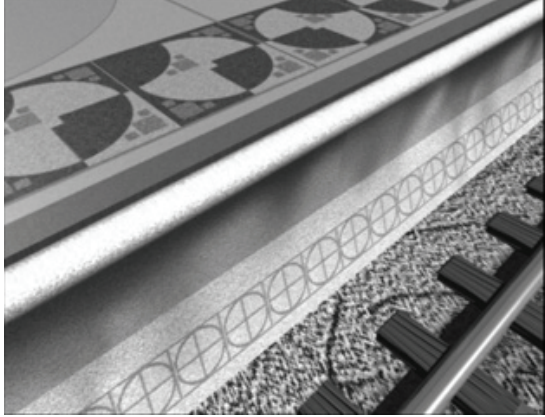
Carrara comes with a library of predefined shaders ready to use right out of the box; however, you can also purchase shaders from other vendors or create your own.

To locate other sources of shaders, visit the DAZ 3D Web site. To learn how to create your own shaders, refer to "Creating and Editing Shader Trees."

The best way to learn about shaders is to try them. As you experiment with different shaders, keep a log of how you think each shader looks so you can quickly select an appropriate shader for the desired effect.

What are Shaders?

A shader is a set of surface characteristics. Each shader can contain settings for one or more of the following attributes: Color, Alpha, Highlight, Shininess, Bump, Reflection, Transparency, Refraction, Glow, Subsurface Scattering, and Translucency.

In time, you'll build your own collection of shaders by editing existing shaders and creating new ones from scratch.

NOTE! As you learn how to build your own shaders, you'll find the pre-defined shaders to be invaluable. Examine them closely: they demonstrate techniques you might not immediately discover through experimentation or reading these chapters.

Refer to "Learning from Existing Shaders" for more information.

How Shaders Work

To work effectively with shaders, it helps to have an understanding of how they work. In traditional animation, images are drawn on different layers of celluloid. For example, one layer is used for background, another for items in the mid range, and another for the foreground.

Shaders work in a similar way. Each channel of a shader is a different layer. Instead of celluloid layers that display background, midground, and foreground layers in an animation, shader channels display Color, Alpha, Highlight, Shininess, Bump, Reflection, Transparency, Refraction, Glow, Subsurface Scattering, and Translucency.

Shaders can get quite complex, but you can start with a simple example. For example, imagine how to create a simple shader for a piece of fruit, such as an orange.

- You begin by placing a shade of the color orange in the Color channel.

- Real oranges have some highlights and shine, so you need to set some values in the Highlight and Shine channels.

- Oranges have a distinctive bumpy texture, so you can either place a bump texture or a Cellular shader into the Bump channel.

- Oranges aren't reflective or transparent. They don't reflect or refract light, and they don't glow. So you do not need any values for the Transparency, Reflection, Refraction, or Glow channels.
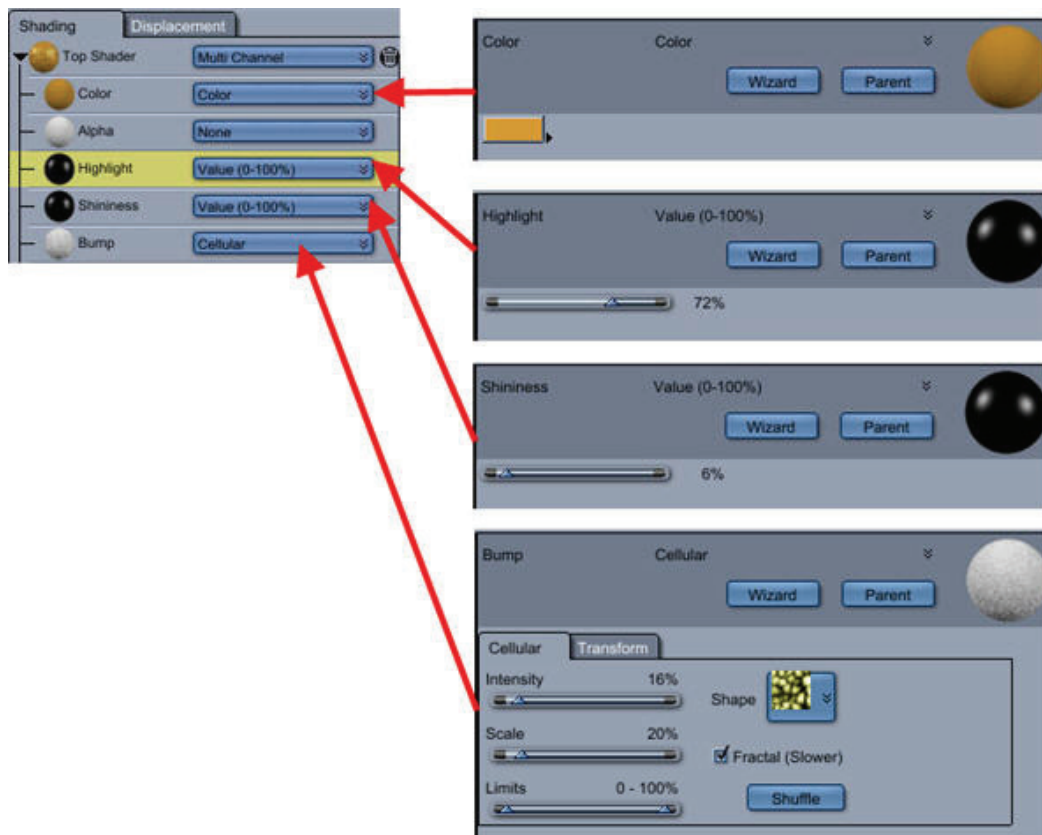
- And the result:
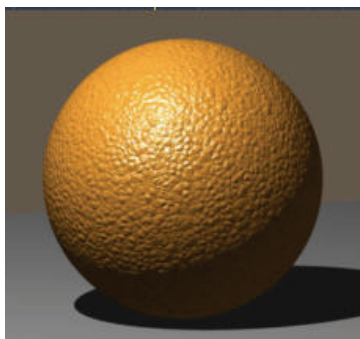
Carrara makes it easy to create complex shaders, where you use functions, operators, and even other shaders are used to create new shaders. Refer to "Creating and Editing Shader Trees" to learn more.

Carrara applies one Master shader to each object. Though each object has only one Master shader applied to it, that shader may contain other sub-shaders. Refer to "Choosing a Top Shader" to learn more.

To learn more about using shaders, refer to "Shading Objects."

Master Shaders

Once you create or place a shader in a scene, Carrara adds it to the Master Shaders list, which you can find in the Sequencer tray.
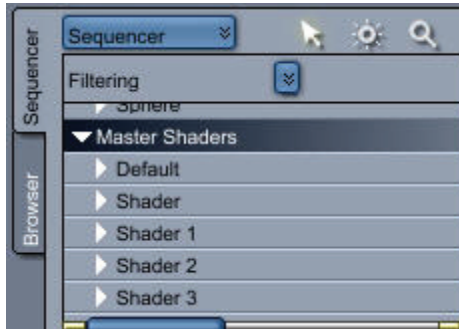
You'll also see a list of the shaders in your scene in the Properties tray: Shaders tab. These shaders are stored in the document as a part of the scene. They can be edited or animated.

When you add objects to your scene, they are assigned the first Master Shader in the list (named **Default**). You can modify this Master Shader as described in "Modifying the Default Shader for a Scene."

You always apply a single shader to an object; however, that shader might be composed of other shaders in what is known as a **shader tree**.

You can vary the effect of using a shader on more than one object by using a Layers List shader and applying layers to the object. To use the chair example, if you wanted to add some scuffs to the chair legs, you could create and position layers, then place a scuff texture in them.

Assigning a Master Shader to an Object

You can assign any Master Shader from the list to one or more objects in your scene. For example, if you make a shader for an antique chair leg, you could apply the shader to all four legs of the chair. When you make any changes to a Master Shader, all objects that use that master shader are automatically updated.

There are two ways that you can apply master shaders to the objects in your scene:

- Drag a master shader from the Master Shaders list in the Sequencer tray and drop it onto an object in your scene.

- Drag a master shader from the Shaders tab in the Properties tray and drop it onto an object in your scene.

Editing a Master Shader

If your scene contains several objects using the same Master Shader, you can use the Master Shader to make changes to the shading of all these objects.

When you create an object based on a Master object, Carrara automatically applies an instance of a specific Master shader to all these objects. You can then assign different Master Shaders to each individual object. .

For example, you might create a bottle and shade it with a green glass shader. You could then duplicate the bottle so that three copies appear in the scene. Suppose you decided later that you wanted marble, rather than glass bottles:you could reopen any object using that shader, affecting all three bottles. Carrara prompts you to confirm whether you want to apply the change to all the objects using that shader or whether you want to create a new Master Shader.

If you apply the change to all the objects, the Master Shader and all the objects using that shader are updated. If you want to create a new Master Shader, then Carrara switches to the Shader Tree Editor so you can create the new shader.

If you use a Master object and want to create variations of a shader, the individual objects will not share the same Master Shader. For example, if you want to create a series of bottles with different labels, you must create separate Master Shaders for each one. Each bottle will use a different Layers List shader.

You can assign a Reference shader as the bottle layer so that several Master Shaders can refer to the same sub-shader. This allows you to edit the Reference shader and automatically update all the Master Shaders that use that Reference shader.

To edit a Master shader:

1. Switch to the Master Shaders tab in the Sequencer.
2. Double-click the icon for the Master Shader your want to edit. The shader opens in the Shader Tree Editor window. Or you can select a Master Shader and switch to the Texture room.

Removing Unused Shaders

Carrara allows you to purge any unused shaders from your document using the **Edit > Remove Unused Shaders** command. You'll want to do so periodically to reduce unnecessary overhead from your scene.

Using Master Shaders in Other Scenes

To store a Master Shader for reuse in another scene, you must add it to the Browser. See "Browser Tray: Shaders Tab."

Editing Master Shader Names

Once a Master shader becomes part of a scene, it is listed in the Sequencer. You can rename the shader if you want to.

Renaming Master Shaders

To rename a master shader:

1. Switch to the Assemble room by clicking the Assemble room button.

2. Display the Sequencer tray by dragging it open if needed.

3. Click the Master Shaders button in the Sequencer.

4. In the Master Shaders list in the Sequencer, select the name of the shader you want to change.

5. Display the Properties tray if needed.

6. Edit the name of the shader in the Properties tray.

Tracking Usage of Master Shaders

To see to which objects a Master Shader is applied:

1. Click the Master Shaders button in the Sequencer tray. Previews of all of the Master Shaders display.

2. Double-click the Master shader to switch to the Texture room.

3. Click the pop-up menu in the upper left corner of the Preview window. A list of objects that use that shader displays.

Editing an Object's Shader

To edit an object's shader:

1. Click an object to select it.

2. Click the Texture room button to switch to the Shader Tree Editor.

3. Edit the shader tree by adding, removing, and modifying shader components.

Editing a Layer's Shader

To edit a layer's shader:

1. Choose the layer component within the Shader Tree Editor and make your changes to that layer.

2. If you want to revert your changes, use Undo.

The Object Surface

Every object has a surface that covers the entire object. A shader gives a surface its characteristics. The basic default surface for a new object is a simple gray color, but you can easily assign another shader to the object.

To replace an object's surface:

1. Locate the shader that you want to use in one of the following areas.

    o   The Master Shaders list in the Sequencer tray.

    o   The Shaders tab in the Properties tray.

    o   The "Browser Tray: Shaders Tab."

2. Drag the desired shader onto an object in the document window. Carrara automatically applies the shader to the object.

3. If you change your mind, use Undo to undo your changes.

TIP! You can also use the Eyedropper tool to "grab" the shader from an object in the document window. Then click the object to apply the shader.

Shading Domains

Real objects are often composed of different textures in different areas of the object. For example, a log has bark on the outside and tree rings on the sliced cross-section.

Shading domains allow you to easily texture different areas of an object in a logical manner. For example, when you model a log as a cylinder it can have several different shading domains:

- The side face

- The front face

- The back face

    (In this example, the front and back face could use the same tree ring texture, but that isn't always the case.) The modeler of the object determines the number of shading domains. The shader domains are separate UV mapping spaces of the object, and each one has a separate number.

You have two ways to assign different shaders to each shading domain:

- The simplest way is to use the shading properties where you can directly assign a shader to a specific shading domain. This panel is available in the **Shading** tab in the Assemble room when you have selected an instance. It is also available in the shading room. The Global shader is the shader that is going to be used if a particular shading domain is empty. Note that it is overridden if you assign a shader to one shading domain. To learn more how to apply a shader to an object with multiple shading domains, refer to "Shading Part of an Object."

- • Or you can use the Layer List shader. You need to create a Layers List shader that contains mapping shaders that allow you to assign layer shaders to specific domains, and the layer shaders themselves. For example, when creating the component shaders used on a log object, the side of the log would have a bark shader, and the ends of the log would have a tree ring shader. The resulting Layers List shader would be known as a log shader, containing the two different component shaders. To learn more, refer to "Mapping Shaders."
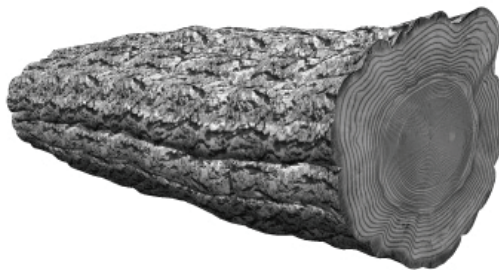
Before you learn more about shaders, you must know more about the windows and tools available in the Texture window.

The Texture Room

The main elements in the Texture room are the Shader Tree Editor, Shader Preview, and Layers tools. You use the Browser, introduced earlier, extensively when applying shaders or saving them for reuse.

The Shader Tree Editor

The Shader Tree Editor gives you complete control over the appearance of your shaders. You can use the Shader Tree Editor to edit shaders stored in the Browser or shaders you've applied to objects in your scene. The Shader Tree Editor lists the type of shader and displays the appropriate branches and parameters.
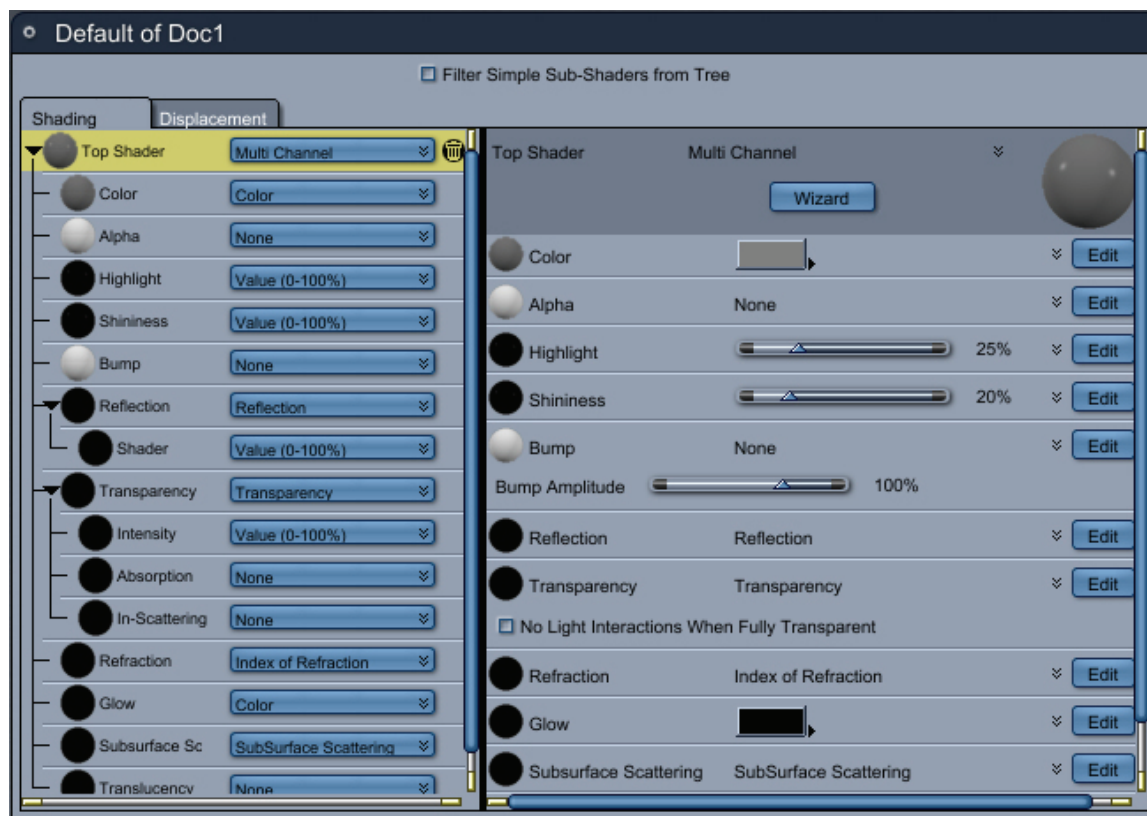
Previewing the Shader

The small previews in the left hand side of the dialog show you a quick view of the whole shader tree. This helps you to easily identify a part of the tree or sub-tree.

The shader preview window shows you how your shader will appear on the surface of an object and supplies you with immediate feedback when you change shading parameters.

Navigation in the Tree

The shader tree editor window displays the type of shader and the components used to create it. You have two ways to navigate through the tree:

- You can select an interesting part of the tree displayed in the left hand side part of the dialog. You will see the detail for this element, or;

- You can use the **Edit** button in the right hand side part of this editor to jump in and see a sub-element. You can also use the **Parent** button to go from a child to its parent in the tree.

If you'd like to see a very condensed view of the tree, you can use the Filter Simple Shader function by enabling the checkbox located at the top of the editor. This mode will filter the information and display only the interesting nodes in the left hand side tree.

Wizard

At each level in the tree you have access to the shader wizard. You can easily import a shader from the library and assign it to the whole tree on only part of it.

Multiple Editors

When you select one or more objects and click the Texture room button, you'll see a Shader Tree Editor window for the shader used in each selected object. For example, if you selected three objects and two of them used the same shader, you'd see two Shader Tree Editor windows.

You can edit multiple shaders at the same time. This allows you to easily drag and drop between shader tree editors.

Shader Preview Window

You can set up the Shader Preview window to preview your shader in a flat preview, sphere, or even on any of the objects that use that shader.
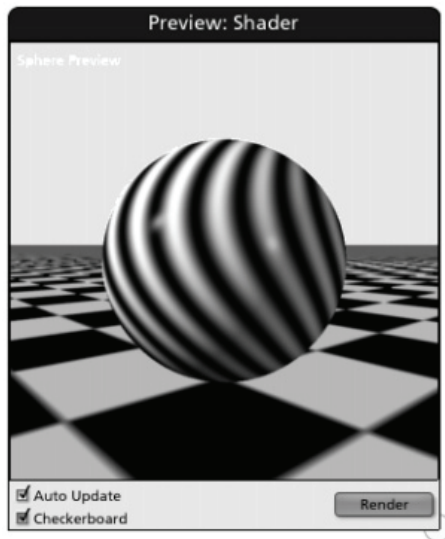
Refer to "Building and Editing Shaders" to learn how to change the preview modes.

The shader preview is rendered every time you modify the shader. You can change this by disabling the **Auto Update** checkbox. In that case, to see the result after modifying a parameter you will have to press the **Render** button. This mode can be useful when working with very complicated and slow shaders.

Layer Tools

The Layer tools help you create layer shaders and mapping shaders interactively in the shader preview window. You can also create layer shaders and mapping shaders in the Shader Tree Editor and position them numerically or with the Select Layer tool.

Here are the four available layer tools:

- Select Layer
- Polygon Layer
- Rectangle Layer
- Oval Layer

To learn more, refer to "Using Layer Shader Tools."

Managing Shaders

To view shaders in your scene:

1. Drag open the Sequencer tray.

2. Click the Shaders button to display previews of the shaders.

To add shaders to the Sequencer:

- Drag a shader from the Browser to the Sequencer.

To remove a shader from the Sequencer:

1. Select the shader you want to remove from the Sequencer.

2. Choose **Edit > Delete** or press Cmd/Ctrl+D to delete it.

If an object was using that shader, the default shader will be applied instead.

To remove all unused shaders from the Sequencer:

- Choose **Edit > Remove Unused Shaders**.

Previewing Shading

Once you have created or edited a shader, you'll want to see how it looks when applied to a particular object. When you view the shader in the Texture room, you'll always see it in a high-quality mode.

In the Assemble or Storyboard room you need to use the Textured preview mode to accurately see the color variations created by the shaders over the surface of your objects. Other preview modes use a single color for each object. To preview the final result in the rendered image, you can use the Test Render tool.

To use the Test Render tool, refer to "2D View Tools."

Learning from Existing Shaders

One of the best ways to learn about shaders is to closely examine the shaders provided with Carrara. As you view a shader in the Shader Tree Editor, you'll find out how it was composed and learn tricks you can use when creating your own shaders.

For a thorough discussion of shader channels and settings, refer to "Shader Structure and Content".

Shading Objects

This chapter explains shading, the process of assigning surface properties to your objects. By designing and applying shaders, you can precisely control the color, texture, and finish on your objects. When Carrara renders your final artwork, these characteristics determine whether an object appears rough or smooth, shiny or dull, transparent or opaque.

Carrara's drag-and-drop interface for applying shaders makes shading easy. You'll keep a collection of shaders in the Browser tray and simply drop them onto your objects in the Assemble, Storyboard, Texture rooms, or the Scene Preview window. As you add a shader to the scene, Carrara stores it in the Master Shader List for the scene. You can also drag shaders from the Sequencer to apply them to objects.

You can apply shaders several different ways: you can apply a shader from the Browser; apply a shader from the Sequencer; create a layer on an object, then apply a shader to that layer; or drag a shader from one scene to another.

You can apply shaders to the entire surface of an object or only to certain areas, such as to layers in a Layers List shader or to the front face of a cylinder. Refer to "Mapping Shaders" to learn how this is done.

Shading Vertex Objects

Since vertex objects can have a large number of sides and complex geometry, it can be difficult to apply shaders to vertex objects. Using standard mapping you may get unexpected results. In some cases, you may need to specify the mapping mode for a specific model to get the results you want.

The Shader room provides a number of controls for adjusting mapping on objects in a scene. The Vertex modeler provides additional control by allowing you to specify how shaders are mapped to individual objects in a model.

Shading an Entire Object

When you create a new object, Carrara creates a default surface for it. To shade an entire object, you simply apply a shader to the object's surface. You can drag and drop a shader from the Shaders Browser or from another object.

You can also edit an object's existing shader. For instructions, refer to "The Shader Tree Editor."

By default the shader is assigned to the shading domain located under the mouse cursor. If you want to assign a shader to the whole object surface (not just one of the shading domains) you can force that by pressing the Shift key.

To apply a shader to an object's surface (drag and drop):

- Press the Shift key and drag a shader from the **Browser: Shaders category** onto an object in the document or Scene Preview window, or onto an object's name in the Sequencer tray.

- Press the Shift key and drag a shader from the Sequencer tray onto an object in the document or Scene Preview window.

To apply a shader from another object:

1. Select the Eyedropper tool.

2. Click an object in the Assemble, Storyboard room, or Scene Preview window to "grab" its shader.

    - Press Shift+Click to apply the shader and keep the Eyedropper "full" of the current shader so you can apply it to additional objects.

    - Press Cmd/Ctrl+click to select a different shader instead of applying the shader currently selected by the Eyedropper tool.

3. Click the Eyedropper on an object to apply the shader.

To apply a shader from an object in another scene:

1. Select the Eyedropper tool and click to "grab" the shader from an object in the document or Scene Preview window.

2. Switch scenes using the Scene Preview window. Refer to "The Interactive Renderer."

3. Click an object to apply the shader. The shader is automatically added to the Sequencer as a Master Shader.

You can also drag and drop a shader from the Master Shaders list in the Sequencer window onto an object in the Assemble or Storyboard rooms, or in the Scene Preview window.

If you are creating a non-animating scene, you might want to apply a shader only to the facing surfaces of an object.

Shading Part of an Object

There are several methods for shading part of an object.

Objects with Multiple Shading Domains

If an object has more than one shading domain, you can drag and drop a shader from the master shader tab directly onto one shading domain. While dragging over the 3D View window, the shading domain candidate for the drop will be highlighted in red. You can quickly shade an object with multiple shading domains using this technique

You can also drag your shader in the Shading tab (this tab is located in the properties when you select the object) and drop it on the correct shading domain. In this tab, you can also access the shader wizard as well as assign a shader to one shading domain by using the popup that lets you choose one of the master shaders of the scene.

Creating Shading Domains on a Vertex Object (Standard, Pro)

In many cases, it can be useful to apply different shaders to different parts of a vertex object. On a model of a face for example, we might want to separate the texture of the skin from the one of the eyes, the teeth, or other specific parts. This is in order to have a shading tree easier to create and a UV mapping simpler to edit.

The part of the object where different shaders are applied is called Shading Domains. For more information, see "Shading Domains."

Note that to be able to apply different shaders on the shading domains, the shader should be in the Layer List of the Parametric Mapping shaders.

By default, all the polygons of a vertex object are part of the default Shading Domain called Texture 0'.

You can then create a new Shading Domain for each polygon of the model and give them different names. These names are important since they are the names that will then be displayed in the shaders of the shading room.

To create a new Shading Domain:

1. Select the polygons on which you want to create a new Shading Domain. Their current shading domain is displayed in selection panel of the Modeling tab.

2. If several polygons are selected, they may have several different Shading Domains. In that case, "Multiple Domains" is visible in the pop-up menu.
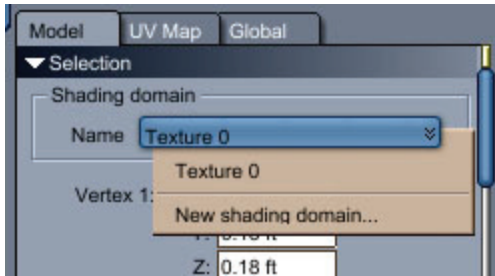
3. Select New Shading Domain at the bottom of the Shading Domain pop-up menu. The Shading Domain dialog appears.

4. Enter a name for the new Shading Domain.

5. Click OK.

To change a Shading Domain to an existing one:

1. Select the polygons for which you want to change the Shading Domain. Their current shading domain is displayed in the selection panel of the Modeling tab.

2. If several polygons are selected, they may have several Shading Domains. In that case, "Multiple Domains" is visible in the popup menu.

3. Select in the Shading Domain pop-up menu the shading domain you want to set the selected polygons to.

To open the Shading Domain Management panel:

The Shading Domains in a vertex model can also be managed from the Shading Domain Management panel. The Shading Domain Management panel gives you the option to add, delete or modify the names of the Shading Domains.

- Open the **Properties tray: Global tab**.
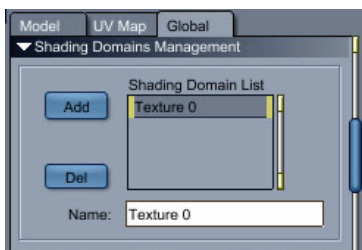
To modify the name of a Shading Domain:

1. Select it in the Shading Domain list. Its name appears in the Name text field.

2. Enter a new name in the text field.

3. Validate with Enter.

Specifying an Object's Mapping Mode

In addition to the three projection mapping modes supported at the scene level, the Vertex modeler supports a custom mode that allows you to specify UV coordinates for particular vertices. For more information about mapping modes, refer to "Mapping Shaders".

If you specify mapping modes for individual objects in the Vertex modeler, and then specify a projection mapping mode for the entire model at the scene level, the modes you specified in the Vertex modeler are overridden. To use the settings specified in the Vertex modeler, use the Parametric Mapping mode at the scene level.

To change an object's mapping mode:

1. Select the object in the Vertex modeler.

2. Switch to UV Editing Mode by clicking the UV Editing button at the top of the Properties tray (Standard, Pro).

3. In the **Properties tray: Model tab**, click the **Mapping** menu and choose a mapping mode: **Custom**, **Box Face**, **Cylindrical**, or **Spherical**. (The 3D Express Properties tray will have different tabs available.)
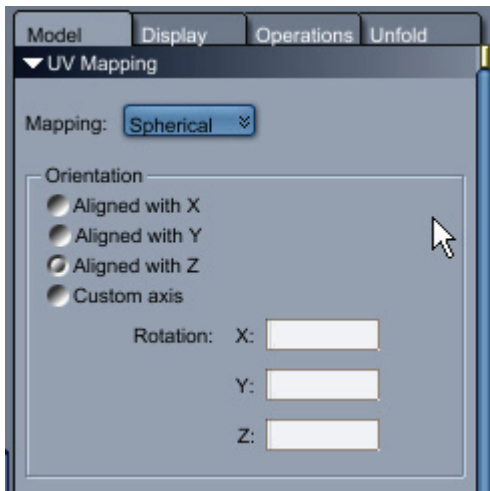
4. Set your options as described in the sections that follow.

5. Apply your settings.

You can use predefined mappings or create your own custom mapping.

Setting a Predefined Mapping

There are three types of predefined mapping modes:

- **Box Face** is designed for mapping 6-sided cubic shapes like cubes and 3D rectangles.
- **Cylindrical** is designed for cylindrical objects like tubes
- **Spherical** is designed for spherical shapes, such as globes and planets

Box Face Mapping

To set Box Face options:

1. In the Vertex modeler, select the object that you want to map.

2. Switch to UV Editing Mode by clicking the UV Editing button at the top of the Properties tray.

3. In the Model tab, click the **Mapping** menu and choose **Box Face**.

4. Enable the mode you want to use:

    - **Full Box**: Wraps the 2D image onto the box much like you'd wrap a package.

    - **Top**: Projects the mapping on the top of the object, with projection continuing through to the bottom.

    - **Left**: Projects the mapping on the left of the object, with projection continuing through to the right.

    - **Front**: Projects the mapping on the front of the object, with projection continuing through to the back.

    - **Bottom**: Projects the mapping on the bottom of the object, with projection continuing through to the top.

    - **Right**:: Projects the mapping on the right of the object, with projection continuing through to the left.

    - **Back**: Projects the mapping on the back of the object, with projection continuing through to the front.

5. By default the image is aligned with the object's bounding box axes (i.e. Align with Global Axis is enabled). If you need to, you can change the orientation of the image on the object.

6. Enable Custom to change the orientation.

7. Enter values in the X, Y and Z rotation fields to change the orientation of the image.

Cylindrical Mapping

To set Cylindrical options:

1. Switch to UV Editing Mode by clicking the UV Editing button at the top of the Properties tray.

2. In the Model tab, click the Mapping menu and select Cylindrical.

Cylindrical
X

Cylindrical
Y

Cylindrical
Z

Custom Cylindrical
Rotated 45 X

Custom Cylindrical
Rotated 45 Y

Custom Cylindrical
Rotated 45 Z
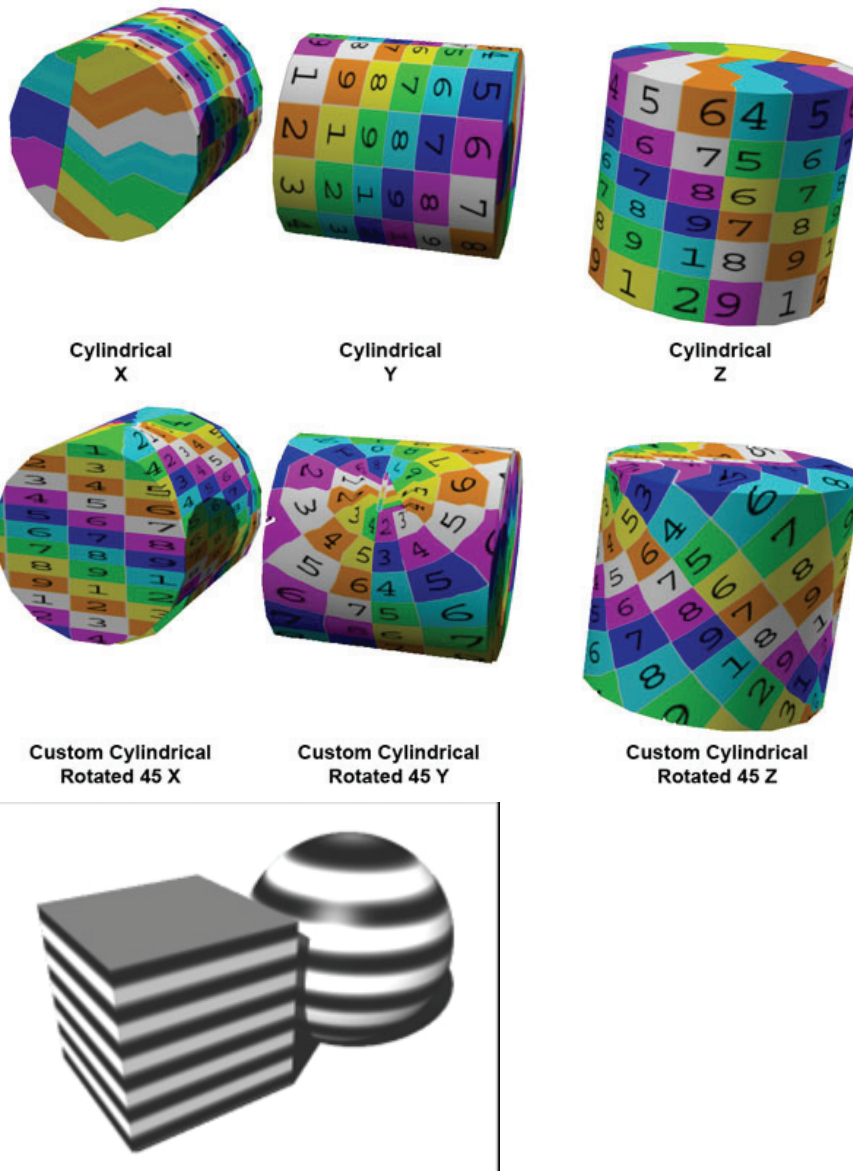
3. Enable an alignment option: Align with X, Align with Y, or Align with Z.

4. You can also design your own orientation:

    •   Enable the Custom option.

    •   Enter values in the X, Y and Z rotation fields to change the orientation of the image.

Spherical Mapping

TBD

Setting a Custom Mapping

When you choose the Custom mapping mode in the **Properties tray: UV Editing Mode: Model tab: UV Mapping panel** you can specify how UV coordinates from the shader should be applied to the selected vertex or group of vertices.

When the mode is set to Custom, the UV values determined by the previous mode are assigned on the vertices. So, if you want to create a custom UV mapping that is similar to a spherical, cylindrical, or box mapping, set the mapping to one of these before setting it to Custom.

When you enable Wrap, the shader is wrapped around the object from the specified U or V coordinate.

To view the UV coordinates that are set for a particular vertex:

1. Select the vertex.

2. Switch to UV Editing Mode by clicking the UV Editing button at the top of the Properties tray.

3. In the Model tab, make sure the UV Mapping Mode is set to **Custom**.
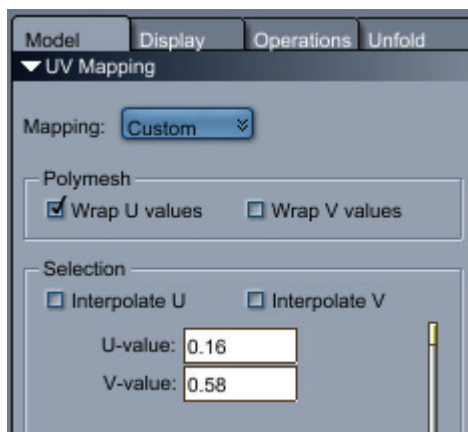
4. Update the settings as needed:

- The current U and V values are displayed in the number fields.

- The UV mapping on an object might not be continuous. This means that one single vertex can have different UV values used by polygons sharing this vertex. When a vertex is selected, all its existing UV values are displayed.

  An example of non-continuous mapping is the box mapping when applied to all the faces of a cube. In this case, the vertices at the corners will have 3 UV values each.

- The mapping on a polymesh can also wrapped in U and/or V values. Wrapping a texture map avoids going through all the texture maps when a vertex has a U or V value near 1 and is linked to a vertex with a value near 0.

- A vertex can also be in Interpolated mode. In that case, its UV values are determined by the UV values of its surrounding vertices.

Editing Custom UV Values (Standard, Pro)

The UV Editor contains a 3D view of the polymesh, a 2D display of the UV values on the polymesh vertices, and a set of tools to manipulate them.

To edit the UV mapping on a polymesh:

1. Select the polymesh or part of it.

2. Switch to UV Editing Mode by clicking the **UV Editing** button at the top of the Properties tray.

3. The view should automatically change to the UV View but if not, choose **UV View** from the View menu. If needed, select the **Two Views Vertical** layout.

The viewport should look similar to the following image:
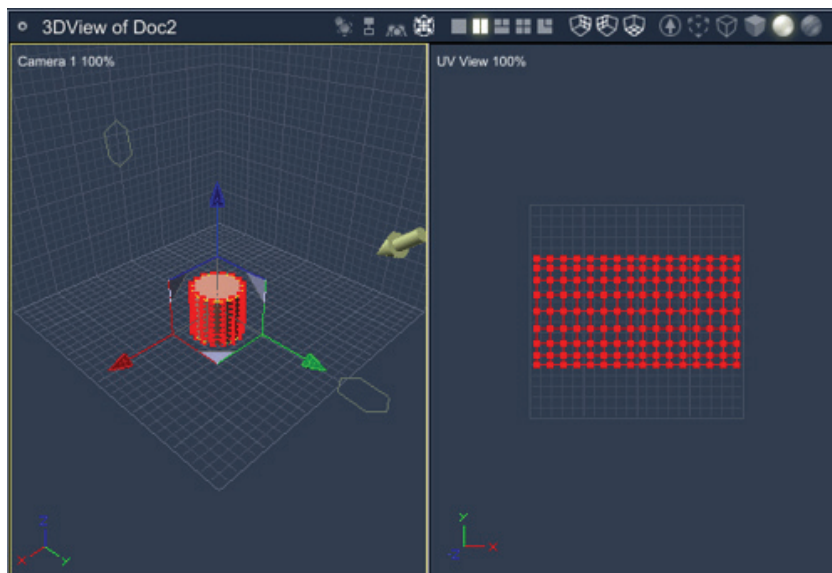
1. The pane on the left is the 3D view and the vertices and polygons can be selected and manipulated using the default view controls. You can select the vertices in the 3D View and their corresponding UV values will be selected in the UV View.

2. The pane on the right (UV View) is where you manipulate the UV mapping. See "The UV View" for more on using this view.

4. Edit the UV values in the UV Editing tabs.

If there is no texture on the object, the 2D view will only display the mesh of the vertices in their UV values. If the shader has a 2D texture map, it appears in the background of the mesh.

The UV View

The UV View displays a flattened mesh of the model using the UV values of the vertices to determine their position. When you modify the position of the point in the UV View, you modify the mapping on the object but do not affect the geometry of the polymesh.
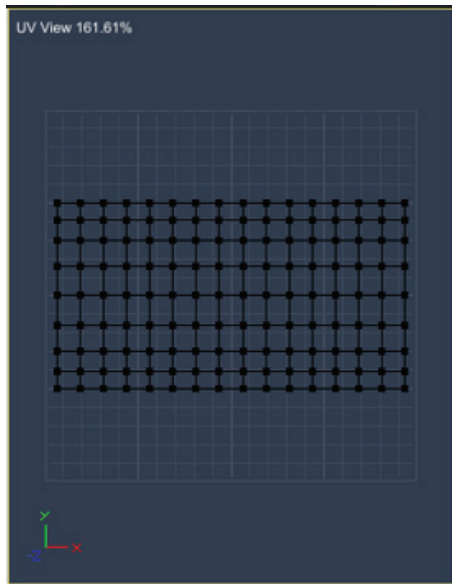
The **Zoom** and **Pan** tools can be used in the 2D View to display more specific areas of the texture.

To select one or several points in the 3D or UV Views:

1. Click on one of the **Selection Mode** buttons in the Properties Tray.

    - **Regular Selector**: Allows selection of polygons (3D View only), edges, or vertices
    - **Vertex Selector**: Only allows selection of vertices.
    - **Edge Selector**: Only allows selection of edges.
    - **Polygon Selector**: Only allows selection of polygons in the 3D View.

2. Select the vertices, edges, and polygons:

    - Click on the vertices, edges, or polygons. Hold down **Shift** to select multiple items.
    - Click in an empty area and drag to use a Marquee Selection.

- If there is more than one Shading Domain on the polymesh, selecting the one you want to edit in the Shading Domain List (on the Display tab) will select all the points of that domain.

When a shading domain is selected, its associated texture is displayed in the 2D view. For more information about Shading Domains, see "Creating Shading Domains on a Vertex Object."

3. Depending on the tool currently selected, move, rotate or scale the selected vertices, edges, or polygons.

4. To move the selected items in snap to grid mode, press **Command/Ctrl** before moving the items.

5. You can also move a selected vertex by directly entering its U and V values in the Model tab.

The UV Editor Model Tab

The Model tab provides options for mapping the current selection.

The top of the tab shows options available for the current mapping mode (shown further down) while the bottom of the tab displays the current statistics for the mesh. See "Setting a Predefined Mapping" and "Setting a Custom Mapping" for information on using these mapping modes.

The UV Editor Display Tab

The **Display tab** provides several options for displaying the UV values.
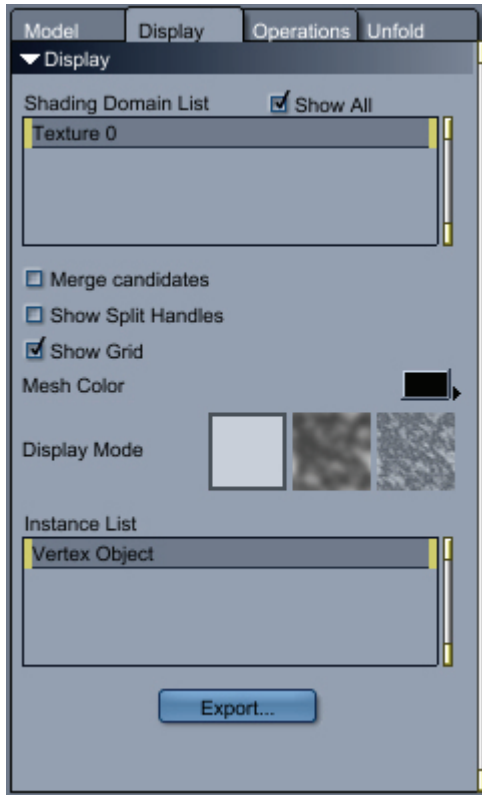
The Display tab provides the following options:

- **Shading Domain List**: Choose a shading domain from the list of shading domains set on the polymesh. Each shading domain can be selected by clicking on its name. By default, only the current shading domain is displayed in the 2D view. To display them all at once, check the **Show All** option.

- **Merge Candidates**: Select to merge different UV values when they are shared by a single vertex on the polymesh.

  To merge several UV values on a single vertex:

  1. Check the **Show Merge Candidates** option in the Display tab.

  2. Click on the **Move** tool.

  3. Select the point you want to merge. If the UV mapping has a discontinuity on this point (several UV values for one vertex), the other UV values shared by this vertex will be highlighted in the 2D View.

  4. Drag the point over another highlighted point to merge them together.

- **Show Split Handles**: Select to create or remove the discontinuities on a vertex. see "Editing Custom UV Values" for more on discontinuities when UV mapping.

  To create a discontinuity on a specific vertex:

  1. Check the **Show Split Handles** option in the Display tab. The points are now surrounded by other small handles, each handle corresponding to one of the polygons attached to the vertex.

  2. Click on the **Move** tool.

3. Click on one of the handles to split the UV value. The same vertex on the polymesh will now have different UV values depending on which polygon it is.

- **Show Grid**: When selected, shows a grid under the UV map.

- **Mesh Color**: The mesh color can be modified in order to see it no matter what the texture underneath might be.

- **Display Mode**: As a background of the mesh, a color (white by default) or a low- or high-resolution texture can be displayed.

- **Instance List**: If there are different instances of the vertex object in the scene, each instance could have a different shader on it. You can choose the shader you want to work with by selecting it in the Instance List.

- **Export**: The mesh can be exported in an image file. Note that only the mesh and the background color will be exported. The texture map, the grid or the points won't be exported.

The UV Editor Operations Tab

The **Operations tab** offers a list of projection methods that can be applied to the current selection.

Misc Section

The **Misc section** provides a tool which splits a set of vertices from the mesh.

To split a set of vertices:

1. Select the set of vertices to be split.

2. Click the **Detach Polygons** button.

   The vertices around the selection will be split to "free" the selected polygons.
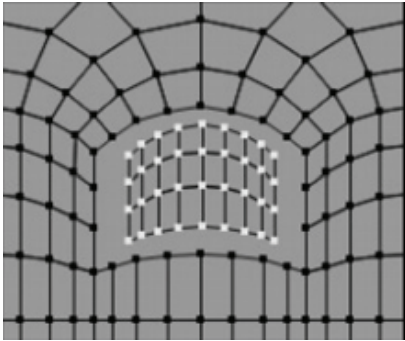
Projection Section

The Projection section is where custom UV maps can be set and modified. If the current mapping is not a custom mode, you will be asked if you want to convert the current mode to custom.

To apply a projection:

1. Select the vertices to apply the projection to.

2. Choose among these four available projection methods: **Box**, **Cylindrical**, **Spherical**, and **Planar**.

3. Adjust the settings on the chosen projection.

4. Click the **Apply** button.

The following options can be adjusted for the different projection methods:

- **Framed**: When selected, adds a small space between the faces so they are more easily visible.

- **Box** projection options:
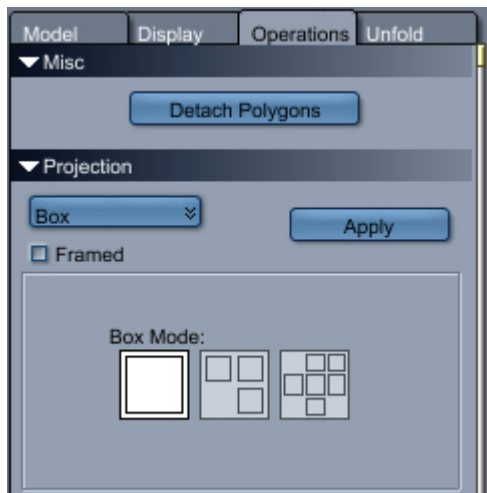
- **Box Mode**:

    - **Cover All**: Projects all faces of the box, one over the other.

    - **Cover Front/Back**: Projects the faces 2x2, creating three distinct areas - the front and back sides together, then the left and right sides, and finally the top and bottom sides.

    - **No Cover**: Projects the six faces of the box separately so none of them overlap each other.

- **Cylindrical** and **Spherical** projection options: Sets the axis of projection and an offset for the center of projection.
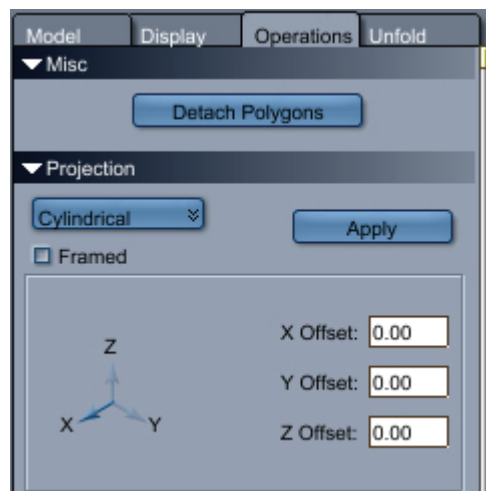
- **Planar** projection options: Provides options that let you split the mesh by orientation or by the offset position from the center. You can also display the projected mesh horizontally or vertically.

The UV Editor Unfold Tab

The **Unfold tab** allows you to unfold a UV map. This is done by defining seams and, if necessary, pinpoints, which are points where the unfolded template will be stitched.
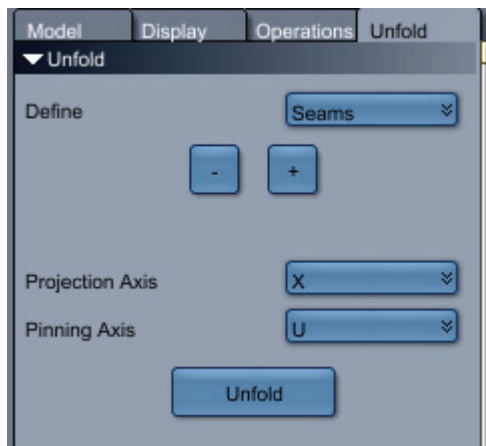
In order to unfold a UV map, the seams must be defined on the mesh. A seam is a selection of contiguous edges on the 3D model from which the UV map will be unwrapped. Seams must be defined in specific areas and created in order to get the optimal unfolded model and avoiding UV distortions which are too obvious.

In addition, pinpoints will help the tool make a uniform and well proportioned unfolding since the unfolding will be performed "around" the pinpoints.

To unfold a map using seams:

1. Select the edges that will define the seam.
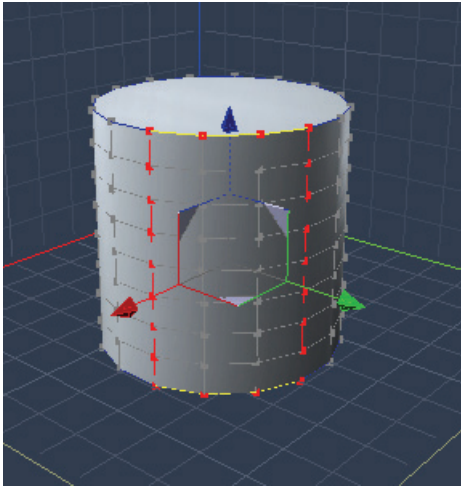
2. Select **Seams** from the **Define** menu.
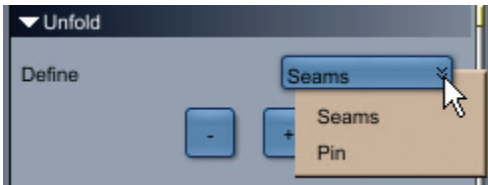
3. Click on the **Plus (+)** button to add the selection to the Seams selection. To remove the selection from the seam set, click on the **Minus (-)** button.

4. Define the **Projection Axis** of the Unwrap tool. Carrara uses a planar projection to perform the unfolding operation so generally, the best axis is the one perpendicular to the planar projection.
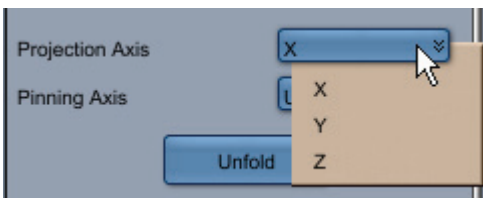
5. Click on the **Unfold** button.

6. If the mapping mode of the object is not currently set to Custom, a confirmation dialog will come up asking if you'd like to change the UV mode to custom. Click on **Yes** (or on **Always** if you'd like it to change the mode without asking).

The action you selected can only be performed on custom UV.
Would you like to change the UV mode to custom?

Always   Yes   No
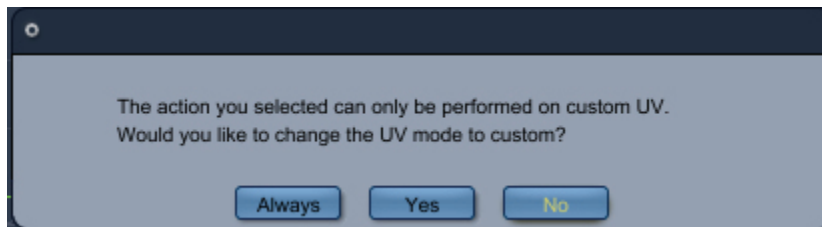
The UV map will be unfolded along the selected seam line which will show as a light blue line.
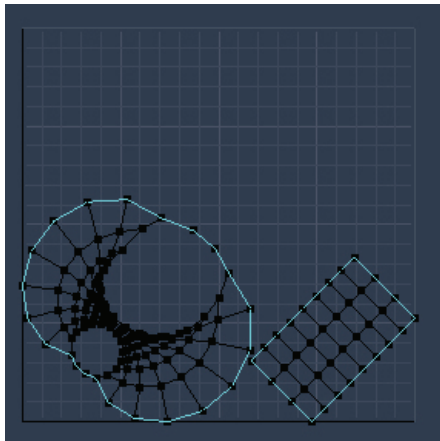
To add to an unfold selection using pinpoints:

1. Select the pinpoints that will define the seam to be added.

2. Select **Pins** from the **Define** menu.

3. Click on the **Plus (+)** button to add the selection to the Seams selection. To remove the selection from the seam set, click on the **Minus (-)** button.

4. Define the orientation of the **Pinning Axis**, in U or V, which lets you perform a 90° rotation when unfolding.
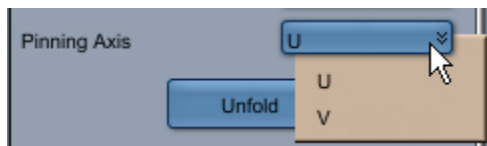
Pinning Axis   U
                          U
            Unfold   V

5. Click on the **Unfold** button.

6. If the mapping mode of the object is not currently set to Custom, a confirmation dialog will come up asking if you'd like to change the UV mode to custom. Click on **Yes** (or on **Always** if you'd like it to change the mode without asking).
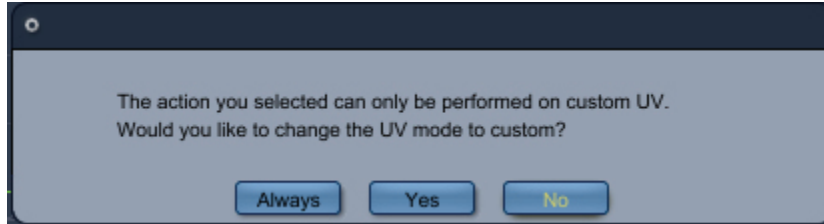
The seam between the selected points will be added to the unfolded map set.

Projection Mapping Shaders

When you model an object such as a beveled text object, you assign shading domains that define different materials. For example, a can of soup might contain three shading domains:Â Label, Top, and Bottom. Mapping allows you to easily texture different logical areas of the object.

Projection Mapping shaders identify UV spaces and UV coordinates. They determine to which shading domain to apply themselves. You can use a Projection Mapping shader to define how each of the shading domains apply the shader to the 2D surface of the object.Â You can apply cylindrical mapping, flat mapping, spherical mapping, or projection mapping to an object or a layer.

Objects that you import from other applications often contain limited material and shader information. When you shade an imported object, you must choose one of Carrara's projection mapping modes: Flat Mapping (for flat planes or boxes), Spherical Mapping (for spheres and elliptical shapes), or Cylindrical mapping (for cylinders and similar shapes). Parametric mapping may be difficult to apply to imported objects that are very complex.

In the different projection mapping modes, Carrara maps the image onto an invisible primitive (a box, a sphere, or a cylinder) that encloses the object. Carrara then projects the image from the invisible primitive onto the object itself. When you choose Flat mapping, you can specify which face of the box primitive you want to map onto. When you choose cylindrical or spherical mapping, you can specify the orientation of the primitive in relation to the object.

Note For the best results, you should choose the primitive that best resembles the object you are mapping. For example, spherical mapping would be appropriate for a basketball and cylindrical mapping would be right for a juice can.

To add a mapping shader to a shader tree:

1. Select an object and view its shader in the Shader Tree Editor.
2. If the top shader of the Master Shader is not a Layer List Shader, click the top branch list item and change it to one of the Projection Mapping types (Cylindrical Mapping, Flat Mapping, Parametric Mapping, or Spherical Mapping).

3. Carrara displays the options that are appropriate for the selected mode. Set your options as described in the sections that follow, and apply your changes.
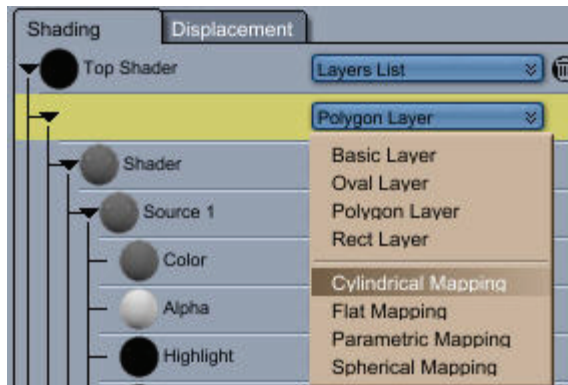
Cylindrical Mapping Shader

To set the mapping alignment:

- Click the icon to choose the alignment you want:X, Y, or Z axis.

If you slice a cylindrical object, each slice uses the same front face and back face information.
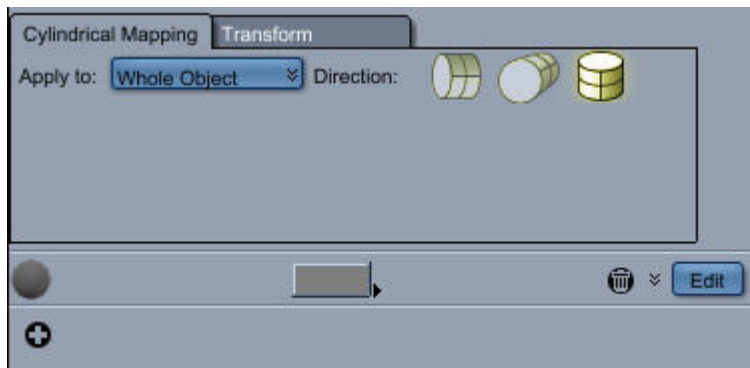
Flat Mapping Shader

The **Single Face** mappings project the image on one side of the object. The projection continues through to the other side of the object.
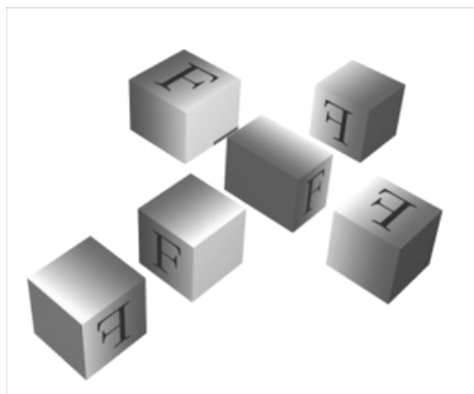
This is how the various face Flat mapping options appear on an object.

By default, the image is aligned with the object's bounding box axes.
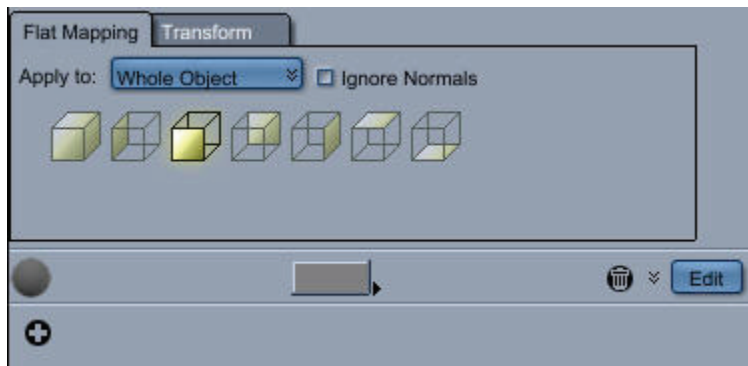
Parametric Mapping Shader

Carrara's default mapping mode is known as parametric mapping. Parametric mapping is like applying a decal to an object's surface:each pixel in your image maps directly to a specific point on the surface of your object. This straightforward approach minimizes distortion and loss of image quality. You'll want to use parametric mapping for most objects you create in Carrara. However, parametric mapping cannot be used on Metaball objects.
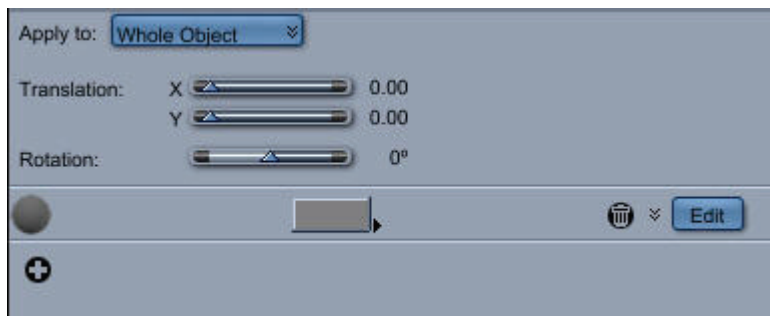
Spherical Mapping Shader

To set the mapping alignment:

- Click the icon to choose the alignment you want:X, Y, or Z axis.

If you slice a cylindrical object, each slice uses the same front face and back face information.
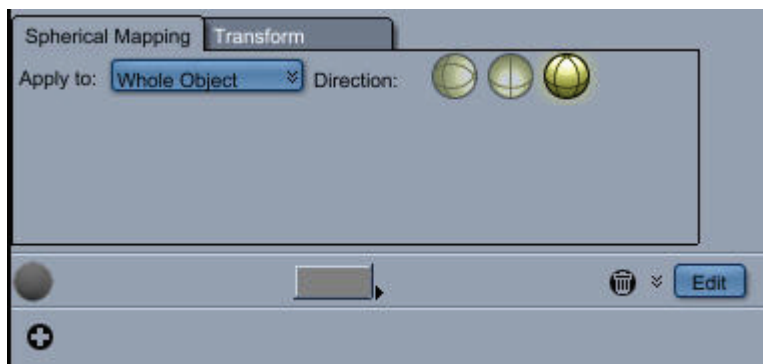
Layer Shaders

Many real-world objects have non-uniform surfaces. Variations on an object's surface might be as simple as a painted-on logo, or as complex as an intricate inlaid wood design.

To achieve effects such as these, you can use Carrara's layers. These tools are located in Texture room, directly beneath the menu bar.

The Layers tools enable you to create a Layers List shader. This allows you to apply different shaders to various regions on the surface of an object. These regions, called layers, take the form of geometric shapes. Once you create them, layers can be selected, moved along the surface of the object, modified (in shape, size, or shader content), overlapped, or deleted. You can create any number of layers on the surface of an object. Each layer exists in the Layers List shader and you can edit its shape, component shader, or opacity mask in the Shader Tree Editor.

Because the Layers tools allow you to place other shaders in the layers, you're not limited to painting with color. You can paint with gold, marble, or concrete and apply not only color, but bump, reflection, transparency, and the other shading attributes as well.

This section introduces the Layers tools and explains how to use them. It also explains how Layers List shader layers relate to each other and to the object's surface. Finally, it discusses advanced topics such as mapping modes, and the Master and Object shading layers.

Using Layer Shader Tools

Carrara's Layers tools allow you to create layers directly on your 3D objects. Each tool provides visual feedback to show you how your layer appears on the surface of your object.

The Layers tools work best on objects created in Carrara. You can create layers on an object imported from another application, but you might need to change the object's mapping mode to achieve satisfactory results. For more information, refer to "Editing Layer Shaders."

To use a Layers tool:

1. Choose a Layers tool. The behavior of each tool is described below.

2. Draw on the surface of an object in the Shader Preview window.
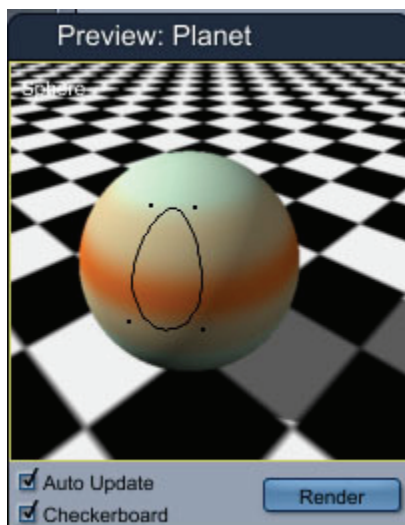
Rectangle Tool

Drag to create a rectangle. As you drag, Carrara draws a "rubber band" preview in real time, to show you the boundaries of the rectangle. Carrara shades the layer when you release the mouse button.

Polygon Tool

Click once to position each vertex of your polygonal layer. As you position vertices, Carrara draws the line segments connecting the vertices. Double-click at the last vertex to automatically close the shape.

Although you can later resize or "stretch" a polygonal layer, you won't be able to re-edit its vertices.

Ellipse Tool

Drag to create an ellipse. As you drag, Carrara draws a "rubber band" preview in real time, to show you the perimeter of the ellipse.

When you draw with the Rectangle or Ellipse tool, your first mouse click positions one corner of your layer's bounding rectangle, then you drag to position the opposite corner. Carrara normally uses the shortest path to connect these opposite corners:that is, the layer will not wrap all the way around the object. If you want the shape to wrap "the other way" around your object, hold down the Option/Alt key as you drag.

Working with Layers

Once you've created layers, you can move, resize, overlap, and delete them.

To select a layer:

1. Choose the Layer Selection tool.

2. Click the layer you want to edit. Carrara shows the layout of the layer. The layer has four handles, one at each corner.

To move a layer:

- With the Layer Selection tool, drag the shape along the surface of the object.

- You can also use the Shader Tree Editor to numerically reposition a layer. Type the numeric value in the parameters to reposition the specified layer.

To delete a layer:

1. Select the shape.

2. Click the Trash can in the Shader Tree Editor.

To resize a layer:

- With the Layer Selection tool, drag one of the corner handles on the shape's bounding rectangle. Drag toward the center of the shape to shrink it. Drag outward to stretch it.

- You can also use the Shader Tree Editor to numerically resize a layer. Type the numeric value in the parameters for the specified layer.

When you use the Shader Tree Editor to edit a Layers List shader, the layers are listed in the shader tree. You can edit the contents of the layer layers from within the Shader Tree Editor. Refer to "Creating and Editing Shader Trees" to learn more.

Layering: How Layer Shaders Interact

When you create a layer on the surface of an object, the layer's shader overrides the shading characteristics of the object's surface. Likewise, when several layers overlap, the topmost layer's shader overrides those below. That is, the last layer in the list will appear to be "in front" of the other layers.

There is one important corollary to this rule, however. When a layer's shader has one or more empty channels, it inherits the settings for these channels from the layer immediately above it (or from the surface, if there is no layer above).

For example, if you create a layer on an object with a bumpy surface, any layer you create will also be bumpy unless the shader you're using specifies different bump settings. To create a smooth layer on a bumpy object, you would use a shader with a constant value in the Bump channel.

The Shader Tree Editor allows you to control the opacity of an object's layers. Unlike a shader's Transparency setting, layer opacity does not affect an object's translucence; rather, it controls how layers interact. If a layer's opacity is less than 100%, its shader does not fully override the shaders of the surface and layers above; rather, the shaders effects are mixed.

Editing Layer Shaders

The parameters for the different branches in the Shader Tree Editor allow you to edit the position and size of an object's layer with numerical precision. You can also add and delete layers, change layering, specify a layer's opacity, convert a layer from one type to another, remove an object's surface, and change an object's mapping mode.

The Layers List shader displays the component shader and opacity mask for each applied layer. Select which type of component shader you want to use and the appropriate branches display in the Shader Tree Editor.

To change the layering order:

1. Select the layer you want to reorder in the Shader Tree Editor.

2. Drag the layer into the order you want. In a moment, the object preview updates to your changes.
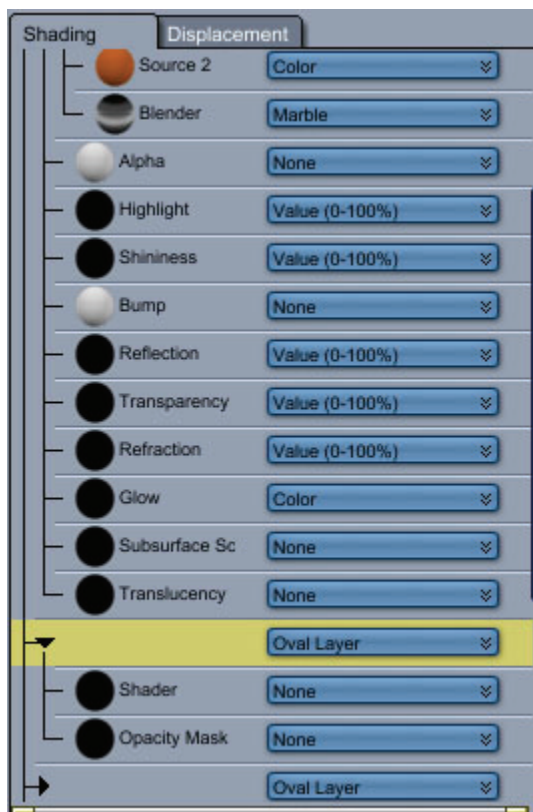
To create a new layer shader:

1. Click the Plus icon from within a Mapping branch of the Shader Tree Editor.
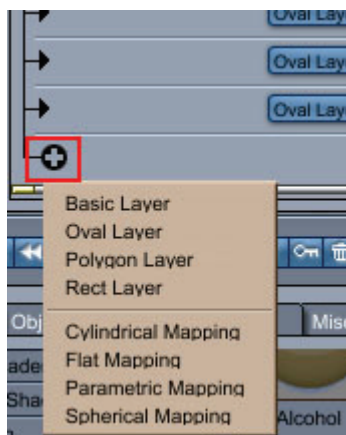
2. Select the type of layer from the list.

3. Carrara creates a new layer on the surface of the object. The new shape has the default size, position, and shader:all of which you can change.

To delete a layer shader:

1. Select an element in the Shader Tree Editor.

2. Click the Trash can icon, or press Delete.
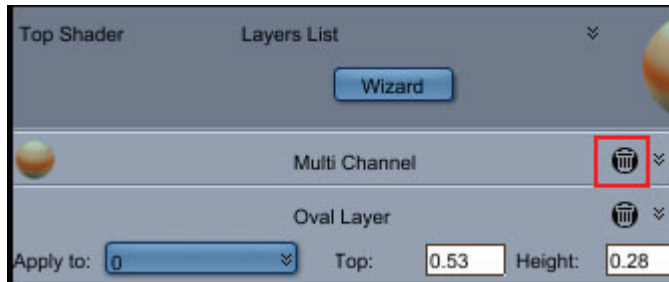
To change the layer type:

- Choose the type you want from the Shape pop-up menu that lists the name of the current type of layer.

Mapping Shaders

Most shader content is two-dimensional. Texture maps, for example, are nothing more than 2D images. Many procedural shader functions:including Carrara's checkers and wires:also produce two-dimensional image data.

Carrara uses a process known as mapping or projection mapping to apply this 2D shading information to the surface of a 3D object.

Carrara's interface allows you to shade objects without worrying about the internal "nuts and bolts" of mapping. Most of the time, you can simply apply shaders on objects with the Layers tools and let Carrara take care of the details. Occasionally, however, you might find that changing an object's mapping mode makes it easier for you to achieve the results you want.

When you change an object's mapping mode, Carrara changes the method it uses to map 2D shapes and images to the object's surface. As a result, the Layers tools behave differently on the object. Depending on the shape of your object and the mapping mode you choose, the difference can be subtle or quite dramatic.

The four different mapping shaders are:

- Parametric Mapping shader

- Flat Mapping shader

- Cylindrical Mapping shader

- Spherical Mapping shader

Each of these mapping shaders allows you to change the way sub-shaders are projected on the object.

Mapping shaders also allow you to define to which shading domain of the object they can be assigned. When you use the Layer Shader tools, Carrara automatically inserts Mapping Shaders for you so that

your layer shaders get applied to the appropriate shading domain of the object. You can also add Mapping Shaders manually. Refer to "Shading Channels Reference."

Texture Maps

You can use a texture map to apply an image to an object. You can create a texture map with a separate 2D drawing program such as Painter, Photoshop, or Paint Shop Pro. Then, the map is applied to the object. If the image file has an alpha channel associated with it, you can use that information as well.

Texture maps allow you to create effects that would be cumbersome or impossible to create using other methods. Texture maps are useful when you must create 2D effects, such as signs of wear on an object or lettering.

Note If any of the shaders in your shader files contain Texture Map components that reference external image files, you must maintain the relative path from the shader files to the image files.

If you move either the shader files or the image files to a different folder, Carrara prompts you to locate the missing image files when it loads the shader files into the Browser tray.

Refer to "Layer Components" to learn how to load a texture map and apply it to an object.

Here is an example of a spacecraft created using a texture map.

Here is a similar spacecraft created without using a texture map.

Modifying the Default Shader for a Scene

When you create new objects in a scene, they are assigned the Master Shader named "Default." You can modify this master shader to control what will be the default shading of the objects in your scene.

To modify the default shader for a scene:

1. Switch to the Master Shaders view in the Sequencer.

2. Double-click the icon for the Master Shader named "Default." The shader opens in the Shader Tree Editor window. Or you can select the Master Shader and switch to the Texture room.

3. Edit the shader in the Shader Tree Editor.

Creating and Editing Shader Trees

Shaders are a powerful way of bringing your 3D objects to life. You can use Carrara's extensive libraries of shaders to add life to your objects, but because there is no limit to the types of objects you can create, you might need to build a unique shader to suit each object you create.

Carrara's shader structure makes it possible to create your own custom shaders. Much like mixing your own colors for painting on an artist's palette, you'll use the Shader Tree Editor to create an infinite number of different textures, colors, and surfaces.

This chapters explains the Shader Tree Editor and guides you through the process of creating your own shaders.

Shader Structure and Content

Shaders are composed of a number of different elements, each of which combine to create a realistic material. This section describes the basic elements of a shader, and also points to a number of other topics that you can study as well.

The basic elements of shaders in Carrara are as follows:

- **A Shader Tree**: In Carrara, the shader tree is a modular structure that allows you to define the different properties in your shader. The shader tree is described further in this section, under "Shader Tree." You can also read "Building and Editing Shaders" for a more detailed look at the Shader Tree Editor.

- **A Top Shader**: The topmost element in a shader tree is the Top Shader, of which Carrara provides several different models. The top shader that you select determines the material properties that you can use in the shader. The various types of top shaders are described in "Choosing a Top Shader."

- **Shader Channels**: Shader channels are the individual properties that are combined together to create a realistic material. Some shaders might use only a Color channel, while other shaders might use a combination of color, bump, reflection, glow, transparency, and so on. A shader does not have to utilize all of the channels that are available in a top shader. Shader Channels are described in "Shader Channels Reference."

- **Shader Components**: Once you determine the types of channels that you want to use in your shader, you then plug shader components into those channels. For example, to define the properties of a Color channel, you can use a solid color, a texture map, a procedural texture, or a formula that defines the colors used. The various shader components are described in "Shader Components Reference."

Shader Tree

A **shader** is a set of surface properties that you can assign to an object. Carrara features a modular structure for defining shaders: the **shader tree**. The shader tree's modular nature allows for great flexibility in shader content.

A shader tree is a set of shaders assembled in a hierarchical way. Each shader in the tree uses the output of its branches in the tree and combines these outputs to provide the final appearance of the surface.

In simple cases, the tree is only one level deep. This is the case with the default Multi-Channel shader.
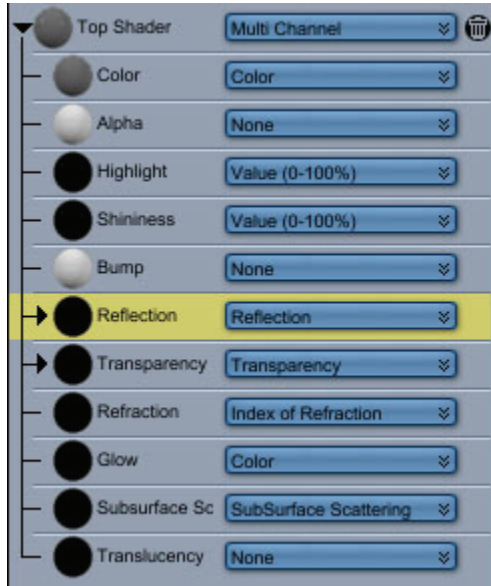
In more complex cases, the tree has multiple levels. For example, the glow branch of a Multi-Channel shader can contain a mixer shader that, in turn, uses several sub-shaders. The tree can be as deep as necessary. However, the deeper the shader tree, the longer it will take to be calculated.

You can use shaders as components when creating other shaders. A sub-shader is simply a shader used within another shader. You can use any type of shader as a sub-shader. The only distinction is whether the shader is at the top of the tree or located under one of the branches.

Editing the Shader Tree

You build a shader by placing components and sub-shaders on the branches of the shader tree. To edit a shader tree, you simply add, remove, and replace shader components.

You can select options from the pop-up menus to create your shader. Notice that you can nest shaders within other shaders to create complex effects. Depending on the type of shader you are creating, you'll see a plus sign or the Trash can icon, which you can click to add or remove branches.

Carrara's drag-and-drop interface pertains not only to applying shaders, but also to building them. You can drag a component or sub-shader from one branch of the shader tree to another. You can also drop a shader from the Shaders Browser onto a branch of the shader tree or drag a sub-shader from the tree into the Browser for storage.

To place a shader component or sub-shader on the shader tree:

1. Choose a shader from the pop-up menu.

2. Choose the component you want to add from the pop-up menu corresponding to the branch you want to edit.

For complete information on the components, refer to "Shader Components Reference."

To move components by dragging:

- Drag a sub-shader or component onto the branch of the shader tree where you want it.

    o You can drag a shader from the Browser.

    o You can drag a component from another open Shader Tree Editor window.

Carrara copies the component or sub-shader you drag onto the branch where you drop it, replacing any contents that might have been there before.

When you drop a shader onto any branch of a Multi-Channel shader, Carrara places only the contents of the applicable channel on the branch. If the shader you're dropping has no components in the applicable channel, Carrara notifies you, and adds nothing to the branch.

Note When you drop a shader onto the Source 1 or Source 2 branch of a Multi-Channel Mixer shader, Carrara places the entire shader you're dropping (all eight channels) on the tree.

To remove the contents of a branch:

1. Select a component or sub-shader.

2. Choose **Edit > Delete,** click the Trash can icon, or press the Delete/Backspace key.

To copy a component or sub-shader to another channel:

- Hold Cmd/Ctrl, then drag a component or sub-shader from its branch and drop it on a different channel tab. Carrara replaces the entire contents of the channel with the component or sub-shader you drop.

Building and Editing Shaders

Most of the work in building and modifying a shader takes place in the **Shader Tree Editor**. The **Browser: Shaders category** and shader document windows are also useful.

You can use the Shader Tree Editor to design a new master shader, modify a master shader already used in your scene, or modify a master shader you've stored in the Browser.

Displaying the Shader Tree Editor

In order to display the Shader Tree Editor, you need to select one or more objects in your scene.

- When more than one object is selected, and they all use the same shader, a single Shader Tree Editor displays and you can select which object you want to display in the Preview window by clicking the pop-up menu in the upper left corner of the preview window.
- When more than one object is selected, and they all use different shaders, a separate Shader Tree Editor window displays for each shader.

For example, if you select three objects and two use the same shader, Carrara displays two Shader Tree Editors, one for each shader.

If you select two of three objects that use the same shader and start to edit the shader, Carrara prompts you whether you want to modify that shader (and apply the changes to all three objects), or clone the shader and apply the new shader with changes to only the selected objects.

The Shader Preview window shows you what the shader you are editing looks like. Each time you change the shader tree, Carrara updates the preview to show how your change affects the shader.

To display the Shader Tree Editor:

1. Select one or more objects in the Assemble or Storyboard rooms, then click the Texture room button to switch to the Texture room.
2. Carrara displays a Shader Tree Editor for each Master Shader. You can resize the Shader Tree Editor windows as needed to view the contents of the Editor. Select the control in the lower right corner and drag to resize the window.
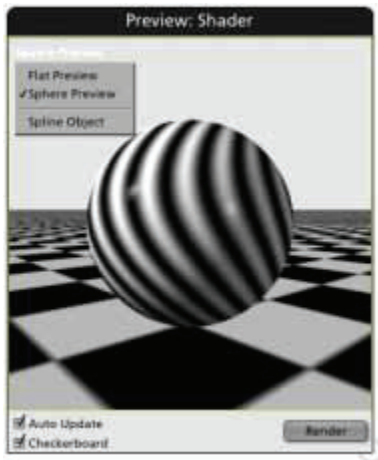
Choosing a Shader Preview Mode

You can choose a spherical preview (as shown in the previous figure), which shows you how your shader will look on the surface of an object, a flat preview, which gives you an undistorted view of 2D image data like texture maps, or a preview of any object primitives that use this shader.

To switch between spherical, flat, and object preview:

1. Click the text in the top left corner of the Shader Preview window.

2. Choose one of the preview options from the list.

This change affects only the preview of the shader you are currently editing,

Creating a New Shader

To create a new shader:

1. Switch to the Texture room.

2. Choose **Edit > New Master Shader**. This menu is also available if you use the left click on the tab that contains the master shaders.

3. Use the Shader Tree Editor tools and features to design the shader.

    -   Choose a shader type, and add a component from the lists.

        NOTE! Refer to "Shader Components Reference" for information on the shader components. Instructions for navigating and editing the shader tree appear below.

    -   Drag a shader from the Browser into an appropriate place in the Shader Tree Editor, where it becomes a sub-shader.

    -   Open a shader document from the Browser and drag its components or sub-shaders into the Shader Tree Editor.

        NOTE! If you drop a shader or sub-shader onto a place holder where its type is inappropriate, Carrara alerts you. For example, you can't put a Single Channel shader at the top of a shader tree.

4. When you're finished designing the new shader, either apply it to an object or add it to the Browser.

Adding New Shaders to the Browser

To add the current shader to the Browser:

1. Drag the shader preview from the Sequencer to the Browser.

    NOTE! You must drop the shader under a directory column.

2. Carrara prompts you to name the new shader.

3. Enter a name and any comments, then click OK.

Editing Shaders from the Browser

To open/edit a shader from the Browser:

1. Double-click on the shader you want to edit. Carrara opens it in a Shader Tree Editor window.

2. Edit the shader by adjusting sliders and other settings.

    NOTE! In the Browser, Carrara saves shaders as documents.

NOTE! If you don't see the Shader Tree Editor window right away, it might be hidden under one of the floating palettes such as the Sequencer or Browser trays.

You can drag and drop components or sub-shaders from another Shader Tree Editor window or other open shader documents.

You can also add or change components using the lists in the Shader Tree Editor. The lists display appropriate choices that you can use for the level of shader tree you are currently editing.

Adding Modified Shaders to the Browser

To save the modifications to the shader:

1. Make sure the Shader Tree Editor window for the shader you want to save is the active window.

2. Choose **File > Save.**

3. Carrara saves the changes into the Browser document file.

4. If you want to save the shader under a different name:

   - Drag the preview into the Browser.

   - Enter a new name.

5. Close the Shader Tree Editor window when you're done.

Choosing a Top Shader

The properties that you can assign to a shader vary, depending on the top shader that you select. Carrara offers several different types of top shaders that you can use to build your materials.

To specify a top shader:

- Choose a shader from the pop-up menu such as Multi Channel, Multi-Channel Mixer, or Layers List. This replaces the current contents of the Shader Tree Editor with an empty shader tree of the appropriate type.
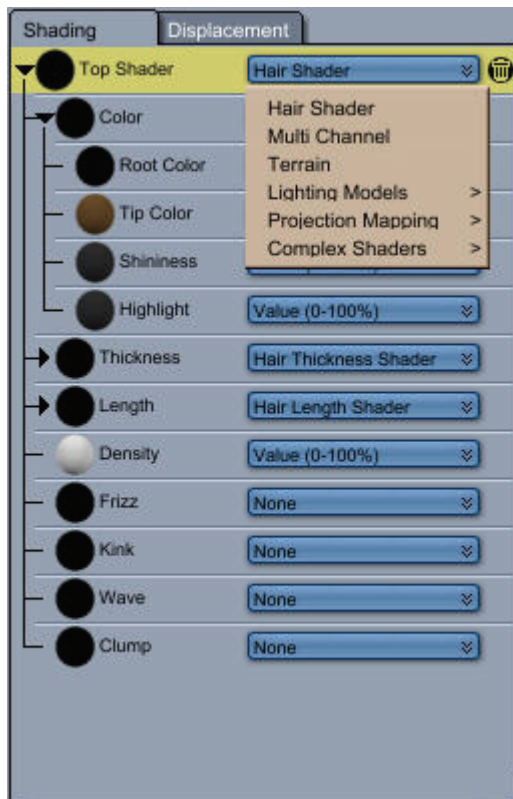
Hair Shader (Standard, Pro)

Hair Shaders are specifically used to control properties of dynamic hair. Further information on the Hair Shader can be found in "Using Hair Shaders and Presets."

- The Hair Color shader is discussed in more detail in "Controlling Hair Color with Shaders" and "Using Hair Shaders and Presets."
- Hair Thickness, Length, Density, Frizz, King, Wave, and Clump shaders are discussed in more detail in "Styling Hair with Shaders."
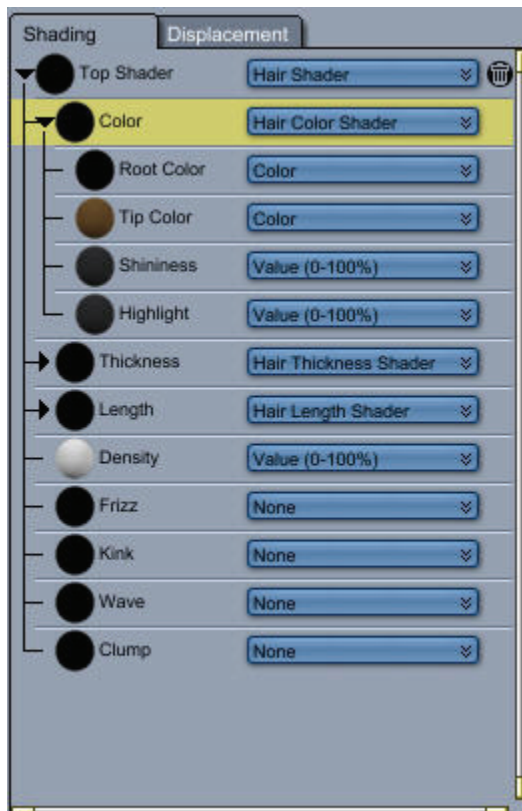
Multi Channel Shader

Choose a Multi Channel shader when you want to create a shader that combines several different shader properties. This shader allows you to combine color, alpha, highlight, shininess, bump, reflection, transparency, refraction, glow, subsurface scattering, and translucency, or any combination of them.
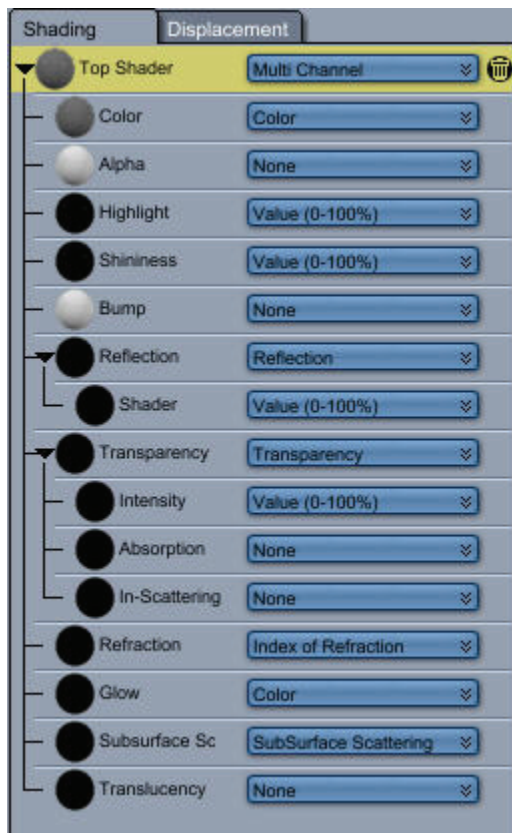
Terrain Shader

Terrain shaders are commonly used to control the appearance of the terrains in your scene. When you choose Terrain as a top shader, the Texture room displays several properties in the right panel of the Shader editor. This allows you to control the size, amplitude, and roughness of the terrain, and also allows you to choose the type of global shader you want to assign to the terrain.

Though you can assign any shader component or function to the terrain's Global Shader, the Terrain Distribution shaders are most commonly assigned. See "Terrain Distribution" for further information on the terrain functions.
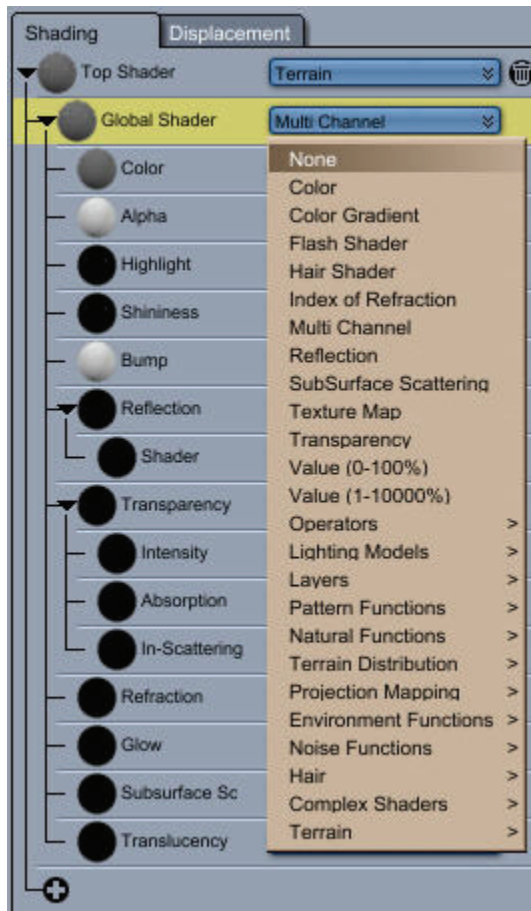
Projection Mapping

Projection mapping shaders relate to how an object takes a flat texture map and applies it to the object or part of an object. For further information, see "Projection Mapping Shaders."

Lighting Models

A lighting model is a special shader that defines the way light interacts with the material. If there is no lighting model in your shader tree, the default lighting model (Phong) is used. A lighting model should be placed at the root of the shading tree. It can also be placed in a layer as long as there is no lighting model above it (only the top lighting model is used).

There are currently three lighting models in Carrara: Anisotropic, Phong, and Shadow Catcher.
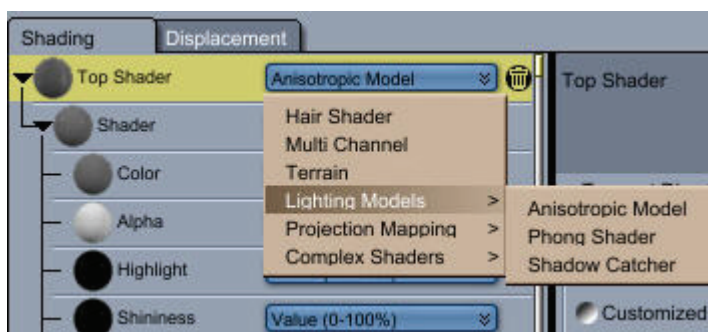
Anisotropic

Anisotropic simulates a material that reflects light differently depending on the direction. It can be used to simulate brushed metals, for instance.
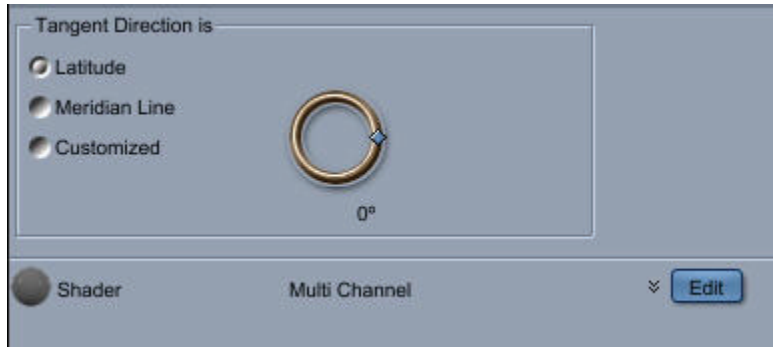
Anisotropic lighting allows you to set the direction of the highlights in the shader, through the Tangent Direction settings:

- **Latitude**: Highlights run in a north/south (or up/down) direction.

- **Meridian Line**: Sets a main direction of the material that is perpendicular to the latitude.

- **Customized**: Choose this option to set the direction with the direction indicator, relative to the iso U and iso V lines (line of constant U / V). The angle is the angle relative to the iso U line, with a value of zero meaning that the direction is set by the iso U line.

Phong

Phong is the default lighting model that is used if you do not choose an alternative lighting model. There are no additional properties to set other than to select the shader that it controls.

Shadow Catcher

Shadow catcher is a special shading mode that is used to cast shadows on a background object (see "Casting Shadows on Background Objects").

Layers List

Using a Layers List component as the root of your Shader Tree enables you to use Layer Shaders and Mapping Shaders, and add any number of Layer Shaders to the object.

When you apply different textures to areas of an object, you are using a Layers List shader. The layers in a Layers List shader are stacked, as if one was painted on top of another. You can modify the layer ordering within the list.

Carrara specifies layers either by the shading domains of the object or by the use of layer components. Refer to "Shading Domains" or "Objects with Multiple Shading Domains" to learn more.

Multi Channel Mixer

The top level of a Multi-Channel Mixer shader tree has only one branch, which affects all of its shading channels. The Multi-Channel Mixer operator always appears on that branch. The Multi-Channel Mixer operator functions identically to the Mixer operator, except that it mixes complete shaders rather than individual shader components. These complete shaders can be Multi-Channel shaders, as in the example above, part of Layered List shaders, or other Multi-Channel Mixer shaders. You can achieve complex shading effects by nesting Multi-Channel Mixer shaders.

A Multi-Channel shader contains eight channels. You can use any type of shader component in each of those channels. The settings within the different channels combine to determine the shader's overall appearance, but settings in one channel have no effect on the settings in the other channels. Refer to "Shader Channels Reference" for more information about the various types of shader channels.

Suppose you want to create a shader that mixes attributes in all the shading channels at once. For example, you might want to create a checkerboard pattern with alternating squares of shiny, reflective gold and rough granite. To accomplish this with a Multi-Channel shader, you would need to apply the same mixer function to each channel. Fortunately, Carrara provides another type of tree structure, the **Multi-Channel Mixer**, which makes it easier to achieve effects like this. The diagram below illustrates how a Multi-Channel Mixer shader works. Any component you place under the Multi-Channel Mixer affects all the shading channels.
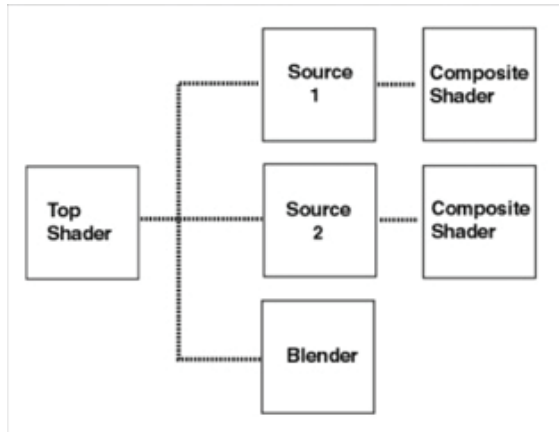
The checker pattern applied to this object has alternating squares of gold and granite. It was created using the Multi-Channel Mixer shader.
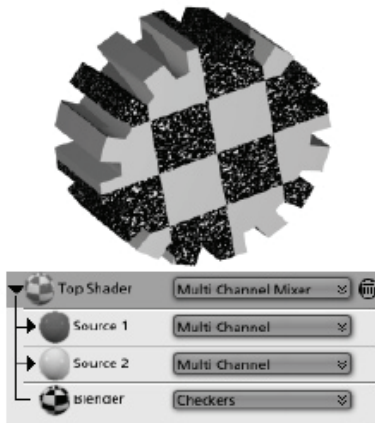
Reference Shader

A reference shader component allows you to specify an existing shader by reference. When you edit that other shader, Carrara automatically updates all references to it.

For example, if you created a series of wine bottles with different labels, you could set up the glass in the bottle to be a reference shader. Then when you edit the Glass shader, Carrara automatically updates all of the bottles that refer to that shader.

To specify a reference shader:

1. In a Layers List shader, select Reference Shader.

2. Select one of the shaders listed in the Shaders list.

Shader Recipes

Here are a few shader recipes to get you started creating your own shaders.

When using refraction, you should have something behind your object to refract. You can set a background in the scene settings.

Chrome

Chrome is a simple shader to create yet it can create interesting effects in a scene as the chrome object reflects what surrounds it.

To create chrome:

1. Create a Multi-Channel shader.

2. Set Color to Black.

3. Set Highlight to White.

4. Place a Value slider in the Shininess channel, and set it to 9%.

5. Place a Value slider in the Reflection channel, then set it to 90% or 100%.

Ice

Ice is another simple shader to create. Simply select a shade of blue, and set a value in a few of the channels.

To create Ice:

1. Create a Multi-Channel shader.

2. Set Color to a black.

3. Place a Value slider in the Highlight channel and set it to 35%.

4. Place a Value slider in the Shininess channel, and set it to 15%.

5. Place a color in the Transparency channel, then set it to an aqua (H-52, L-90, S-100).

6. Place a Value slider in the Refraction channel, then set it to 19%.

7. Set Glow to Black.

Pine

Pine uses a Mixer in the color channel. You can change the colors used to create other types of wood.

To create pine:

1.   Create a Multi-Channel shader.

2.   Set Color to Mixer.

3.   Place a Value slider in the Highlight channel and set it to 20%.

4.   Place a Value slider in the Shininess channel and set it to 7%.

5.   Set Glow to color, and select Black.

To set up the Color channel Mixer:

1.   Set Source 1 to light yellow.

2.   Set Source 2 to light tan.

3.   Set Blender to the Wood function and set the following values:

4.   Set Global Scale to 50%.

5.   Set Undulation to 37%.

6.   Set Vein Count to 6.

7.   Set Vein Blending to 50%.

8.   Set Perturbation to 100%.

Beach Ball

This beach ball is created by using two Mixers and the Checkers shader. You can add more colors to your beach ball by placing another Mixer in the Source 2 channel (where yellow is currently selected) and selecting two colors. Other than selecting the new colors, you would set it up like the Mixer placed in the Source 1 channel.

To create a beach ball shader:

1. Create a Multi-Channel shader.

2. Place a Mixer in the Color channel.

3. Place another Mixer in the Source 1 channel.

4. Place the color yellow in the Source 2 channel.

5. Set Blender to Checkers, and set the Checkers options as follows:

   • Set Squares H to 12.

   • Set Squares V to 0.

6. Place a Value slider in the Highlight channel and set it to 89%.

7. Place a Value slider in the Shininess channel and set it to 3%.

8. Place the color black in the Glow channel.

To set up the secondary Mixer in the Source 1 channel:

1. Set Source 1 to Blue.

2. Set Source 2 to Red.

3. Set Blender to Checkers and set the following options:

4. Set Squares H to 6.

5. Set Squares V to 0.

Shader Channels Reference

Shader channels define several attributes of an objects appearance. Shaders can consist of one or more of the following types of channels: color, alpha, highlight, shininess, bump, reflection, transparency, refraction, glow, subsurface scattering, and translucency. For more information about shader channels, refer to "Shader Channels Reference."

When shading is calculated, several shading channels are evaluated: Color, Alpha, Highlight, Shininess, Bump, Reflection, Transparency, Refraction, Glow, Subsurface Scattering, and Translucency.

To specify shader settings, you add components to the shader tree in one or more of the channels. The components beneath a particular channel represent that channel's settings. Depending on the components you use, the settings in each channel can be simple or complex. The Color channel might specify either a plain color or a multi-color pattern. Likewise, the Reflection channel might specify uniform or varying levels of reflectiveness across the surface of an object. You can also place functions in a channel. For example, you could place the Marble or Wood functions in the Color channel, instead of a color.

A shader need not contain settings in all of the channels. If you don't want to define a particular shading attribute, you can simply leave that channel empty.

In addition to the shading channels, the shader also defines **displacement mapping** that is applied to the surface of the object. The displacement mapping uses a function defined on the surface (shader) to displace the surface of the object. To learn more about displacement mapping, refer to "Displacement Mapping."

Color

No shading channel has a more obvious effect on the appearance of an object or layer than the Color channel. Depending on the components you place beneath the Color channel, you can specify anything from a plain color to a complex, multi-color pattern.

- To specify a plain color, simply place a Color component.

- To specify a multi-color pattern, use a Texture Map component or combine colors with the Mixer operator.

If you find that your single-color objects look unrealistic, try using the Mixer operator with the Spots function to create minute color variations. Simply mix two slightly different hues in a pattern of tiny spots, with Blending set to 0 for smooth blends. When viewed from a distance, these subtle color variations can give an object a more realistic appearance.

The default content of the Color channel is a Color component set to a shade of gray. Carrara assigns the default content of each channel, as part of the default primer, to each new object you create. Carrara also uses the default content to calculate the preview of a shader in the Shader Tree Editor when a particular channel is empty.

Alpha

The alpha channel controls the opacity of the material. An alpha value of 0% will make the shader fully transparent, whereas an alpha value of 100% will make it fully opaque.

The alpha channel differs from the transparency channel in the way the transparency is calculated:

- The transparency is additive, which means that adding transparency to an object will only make it brighter.

- The alpha creates a fading effect: as the alpha decreases, the object disappears.

The alpha channel is usually used for compositing, whereas the transparency channel is used to simulate transparent materials (such as glass).

Texture Map with Alpha Channel

Texture map files can contain an alpha channel. When you put such a texture map in the color channel, its alpha channel is multiplied by the value in the alpha channel to calculate the alpha of the material. So if the alpha value is 100% and you put a texture map in the color channel, its alpha channel will be directly applied to your material.

Highlight and Shininess

Most objects in the real world show highlights when illuminated. These bright spots or streaks are direct reflections of light sources, like the sun glinting off a chrome bumper.

Metallic objects have small, bright highlights. Plastics have dim, but large, highlights. And stones usually have no highlights, unless polished.

You can control the color, intensity, and size of an object's highlights by placing shader components beneath the Highlight and Shininess channels.
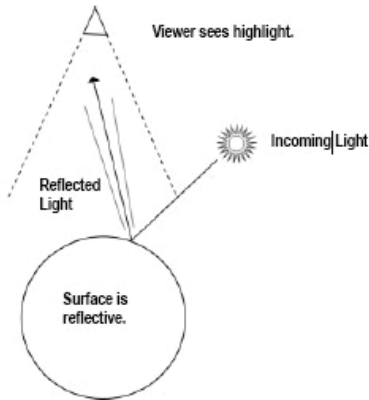
The Highlight channel controls highlight intensity. A high value produces bright highlights; a low value produces dim highlights.

The Shininess channel controls highlight size. A high value produces small highlights; a low value produces large highlights.

Note Typically, highlights that are extremely bright are very small; those that are softer spread larger.

By default, highlights are white. You can specify color highlights by placing a Color component (or any component that produces color information) in the Highlight channel. Color has no effect on the Shininess channel; any color information in the Shininess channel is internally converted to values.

The default content of both the Highlight channel and the Shininess channel is a Value component set to a value of 50.

If you want varying levels of highlight or shininess across the surface of an object or Layer, you can use a texture map or a function.

Bump

Bumpy and rough surfaces are difficult to model geometrically. Imagine modeling the irregularities of the skin of an orange, point after point:the task would quickly become a nightmare. Fortunately, Carrara allows you to use a shader to simulate variations on the surface of an object.

By placing shader components in the Bump channel, you can "perturb" a smooth surface to produce irregularities. This technique is known as bump mapping. Bump mapping can produce subtle effects like the bumps and pits on the surface of an orange, or more pronounced effects like the seams on a baseball.

Bump mapping does not change the actual topography of an object:no points are displaced. Rather, it changes the angle at which light rays reflect off the surface at certain points. This technique tricks the eye into perceiving texture. Because the bumpiness is just an illusion created by the renderer, the edge of the object remains smooth. In most cases, this is not a problem. However, if you require true surface variations to appear on the silhouette of your object, you must create them in the modeler.

To simulate bumps, the components you place beneath the Bump channel must specify some variation in the colors or values of adjacent pixels. Consequently, placing a simple color or value in the Bump channel does not produce bumping.

Texture maps and functions produce the best results in the bump channel. As with the other non-color channels, the Bump channel always converts color information to grayscale.

When you use a grayscale image in the Bump channel, lighter areas appear to be raised; darker areas appear to be lowered. When you use a function that produces values, areas with higher values appear to be raised; areas with lower values appear to be lowered.

- For the most convincing results, you should create a smooth blend between extreme values like black and white or 0 and 100.

- If a grayscale image with hard edges between black and white does not produce a satisfactory bump effect, try applying a blur effect to the image in an image-editing application.

- The relative height or "steepness" of the bumps depends on how rapidly you blend between extreme values. A blend that occurs over the range of many pixels produces shallow, gradual bumping. A blend that occurs over the range of a few pixels produces steeper, more sudden bumping.

The default content of the Bump channel is a Value component set to a value of 0. This produces no bumping because there is no variation from pixel to pixel.

To change the amount of bump:

1. Adjust the Bump Amplitude slider located after the glow channel in a multi-channel shader.

2. This slider lets you strengthen or weaken the Bump effect.

3. Apply your changes to the object. Use the Test Render tool to check the results on the object.

Reflection

Many real-world surfaces are at least somewhat reflective. Most types of metal and glass are partially reflective. A mirror is so reflective that it takes nearly all of its color from the environment around it.

The reflection channel contains a **reflection shader** which controls the properties of the reflection. You can adjust the following parameters for the reflection:

- **Fresnel Term**: In most materials, the amount of light reflected from a surface depends on the angle of view. This is the Fresnel effect that can be noticed in glass, water and other transparent materials. The reflection is most intense at low angles of view and less intense when directly facing the material surface.

- **Blurry reflections**: Reflections are not always as sharp as a mirror. Most materials have much more diffuse reflection. Enabling blurry reflections allows you to simulate that effect.

- **Maximum ray depth**: Controls how many times a ray of light can bounce against the reflective surface.

- **Maximum Attenuation**: Determines when an extra ray of light should be sent for reflections depending on the attenuation of the previous bounces. This way, if you see a reflective object through a barely transparent object, the reflection is not calculated which speeds up the calculations.

Carrara allows you to specify reflectiveness by placing shader components beneath the reflection shader.
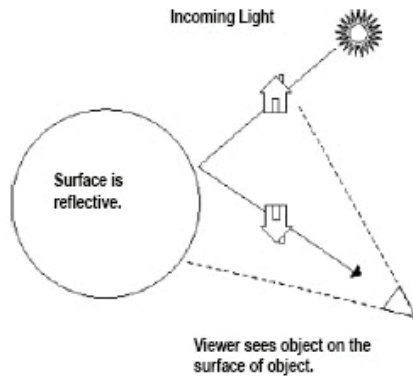
Although a Value component works fine in the Reflection channel, you can achieve both subtler and more fantastic effects by using a Color component instead.

- When you use color data (a Color component, a Texture Map component, or a complex sub-shader) in the Reflection channel, its hue affects the tint of the reflection and its brightness determines the amount of reflection. Darker colors produce less reflection; brighter colors produce more.

- Using values in the Reflection channel produces the same effect as using shades of gray. Only the amount of reflection is affected.

Of course, if you want varying levels and tints of reflection across the surface of an object or layer, you can use a texture map or a function.

The default content of the Reflection channel is a Value component set to a value of 0.

Transparency

When light strikes an opaque surface, it simply bounces off. When it strikes a semitransparent surface, some light bounces off, but some passes through. As a result, you can see through a semitransparent object. Glass, water, and clear plastic are examples of semitransparent materials.

The transparency channel contains a **transparency shader**. You can adjust the following parameters for the transparency:

- **Fresnel Term**: In most materials, the amount of light transmitted through the surface depends on the angle of view. This is the Fresnel effect that can be noticed in glass, water and other transparent materials. The transparency is less intense at low angles of view and most intense when facing the material surface.

- **Maximum ray depth**: Controls how many times a ray of light can go through the surface.

- **Maximum Attenuation**: Determines when an extra ray of light should be sent for transparency depending on the attenuation due to the previous bounces.

You can specify transparency by placing shader components beneath the Transparency shader.

Try using a function or a black and white texture map in the Transparency channel to create a shader that makes transparent "holes" in an object.

For example, placing the Wires function in the Transparency channel creates a wireframe effect.
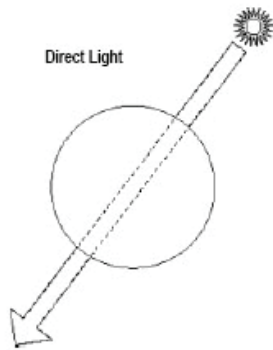
Direct Light

- When you use color data (a Color component, a Texture Map component, or a complex sub-shader) in the Transparency channel, its hue affects the appearance of colors viewed through the transparent shader. Its brightness contributes to how transparent the shader is. Darker colors produce less transparency; brighter ones more.

- Using values in the Transparency channel produces the same effect as using shades of gray. Only the amount of transparency is affected.

- A 100% transparent surface might not be completely invisible:depending on the settings in the other channels, it might still refract and show highlights.

The default content of the Transparency channel is a value component set to 0.

Refraction

When light rays pass through a semi-transparent object, their trajectories are deflected. This phenomenon is known as refraction. Glass, fluids, and other translucent materials refract light to some degree. This results in a distorted view of objects behind the refractive surface. The distorted view of the straw below was created by using refraction on the water glass.

Try a simple refraction experiment: Stand a pencil in a glass of water. Refraction "bends" the pencil where it enters the water.
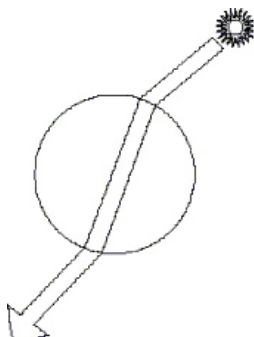
A single Value component is generally all you'll need to place in the Refraction channel.

 NOTE! The setting in the Refraction channel affects only a shader with some degree of transparency. Light must pass through to be refracted.

Shaders simulating glass should have Refraction values around 20. Shaders simulating water or ice should have values somewhat lower. Try experimenting with different values until you achieve the effect you want.

Color has no effect on the Refraction channel. Any color information in the Refraction channel is internally converted to values.

The default content of the Refraction channel is a Value component set to a value of 0.

Glow

The settings in a shader's Glow channel determine the shader's luminance. An object whose shader has a high degree of luminance will appear bright, even if it is not lit by ambient light or external light sources.

You can use glowing shaders to simulate things like neon tubing, lit windows in a cityscape at night, or the LED display of a digital clock. You can also use them for the subtler purpose of brightening a particular object in a dimly lit scene. This allows you to emphasize an object without affecting the lighting and mood of your entire scene.
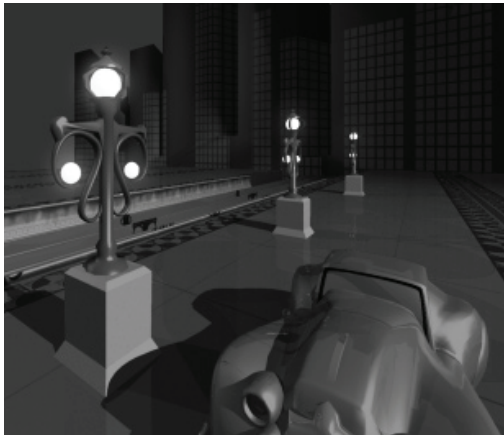
Objects with glowing shaders do not cast light on other objects in the scene.

Although you can use a Value component in the Glow channel, a Color or Texture Map component is more appropriate. For realistic results, try using similar colors in the Color and Glow channels.

To create unusual effects, you can place markedly different colors in these two channels. For example, you might create a shader which appears green when well lit, but glows red when covered in shadow.

When you use color data (a Color component, a Texture Map, or a complex sub-shader) in the Glow channel, its hue determines the color of the glow, and its brightness determines the intensity.

For a related glow technique, refer to "Rendering with an Aura."

Subsurface Scattering

Subsurface scattering is the light that scatters through the volume of translucent materials. It is most visible in semi-opaque materials such as marble, milk, wax, skin, etc. It is characterized by a very smooth and diffuse lighting that filters through the material.

Here is a quick description of the parameters used for subsurface scattering:

- **Intensity** is the intensity of the effect.

- **Translucency** is the most important parameter of the subsurface scattering shader. It represents the depth to which the light diffuses into the material. The higher the translucency value, the deeper the light penetrates into the material.

- **Diffuse Reflection** controls the intensity of the scattering at the surface.

- **Refraction Index** controls the index of refraction of the material (it has some impact on the scattering intensity) that is used for the subsurface scattering calculations.

- **Fresnel Effect** controls how the subsurface scattering and the rest of the lighting is mixed. 100% is the most physically realistic. In that case, when the surface is facing the observer, the light is 96% of subsurface scattering and 4% of light reflected from the surface. When the surface is seen from the side it is the contrary. When the slider is at 0, the subsurface scattering is simply added to the light reflected by the surface.

NOTE! Subsurface scattering varies depending on the size of the objects to which it is applied. The shader works in a way that a given shader will simulate the same material properties for all the objects it is applied to. However, if you want to get the same effect at a bigger scale, you will need to increase the translucency.

Displacement Mapping

Displacement mapping uses a shading function to displace the surface of the object. Contrary to the Bump channel that only modifies the surface normal to simulate the appearance of a bumpy surface, the displacement mapping creates actual geometry. It can be used to create all kinds of bumpy materials (bark, tiles, rocks, etc.) or simply to add details to an object.

The advantage of using displacement mapping is that you get correct shadows and edges.

To add displacement:

1. In the Assemble room, select the object that you want to change.

2. In the **Properties tray: General tab: General panel**, click the **Edit** button to open the shader editor.

3. Click on the displacement tab of the shader document. Options for displacement appear in the Shader Editor.
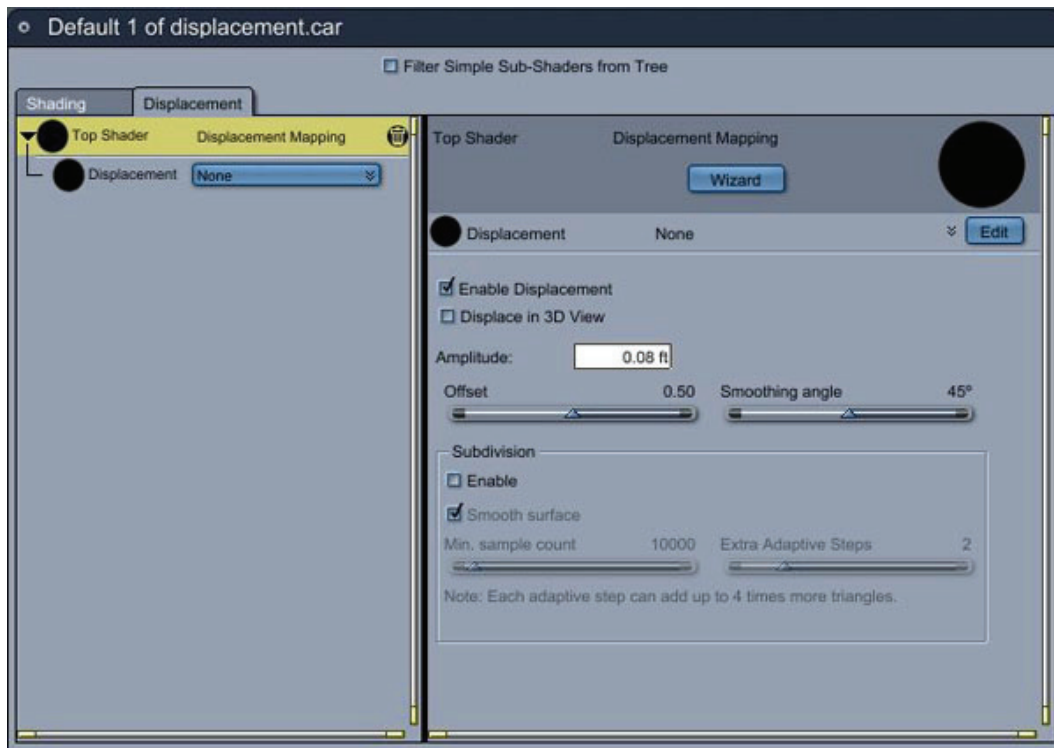
4. Choose the **Top Shader** to configure the following options:

  - **Enable Displacement**: Check this option to enable the displacement shader.

  - **Displace in 3D View**: Displacement mapping is applied to the mesh that is visible in the interactive preview (3D view). Otherwise, the displacement is only visible when rendering the object.

  - **Amplitude**: Sets the height of the displacement.

  - **Offset**: Decides which value of the shading function is 0. The offset is subtracted from the shading function so if the shading function is below the offset the displacement is done in the opposite direction of the normal.

  - **Smoothing Angle**: Controls which edges will be smoothed. If the angle between two triangles is smaller than the smoothing angle, the edge is smoothed, otherwise it is creased.

5. Enable or disable subdivision as discussed in "Using Subdivision with Displacement Shaders."

6. Choose the type of shader that you want to use for displacement. "Using Texture Maps for Displacement" describes how to assign a displacement texture map.

Using Subdivision with Displacement Shaders

To achieve good precision on the displacement, it is often useful to subdivide the mesh as discussed in "Working With Subdivision Surfaces." However, it is often better to use the native subdivision of the object. For instance, if you are displacing an object created with subdivision surface, it is recommended to increase the subdivision count instead of using the displacement shader subdivision scheme. (Standard, Pro)

The displacement shader offers its own subdivision method, which you can configure as described in the steps that follow.

To use subdivision with displacement maps:

- In the Subdivision section, configure the following settings as desired:
    - o **Enable**: Check this option to enable subdivision in the displacement shader.
    - o **Smooth Surface**: Check this option to smooth the subdivided surface before the actual displacement is applied
    - o **Min. Sample Count**: Defines the minimum number of vertices that the mesh should contain after the subdivision. This will create a uniform sampling of the mesh.
    - o **Extra Adaptive Steps**: This is extremely useful if the displacement function is smooth in most places with some big discontinuities (for instance a checked board).

NOTE! Each extra adaptive step multiplies the number of triangles by four. Increase this parameter carefully.

Using Texture Maps for Displacement

After you assign the Top Shader properties for displacement, you can assign a shader type to the Displacement channel. Click the Displacement button to select the type of displacement you want to assign.

One common option is to use a grayscale texture map to control the amount of displacement in a UV-mapped object. If you want to choose a texture map to control the amount of displacement, choose Texture Map. The appropriate items then appear in the Shader Editor.

To set properties for displacement maps:

1. Select **Texture Map** as the displacement type.
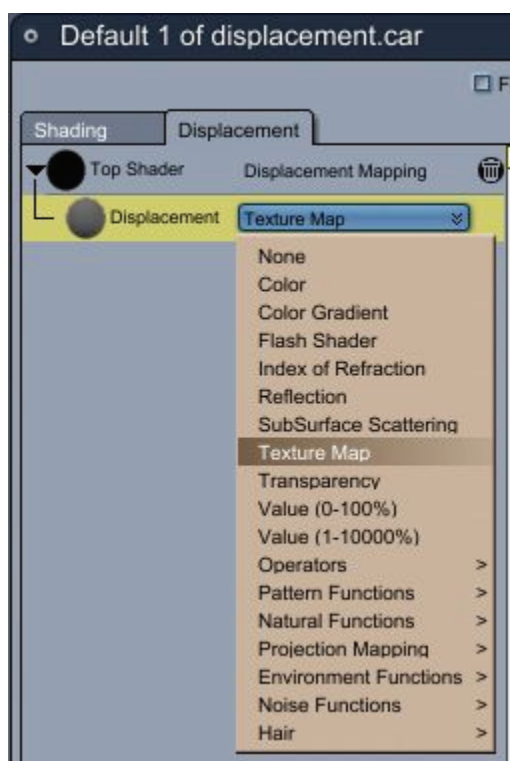
2. The shader editor displays texture map options in the right panel.

3. Configure the settings for the displacement map as described in "Texture Map."

Shader Transforms

Some shaders can be transformed before they are applied to an object. These shaders appear in the shader tree with a pair of tabs, one of which is Transform.

The Shader Transform tab allows you to move, scale, and rotate a shader. Use it for fine adjustments to the look of a shader in your scene, or for creating interesting animated effects. Transforms are particularly applicable to projection mappings and noise shaders. You can transform Cylindrical, Spherical, and Flat mapping shaders, and the transform will affect all the shaders under the mapping.
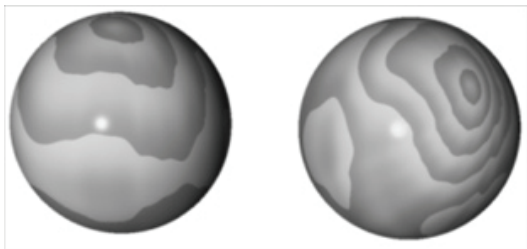
In the image above, the left sphere has a Lumber Yard shader applied. The sphere on the right has the same shader, but with the following transform applied to the Lumber Yard shader.
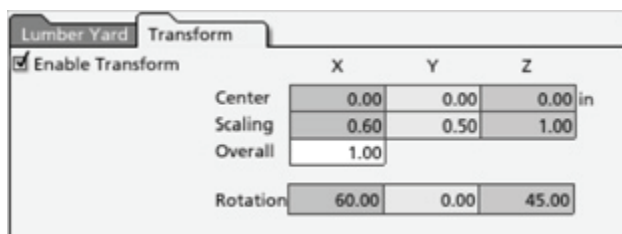
Shader Components Reference

This section describes Carrara's different shader components. It explains what each component does and how to set each component's unique parameters. Within this section, the components are divided by type into groups: basic components, operators, functions, layers, and mapping modes.

"Building and Editing Shaders" explains how to use the Shader Tree Editor to place shader components on the shader tree.

You might have received other shader components in addition to those described here. Carrara's open architecture allows developers to program their own shader components in the form of plug-ins. These third-party extensions join seamlessly with Carrara's built-in shading tools.

Shader Components

There are several types of shader components: basic components, operators, functions, lighting models, layers, and mapping components. These are discussed in more detail in "Shader Components Reference."

Basic Components

Basic components are the fundamental building blocks of shaders. Colors, values, and texture maps are examples of basic components. The diagram below shows a shader tree built entirely with basic components. Notice the structural simplicity of the tree:no branching occurs below the level of the channels. For more information on specific basic components, refer to "Choosing Components."

Operators

To create more complex and varied shading effects, you can use **operators** to combine the effects of two basic components within a shading channel. The arithmetic operators (Add, Subtract, and Multiply) combine components mathematically. An additional operator, the versatile Mixer operator, uses functions to combine components in many different ways. For more information on specific operators, refer to "Operators."

The diagram below demonstrates the use of an operator. In this particular example, the Mixer operator is used in conjunction with the Wood function to mix two colors in a wood grain pattern. Notice that placing an operator in the shader tree causes the tree to branch.
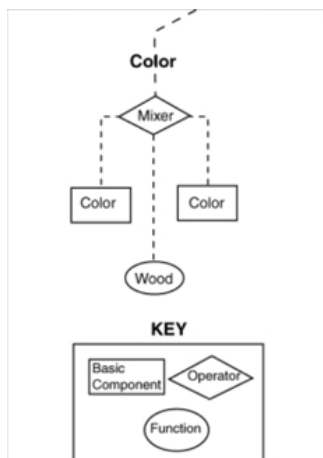
The wood grain applied to the object below was created by using the Mixer operator with two colors and a wood grain function.
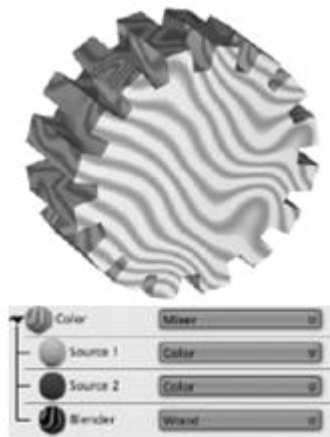
When a component such as the Mixer operator has branches extending below it, the component and all the branches below are referred to as a **sub-shader**. Sub-shaders allow you to create truly complex effects:anywhere you can place a component on the shader tree, you can place a sub-shader instead.

The shader in the example below uses the Mixer operator with the checkers function to create a complex checkerboard pattern in the Color channel. Half the squares in the checkerboard are spotted:the spot pattern is produced by a second Mixer operator, nested as a sub-shader within the first. The appearance of the remaining squares is determined by a texture map, loaded into Carrara from a 2D image file. This image file might contain a multi-color gradient or fractal pattern, for example.
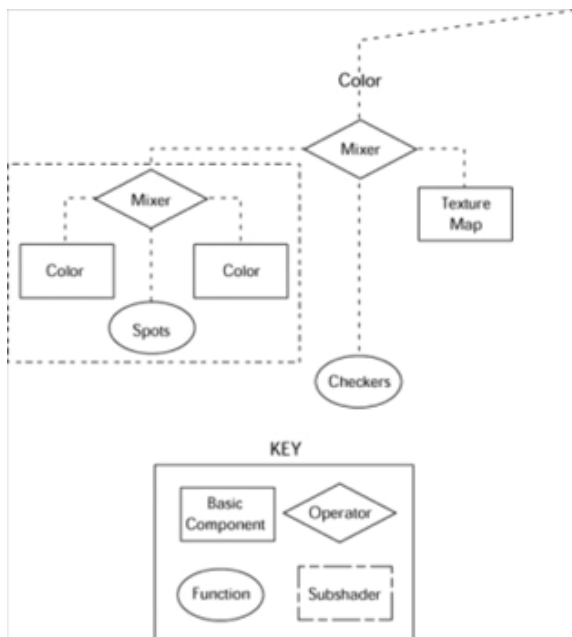
The checkerboard pattern applied to the object below contains a second Mixer operator that produces spots in one half of the squares in the checker pattern.
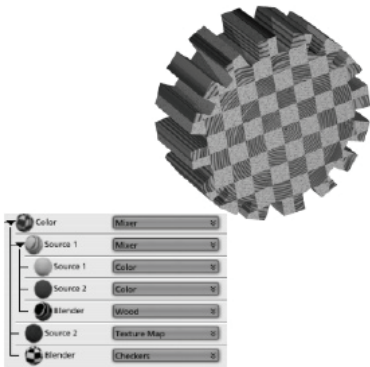
Function Components

By choosing which function to use, you control how the Mixer operator combines a pair of components. Carrara comes with several functions, including Cellular, Checkers, Wires, Wood, Spots, Gradient, Psychedelic, and Marble. You can also define your own function with a formula.

For more information on specific functions, refer to "Functions."

Layer Components

You can specify the following types of layers:

- Rectangular
- Elliptical
- Polygonal

Each Layer uses two sub-shaders:

- A primary sub-shader used to define the appearance within this layer
- An opacity mask sub-shader used to define where the primary shader is applied within the layer

Color Pickers

Carrara provides several different types of color pickers you can choose from when you are selecting colors for shaders or lights. In most cases, the color picker is in a floating window that can remain open as long as needed.

Your choice of a color model is for your convenience only:it has no effect on the contents of your final rendered image. As with other rendering applications, Carrara always renders RGB images. You can use an image processing application to convert rendered images to CMYK for separation.

To select a Color Picker type:

1. Click a color chip to display the Color Picker dialog.

2. Select a color picker type from the list by clicking the name in the upper left corner. The available color pickers are:

    - CMYK Sliders
    - Color Mixer

- HLS Wheel

- Lab Sliders

- RGB Sliders

- RGB Wheel

- Web Colors

3. Depending on which color picker you choose, adjust the value in one of the following ways:

- Type a value

- Drag the sliders

- Click a color to select it

CMYK Sliders

To use the CMYK Sliders Color Picker:

1. Two color squares appear in the color picker. The upper square is the color that is currently assigned. Change the C, M, Y, and K values using one of the following methods:

- Drag the slider left to decrease the setting or right to increase the setting.

- Click the numerical value that appears at the right of the slider and enter a new value.

2. Your color will update in your object.
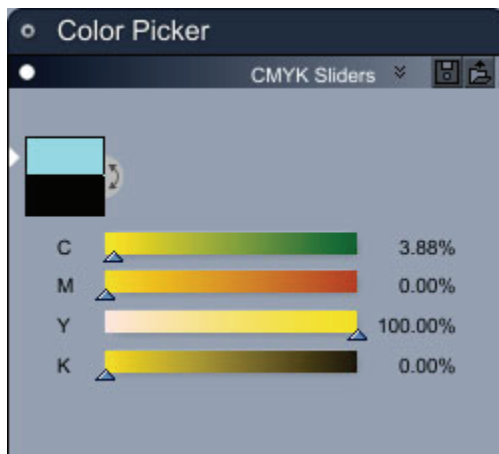
NOTE! If you are using CMYK, the closest color match will be displayed in the color picker window.

Color Mixer

The color mixer allows you to mix four colors together to create a blend of colors that you can choose from.

To use the Color Mixer:

1. Click the first corner color square that you want to change.

2. Click in the vertical HSV color bar to choose a color.

3. The saturation bar at the right of the vertical HSV color bar updates to reflect your color choice with varying degrees of saturation. Click in the Saturation bar to increase or decrease saturation.

4. Repeat steps 1 through 3 for each corner color that you want to change.

5. Click inside the large blended color area to choose your final color. The currently selected color displays in the top color chip on the left.
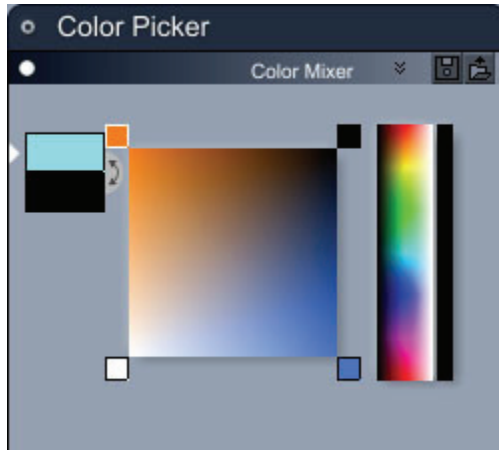
HLS Wheel

The HLS Wheel allows you to choose a color based on hue, light, and saturation values. You can enter HLS values manually or choose a color from the color matrix.

If you want to reproduce a specific color and know its HLS values, enter the values manually in the H, L, and S text boxes.

To visually select a color with the HLS Wheel:

1. Use the outer ring to select the hue. Drag the color marker around in the circle to select the hue value.

2. Adjust light and saturation for the color inside the color triangle as follows:

    •   Drag the color indicator upward or downward to make the color lighter or darker.

    •   Drag the color indicator toward the left to desaturate the color, and toward the right to make the color more saturated.
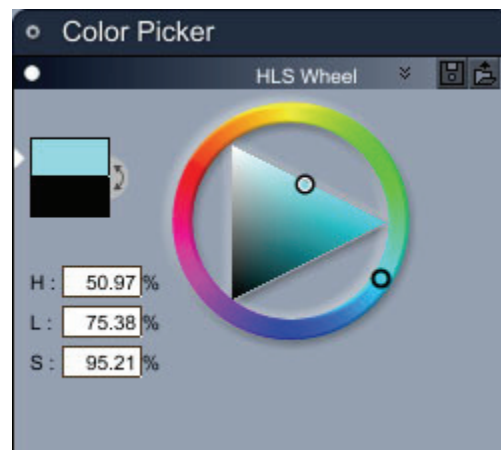
Lab Sliders

The Lab Sliders color mixer allows you to create colors that approximate those seen in human vision. The L component is used to adjust lightness contrast, and the A and B components are used to adjust the balance of green/red and blue/yellow opponent colors.

To visually select a color with the Lab sliders:

1. Set the Light value with the **L** slider, or enter the desired value in the text field at the right.

2. Use the **a** slider or text field to select the balance of green to red.

3. Use the **b** slider or text field to select the balance of blue to yellow.

RGB Sliders

The RGB Sliders color mixer allows you to create colors based on the Red, Green, and Blue values. Each slider can be adjusted to a setting of 0-255.

To visually select a color with the Lab sliders:

1. Set the Red value with the **R** slider, or enter the desired value in the text field at the right.

2. Set the Green value with the **G** slider, or enter the desired value in the text field at the right.

3. Set the Blue value with the **B** slider, or enter the desired value in the text field at the right.

RGB Wheel

The RGB Wheel allows you to select an RGB color within a wheel, and then modify the color based on HSV values. You can also adjust lightness or darkness with a Value bar at the right side of the color picker.

To visually select a color with the RGB wheel:

1. Position the color selector in the RGB wheel to select the color that most closely approximates the color you want to pick.

2. Adjust Hue, Saturation, or Value (lightness/darkness) with the HSV number fields in the left side of the color picker.

3. Use the vertical Value bar at the right side of the color picker to make the color lighter or darker.
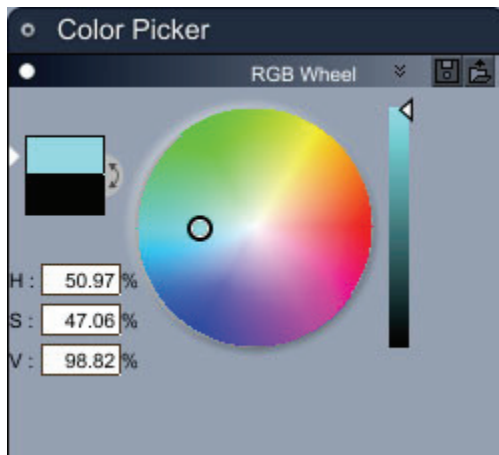
Web Colors

The Web Colors color picker allows you to choose from colors that are most compatible with Web browsers. Only web-safe colors appear in this color picker.

To select a web-safe color:

1. Click inside the color selector area to find the color that is closest to the color you want to use.

2. Other web-safe colors that are close to the selected color appear in a horizontal bar beneath the main color selector area. Choose another alternative that is more suitable, if desired.

3. The color formula for the web-safe color appears in the text field in the left side of the color picker.

4. Web-safe colors always consist of three pairs of the numbers 00, 33, 66, 99, CC, or FF. You can enter numerical values other than these in the text entry box, but the color will no longer be "web safe." The closest color match will be displayed in the color picker window.
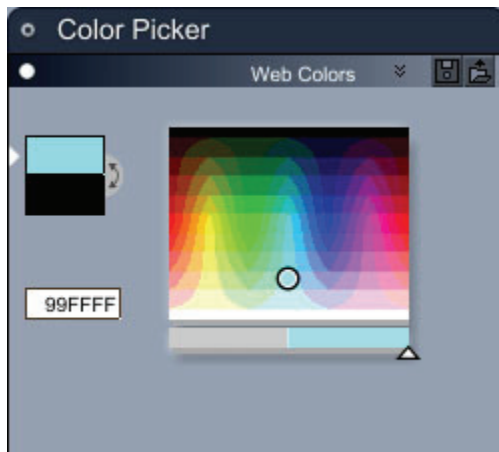
Choosing Components

There are several components that you can assign to a shader channel. The basic components at the top of the components list are frequently used for a number of different types of shader channels.

To choose a component from the Shader Tree List:

1. Select a channel (such as Color, Alpha, Bump, or other) in the Shader Tree List.

2. Click the menu button to choose a component from the menu.

3. Set the options for the selected component in the right portion of the shader tree editor.
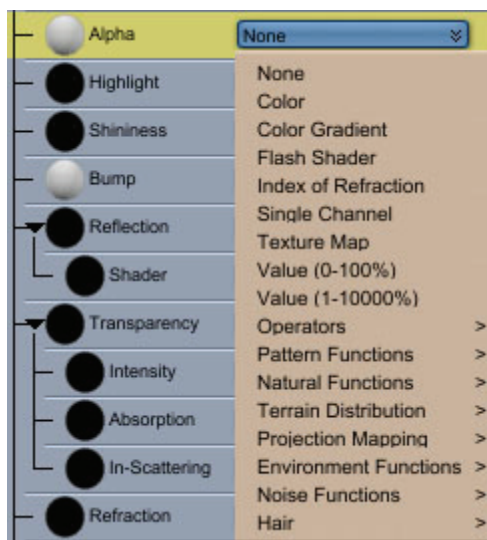
You can also use controls in the Shader Tree Editor to choose a component for the selected shader channel. There are three controls in the right portion of the Shader Tree Editor:

- **Selection Menu (1)**: Click the arrow to choose a shader component using a menu similar to that shown in the previous figure.

- **Wizard (2)**: Click the Wizard button to select a shader preset from the Scene Wizard.

- **Parent (3)**: Click the Parent button (available in a multi-channel shader) to move up one level in the Shader tree to view all shader channels contained in the parent shader.

Common Shader Components

None

Choose **None** when you do not need to assign a component to a shader channel. Existing channel information will be removed.

Color

The Color component allows you to choose any color from any of the color pickers, including a small color-picker that appears when you click the arrow at the right of the color chip.
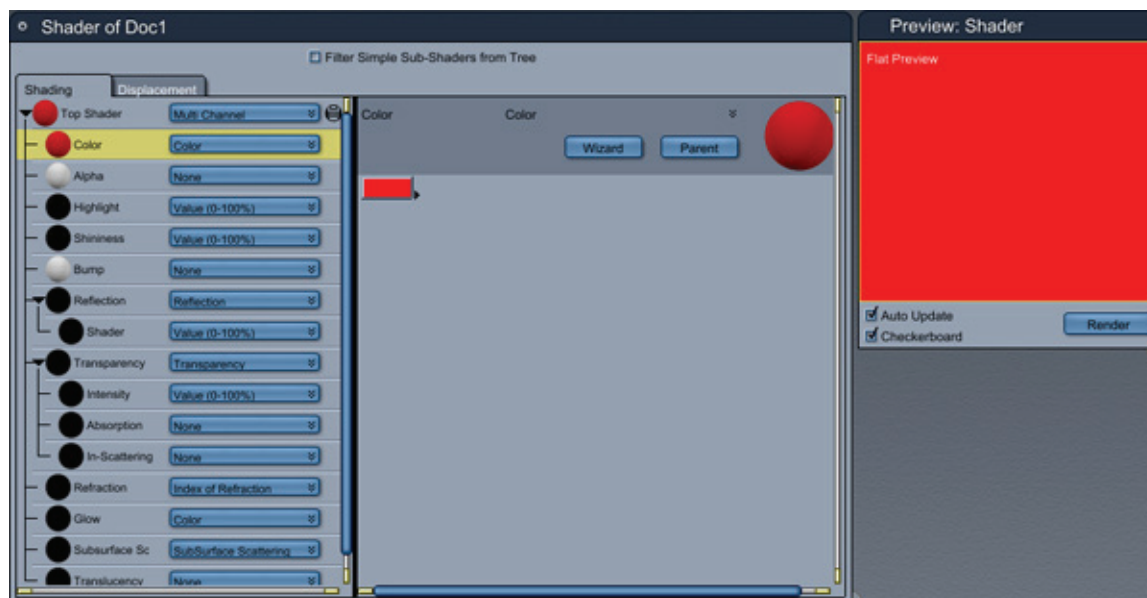
To use the Color component:

1. Select the object for which you want to create a shader and switch to the Texture room. The Shader Tree Editor displays the current shader.

2. Locate the channel that will use the Color component.

    -   Though you can place the Color component anywhere on the Shader Tree, it is best suited for use in the Color, Highlight, Reflection, Transparency, and Glow channels (which are designed for Color input).

    -   A Color component will not produce an effect in the Bump channel, because it gives a constant value across the entire surface. To create the illusion of bumpiness, the Bump

channel requires variation across the surface of an object. For a thorough discussion of the Bump channel, refer to "Bump."

- In a non-color channel such as Shininess or Refraction, Carrara converts colors to values. Dark colors convert to low values, and light colors convert to high values.

- For further discussion of how colors affect each channel, refer to "Shader Channels Reference."

3. Click the component selector menu, and choose Color from the options that appear.
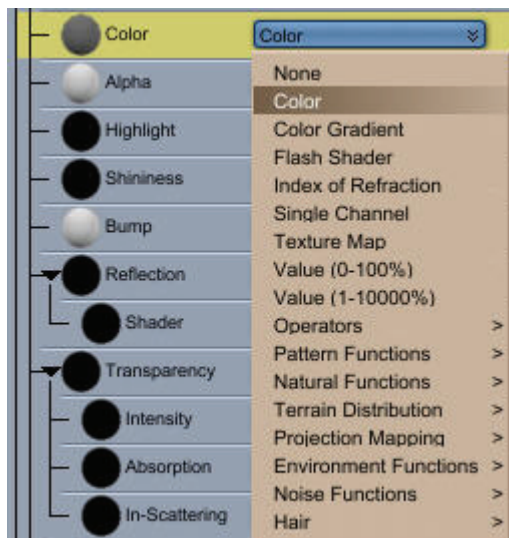
4. Choose your color using one of the following methods:

- Click the color chip in the right section of the Shader Tree editor, and select a color from the Color Picker that appears.
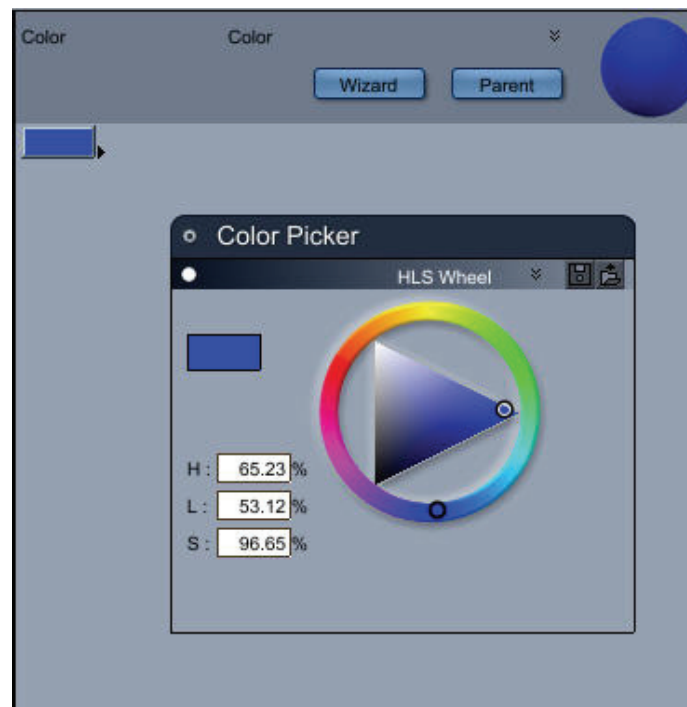
- Click the arrow to the right of the color chip to select a color from a mini-color picker.

Color Gradient

The color gradient shader is used to transform a black and white mapping into a color mapping. It uses two parameters:

- The first parameter defines the colors in the gradient.

- The second parameter defines the black and white shader that you want to transform to the color mapping.

The Color Gradient can be animated. Modifying or adding a key at a time that is different from 0 will create a keyframe.

To use a Color Gradient component:

1. Select the object for which you want to create a shader and switch to the Texture room. The Shader Tree Editor displays the current shader.

2. Locate the channel that will use the Color Gradient component.

3. Click the component selector menu, and choose **Color Gradient** from the options that appear. Two parameters now appear in the tree: A Color Gradient parameter, and a Shader parameter beneath it.
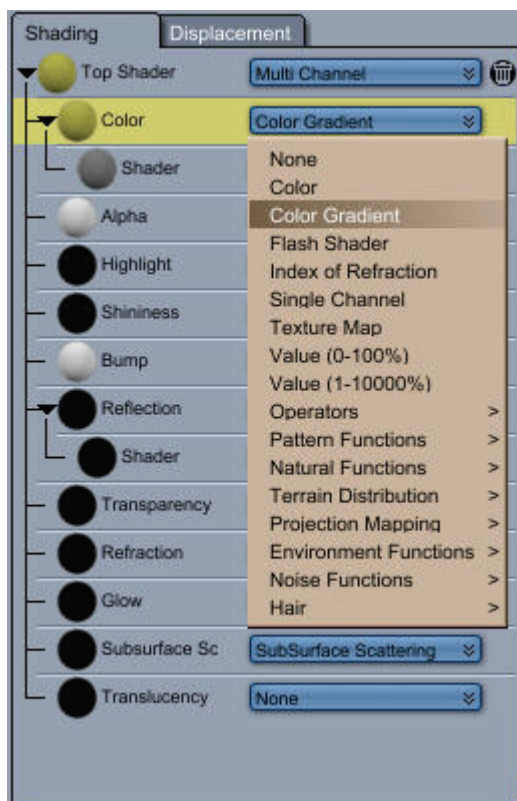
4. Select the channel that has the Color Gradient assigned to it. The parameters appear in the right pane of the Shader Tree editor.

5. Click the gradient to define the colors that you want to use in the gradient. Use the Gradient Picker as follows:

    - Two or more keys define the colors in the gradient. At a minimum, there is one color key at the beginning of a gradient, and another at the end.

    - **To change the color of a key**, select the key to be modified. Then change the color using the color picker in the Gradient Picker.

    - **To modify the position of a key**, either click and drag the key to a new position, or enter a value (between 0 and 1) in the text field at the right side of the gradient editor.

    - **To add a new key to the gradient**, click in the area of the color gradient to set the position of the new key. You can then add a key in one of three ways: Click the 'Add key' button, modify the color with the color picker, or click in the graduated scale area to add the key, then modify its color.

    - **To delete a key**, select the key to be deleted then click on the 'Del key' or use the Delete or Backspace keys of the keyboard.

6. After you define your gradient, choose the Shader (the black and white image or procedure that you want to use in conjunction with the gradient). You can select the Shader component in one of two ways:

    - From the Shader component (shown directly beneath the Color Gradient component in the Shader tree).

- Click down arrows at the left of the **Edit** button in the Shader Tree editor.
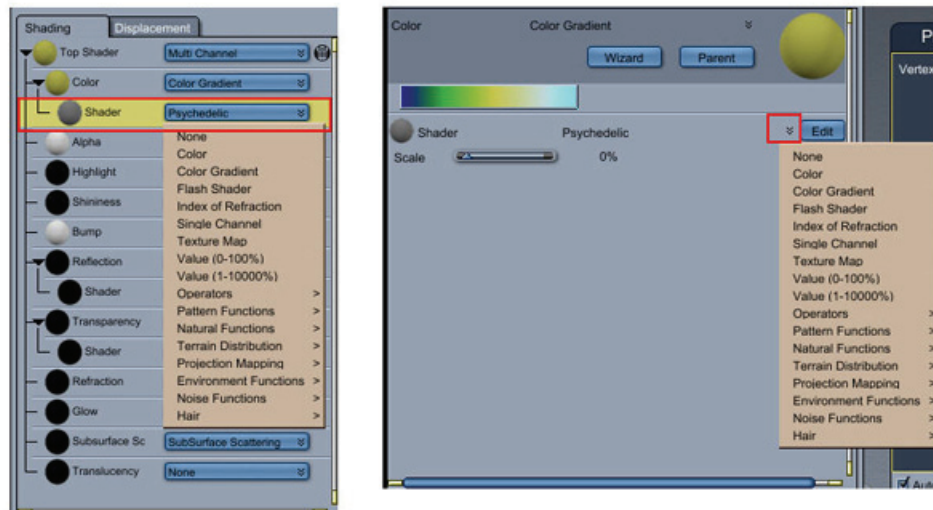
7. After you configure your Color Gradient and Shader, you can adjust the settings for both as needed to refine your shader. The following images show an example of the results when a Color Gradient is used in conjunction with a Psychedelic component for a shader.
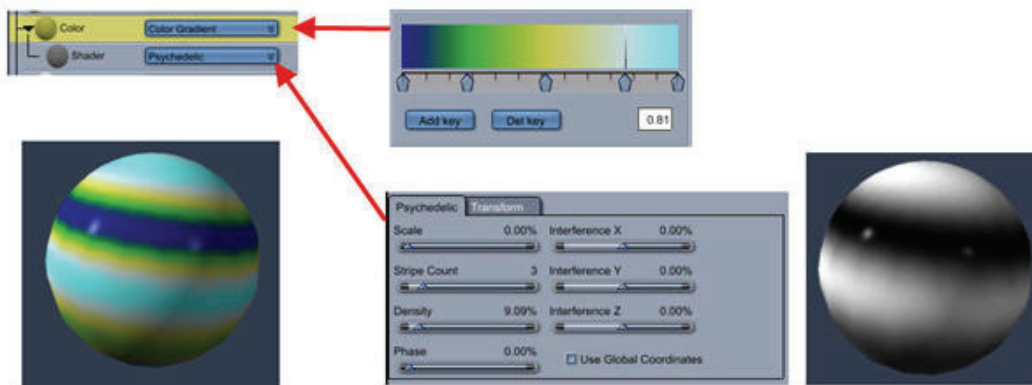
Flash Shader

The Flash Shader component allows you to use a Flash animation (SWF extension) as a shader. However, in order for this shader to be visible in your final animation, you must export the finished animation to the Viewpoint Experience Technology (VET) format (Standard, Pro).
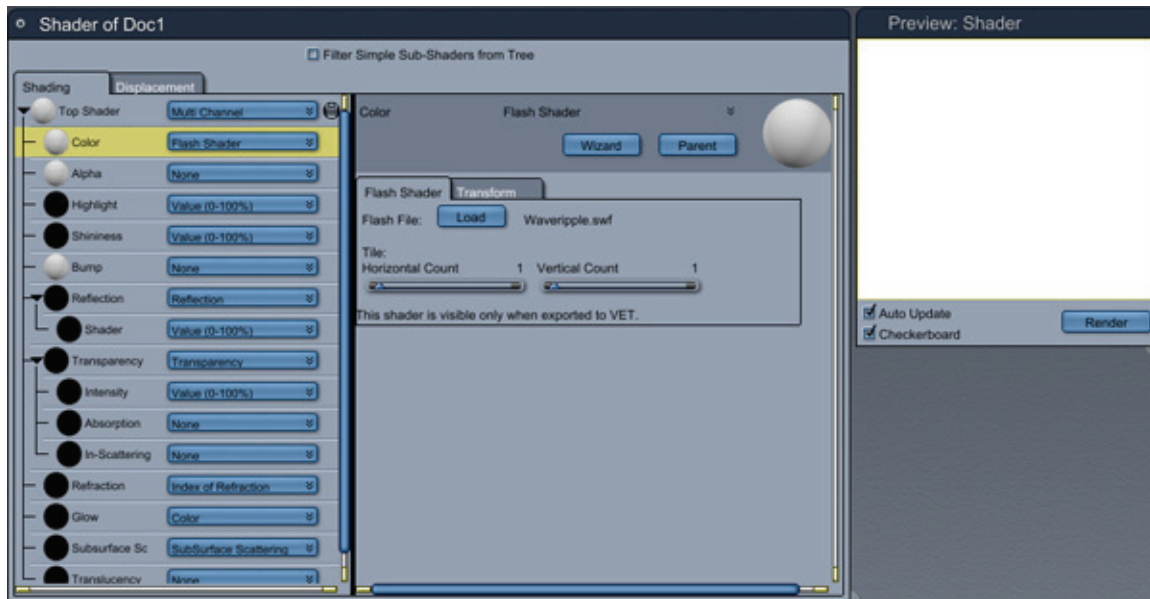
To assign a Flash Shader component to a shader:

1. Select the shader channel that you want to configure.

2. Choose **Flash Shader** from the component menu. The options appear in the right pane of the Shader Tree Editor.
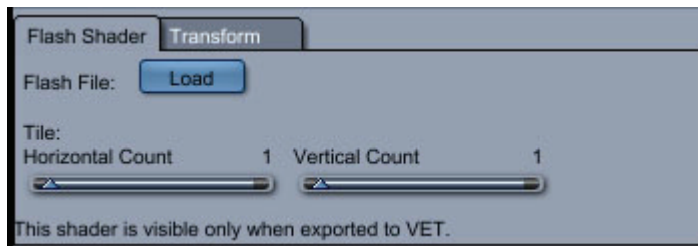
3. In the Flash Shader tab, choose the following options:

    - **Flash File**: Click the Load button to locate the Flash (SWF) file that you want to assign to the shader.

    - **Tile**: By default, the Flash file repeats once across the entire width and height of the object to which it is assigned. You can use the Horizontal Count and Vertical Count sliders to increase the number of times that the Flash file is repeated.

4. In the Transform tab, choose the following options:

    - **Enable Transform**: Check this option if you want to reposition, rescale, or rotate the SWF file.

    - **Center**: To change the center point for the SWF file, enter the new coordinate(s) in the X, Y, or Z fields.

    - **Scaling**: To change one or two dimensions of the SWF file, enter the new size in the X, Y, or Z fields.

    - **Overall**: To change the size in all three dimensions at once, enter a new value in the Overall field.

    - **Rotation**: To rotate the SWF file in one or more directions, enter the degrees of rotation in the X, Y, or Z fields.

Index of Refraction

Light changes speed and bends when it passes through a transparent medium like glass or water. A good example of this is when you place a straw in a glass of water. The straw appears to bend in a different direction above the water than it does inside the water.

The amount of bend depends on the type of medium that the light passes through, and is referenced by a number called the **Index of Refraction**. Carrara provides a number of preset refraction index settings for many different types of transparent or semi-transparent objects including liquids, glasses, and gemstones. You can also manually add other values as needed to reproduce materials not available in the drop-down list.

The image below shows perfume bottles before (left) and after (right) Index of Refraction is added to the bottles and liquid. Crystal is selected for the bottles and alcohol is selected for the perfume.

To assign an Index of Refraction component to a shader:

1. After you create your transparent or semi-transparent object, select the **Refraction** channel in the Shader Tree editor.

2. Choose **Index of Refraction** from the component menu. The options appear in the right pane of the Shader Tree Editor.
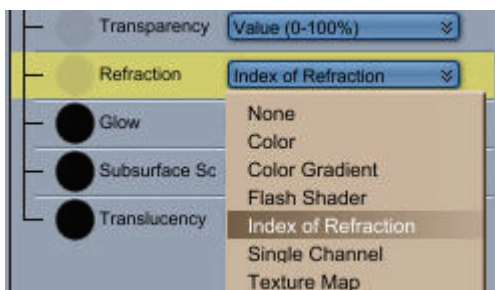
3. Choose one of the following options in the Shader Tree Editor:

   - **Material:** Select a material from the list. Choose the one that most closely approximates the material that the selected object is made of.

   - If the desired material does not appear on the list, choose (**Other**). This enables the Other Value field so that you can manually enter an Index of Refraction value for other materials. Chemistry textbooks (as well as other resources) sometimes provide index of refraction values for other materials.
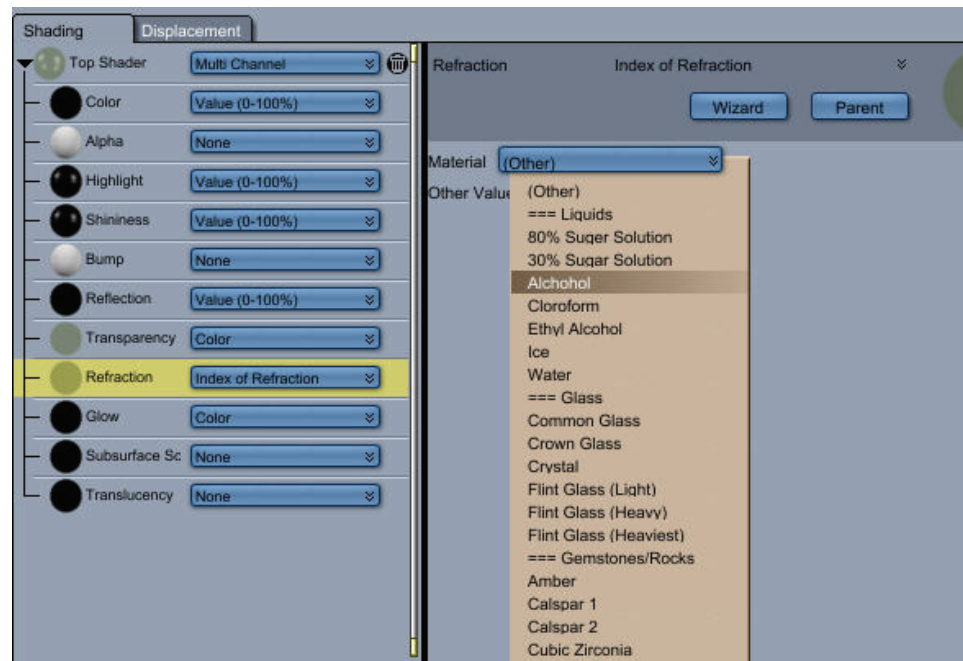
Single Channel

Use the Single Channel component when you want to assign a specific channel to a shader component. The menu allows you to assign a Color, Alpha, Highlight, Shininess, Bump, Reflection, Transparency, Refraction, or Glow channel.
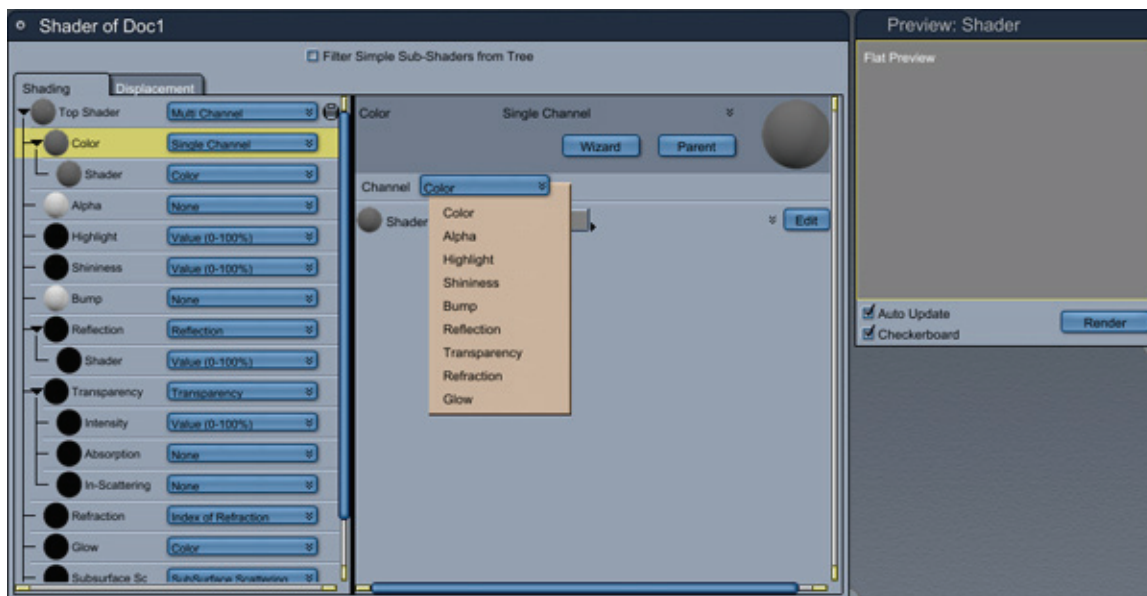
To assign a Single Channel component to a shader:

1. Select the shader channel that you want to configure.

2. Choose **Single Channel** from the component menu. The options appear in the right pane of the Shader Tree Editor.

3. Choose the following options in the Shader Tree Editor:

- **Channel**: From the popup menu, choose one of the channel options (Color, Alpha, Highlight, Shininess, Bump, Reflection, Transparency, Refraction, or Glow).

- **Shader**: To set the shader, click the **Edit** button in the Shader Tree editor to select from the Shader popup menu.

Texture Map

The **Texture Map** component allows you to use a 2D image, such as a scanned photograph or paint-type illustration, in your shader. You can also use animated formats such as AVI and MOV as texture maps.

Used effectively, texture maps can lend your shaders unparalleled realism. Many complex real-world surfaces are nearly impossible to simulate through other means.

You can use a 2D editor to create a texture map. When you place a Texture Map component on the shader tree, you must click the folder icon. Then the standard Open dialog displays, prompting you to choose an image.
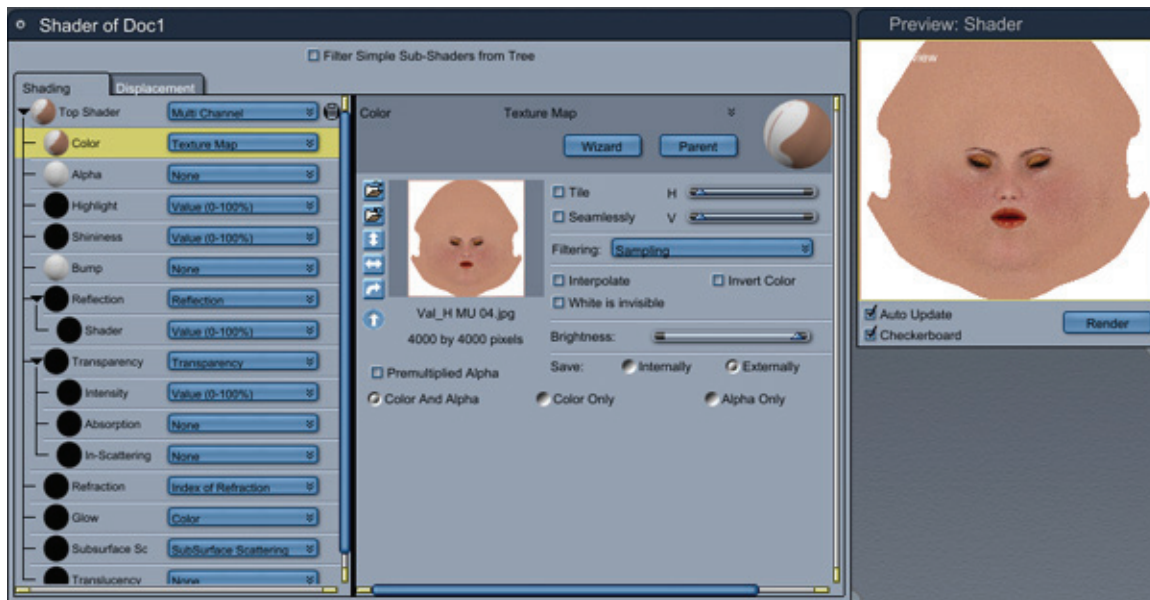
A discussion of the Shader Tree Editor's display options and instructions for navigating the shader tree appears in "Building and Editing Shaders."

To assign a Texture Map component to a shader:

1. Select the shader channel that you want to configure.

2. Choose **Texture Map** from the component menu. The options appear in the right pane of the Shader Tree Editor.

3. Use the controls in the upper-left area of the Shader Editor to select and orient your image:

- **Load**: Click this button to open a dialog that allows you to select a texture.

- **Reload**: Click this button to reload the texture map in its original condition.

- **Flip Vertical**: Flips the texture vertically on the object.

- **Flip Horizontal**: Flips the texture horizontally on the object.

- **Rotate**: Rotates the texture 90 degrees clockwise on the object.

- **Rotate Direction**: Indicates the current direction of rotation.
- **Texture Preview**: A thumbnail preview of the image appears on the left of the Texture map controls with the image's dimensions in pixels displayed below it.
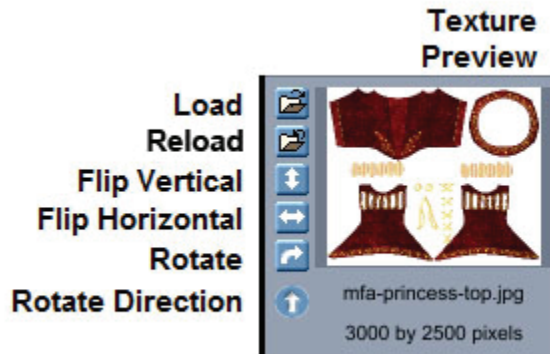
4. The tiling controls allow you to specify how you want to tile the image on your object:

- **Tile**: Check this option to tile your image.
- **Seamlessly**: If you want Carrara to rotate and flip neighboring tiles to maximize continuity, click the Seamlessly check box. If your image is specifically designed for seamless tiling, you don't need to enable this option.
- **H**: Adjust the slider to the number of times you want to tile the image horizontally.
- **V**: Adjust the slider to the number of times you want to tile the image vertically.
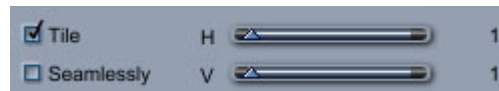
5. The Filtering menu determines how the texture is filtered during rendering:

- **Sampling**: Sampling is the fastest and lowest quality.
- **Summed Area Table**: Summed area table takes a lot of memory and is not recommended for big texture maps.
- **Gaussian Filtering**: Gaussian provides the best quality.

6. Additional options determine how certain colors are represented or rendered in the image:

- **Interpolate**: Select the Interpolate check box, to specify that the texture map be interpolated.
- **White is Invisible**: The White is invisible check box allows you to achieve a type of masking effect by instructing Carrara to ignore any purely white pixels in the image. For example, you could use this option to place a logo with an irregular shape on the surface of an object. Simply create a map with a white background and place it in the Color channel, then apply it to an object using the Polygon tool. Wherever there is a white pixel in the map, the layer or surface below shows through.

- **Invert Color**: Check this option to invert the colors in the image. (For example, in a grayscale image, black inverts to white, and white inverts to black).

- **Brightness**: Drag the slider to the right to increase the brightness.
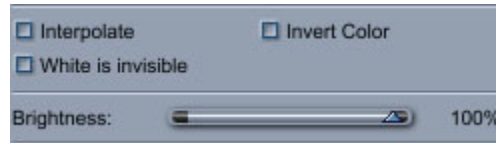
7. By default, Carrara saves copies of all texture maps internally. You can also have Carrara save only a reference to an external file. Use the options in the Save area to determine how images are saved with the scene:

- **Internally**: In general, you can store maps internally unless your file contains particularly large texture maps (or many smaller maps). Storing maps internally avoids organizational hassles, because you don't need to keep track of any external files. However, internally saved maps greatly increase the size of a file, which can result in slower loading and saving.

- **Externally**: With an external map, you can modify it with another program, and you don't need to reload the map. If you move an externally referenced image on your hard drive, Carrara prompts you to locate the image file the next time you open your scene file or a shader Browser document.
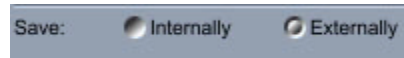
8. The last options control how the alpha channel is used in your texture map:

- **Premultiplied Alpha**: TBD

- **Color and Alpha**: Choose this option to use both the color and alpha channels in your shader.

- **Color Only**: Choose this option to use only color information when an image also contains an alpha channel.

- **Alpha Only**: Choose this option to use only the alpha information from your selected image.
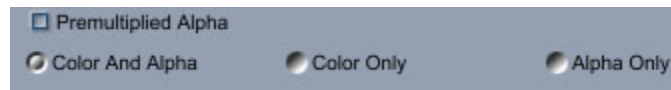
Using Color Images as Texture Maps

Texture maps using color images are extremely useful in the Color channel. You might import an actual product logo as a texture map and apply it like a decal to a 3D package model. Or to mimic an extremely detailed natural surface, you could import a small photographic sample of the surface and tile it:duplicate it a specified number of times:to cover your entire object.
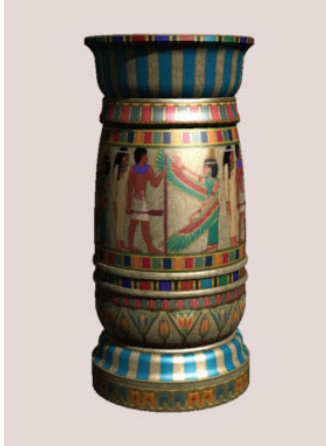
Color texture maps are also appropriate in the Highlight, Reflection, Transparency and Glow channels.

If you use a color image in a non-color channel, Carrara internally converts it to grayscale.

You can use an image of any color depth as a texture map. Note, however, that Carrara cannot use CMYK images as texture maps. If you want to use a CMYK image, you must convert it to RGB before importing it.

Black & White and Grayscale Images as Texture Maps

You can use a black and white or grayscale texture map in any channel (besides Color) to specify varying levels of a particular shading attribute. The shade of each pixel in the image determines the level of the attribute for the corresponding point on the object or Layer.

If you use a black and white image, each black pixel turns the attribute on (sets it to 100%); each white pixel turns the attribute off. An 8-bit grayscale image allows subtler effects, with 256 possible shades for each pixel. For example, a grayscale blend from white to black in the Transparency channel would make an object or layer fade smoothly from opaque to transparent.

You can also use a black and white or grayscale texture map with the Mixer operator, in place of a function. For more information, refer to "Layer Components."

Value

There are two **Value** components. One allows you to set a value between 0 and 100%, and the other allows you to set the value from 1 to 10000%. You'll use the Value component frequently in every channel but the Color channel. The control appears as a slider in the right portion of the Shader Tree Editor.

To assign a Value component to a shader:

1. Select the shader channel that you want to configure.

2. Choose **Value (0-100%)** or **Value (1-10000%)** from the component menu. A slider appears in the right pane of the Shader Tree Editor.
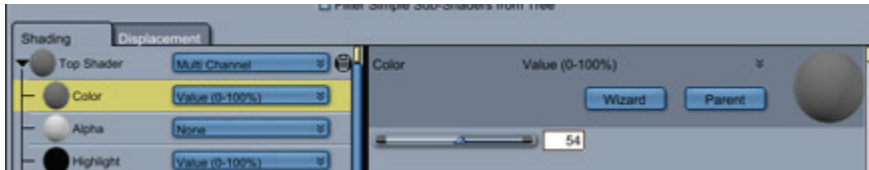
3. Drag the Value slider or enter a value. The number to the right of the slider displays the current value.

    *   When used by itself, a Value component specifies a constant level for a particular attribute across the surface of an object or Layer. For example, a 30% value in the Transparency channel means the shader will make an object 30% transparent.

    *   When used beneath a Color channel, Carrara converts its value to a shade of gray. 0 converts to black, 100 to white.

    *   You can use a Value component to set the Opacity Mask of a layer in a Layers List shader.

Operators

Operators allow you to create complex shading effects by combining two or more components or sub-shaders within a single shading channel.

Add (+)

The Add operator sums the values of two components. When you place the Add operator on the shader tree, it appears as a node with two branches extending below.
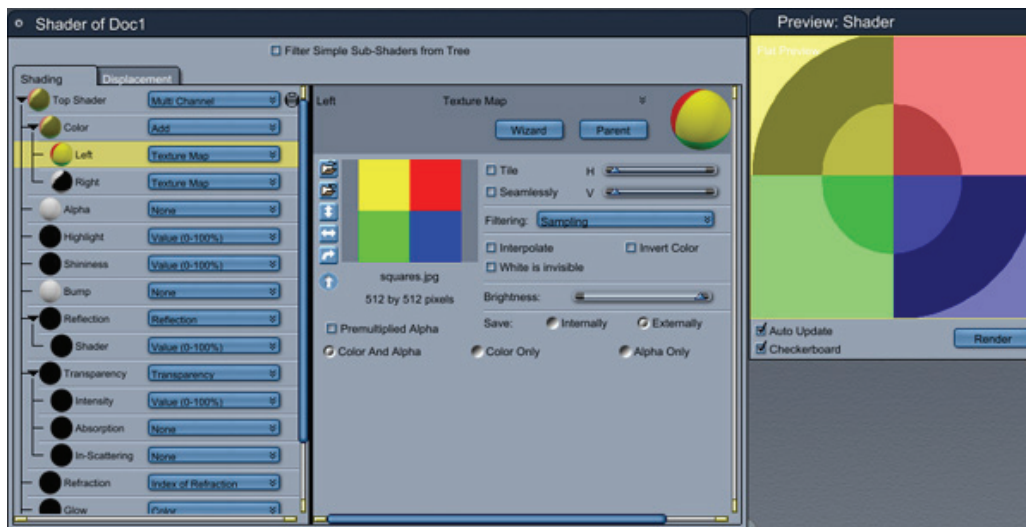
The two branches are placeholders for the components you want to combine. For each point on a layer or object, the Add operator sums the values of the two components.

Curve Filter

The Curve Filter shader modifies the levels of a black and white image.
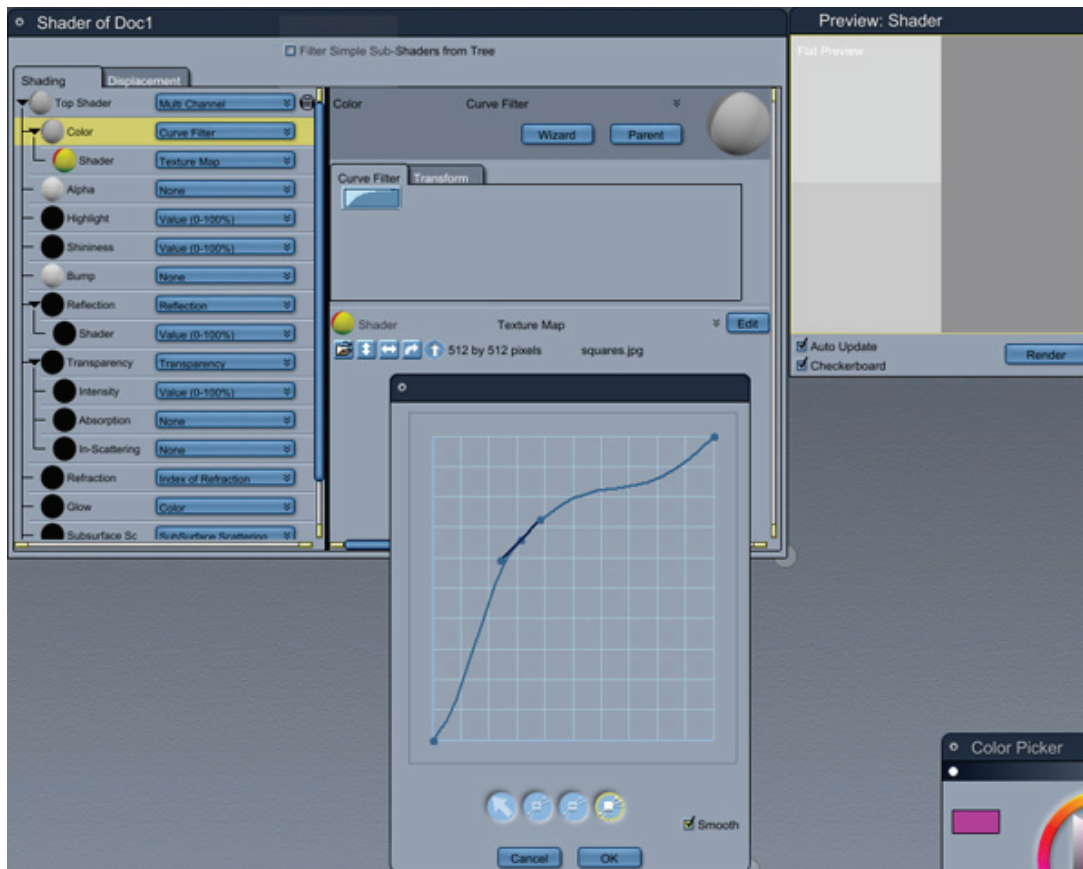
Two parameters can be set:

- the first parameter is the shader to which the filter will be applied. It can be selected by the Shader popup menu.

- the second parameter defines the curve. The default curve doesn't modify the image. To edit the curve, click on the curve button to open the curve editor.

Four tools are available to edit the curve:

- **Move tool**: Modifies the point's location by clicking and dragging it.

- **Add Point tool:** Adds a new point to the curve when it is clicked on.

- **Delete Point tool:** Removes a point from the curve.

- **Convert Tangent tool**: Links or unlinks two tangents of a point (when the curve is a smooth curve)

The curve can be smoothed by checking the Smooth option (note that a smooth curve will be slower to render than a linear one).

Mixer

The Mixer operator is one of Carrara's most versatile shading tools. It can produce a wide variety of results, depending on which function you choose to mix the two source components.

When you place a Mixer operator on the shader tree, it appears as a node with three branches below it. The Source 1 and Source 2 branches are placeholders for the two components you want to mix. The mixing function resides on the Blender branch.
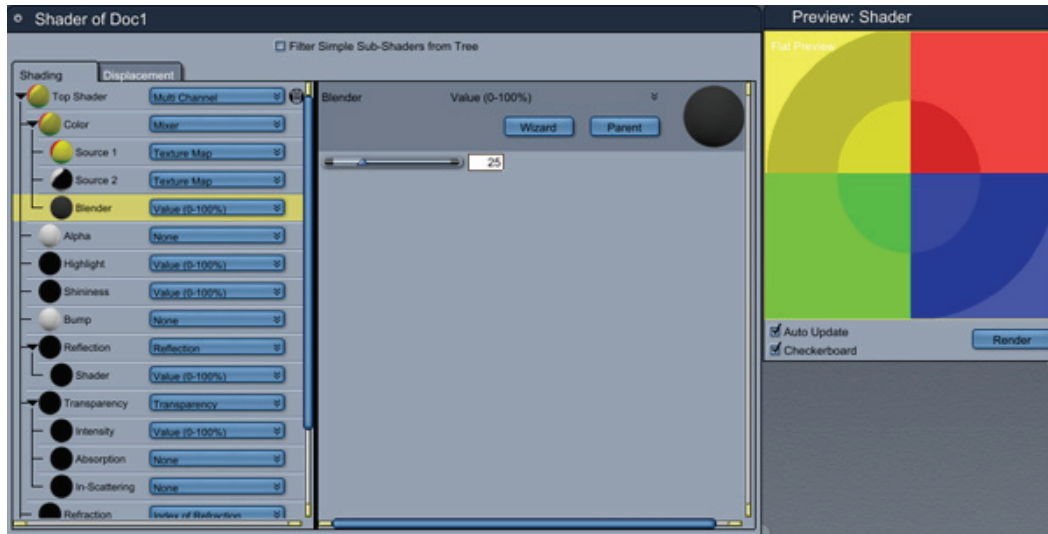
For each point on an object, the Blender function generates a value between 0 and 100. The Mixer operator uses this value to combine the component's sources. When the value is closer to 0, more of the Source 1 component is used. When the value is closer to 100, more of the Source 2 component is used. You can also use a shader in the Blender branch.

Some functions, like Checkers and Wires, generate a value of 0 or 100 for each point. These functions result in clear divisions between the two components. Other functions, like Wood and Spots, generate a range of values between 0 and 100. These functions result in gradual blends between the components.

TIP! The mixer function is very useful when you want to create a shader with two colors from a black and white shader. For instance, if you want a red and green checkerboard, you would create a mixer with the two colors (red and green) and select the checker board function as the blender.

Multiply (*)

The Multiply operator multiplies the values of the source components.
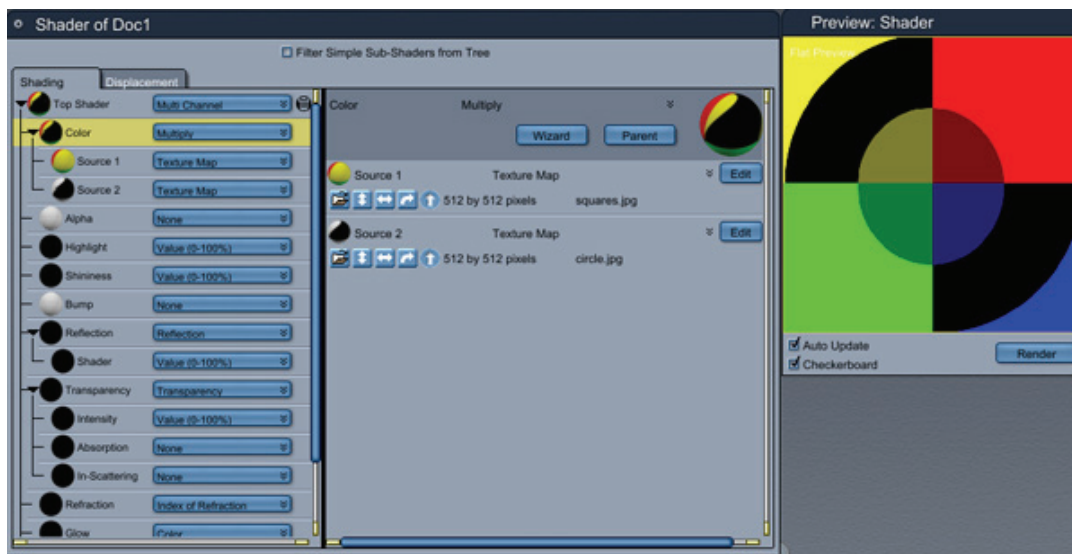
You can use the Multiply Operator to tint a grayscale bitmap by multiplying the bitmap shader by a color shader.

Overlay

The Overlay operator multiplies the values of its two sources but keeps the general look of the first one predominant.

This operator is not a symmetrical operator, so the order of the two sources is important. A 'Swap Shaders' checkbox is available to invert their order easily.

This operator can be used to bring noise to another shader.
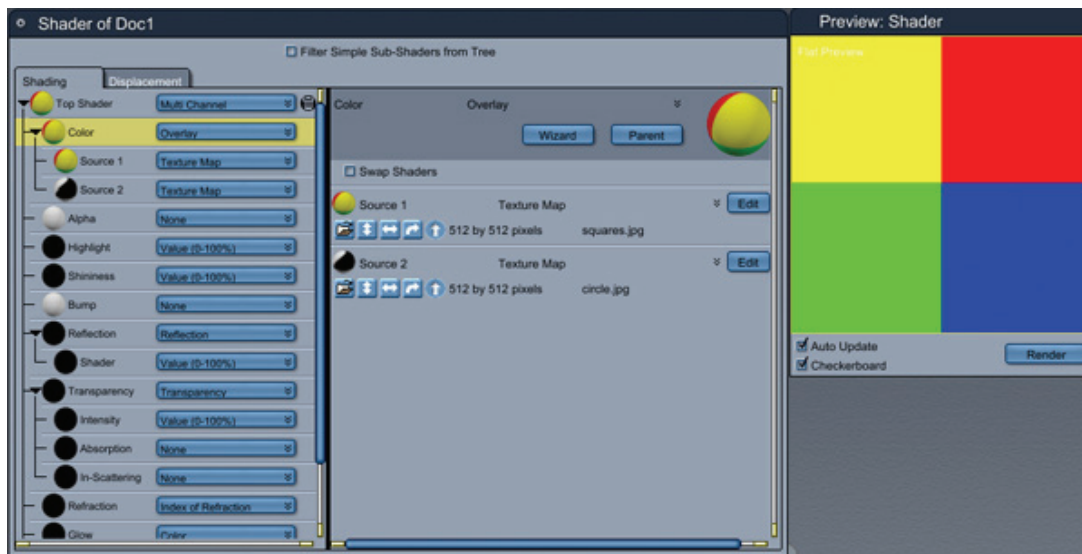
Subtract (-)

The Subtract operator subtracts the value of the Source 2 component from the value of the Source 1 component.
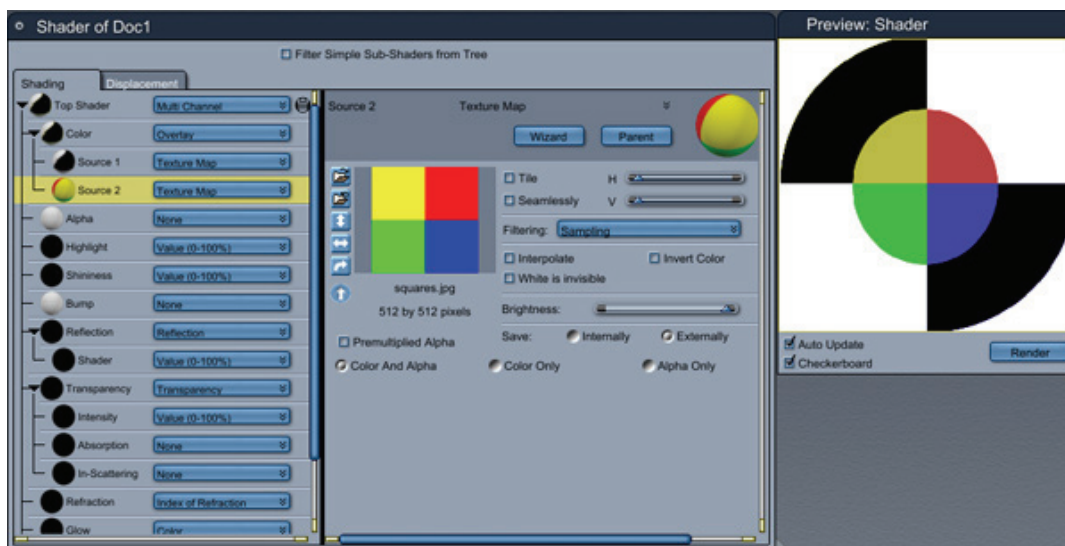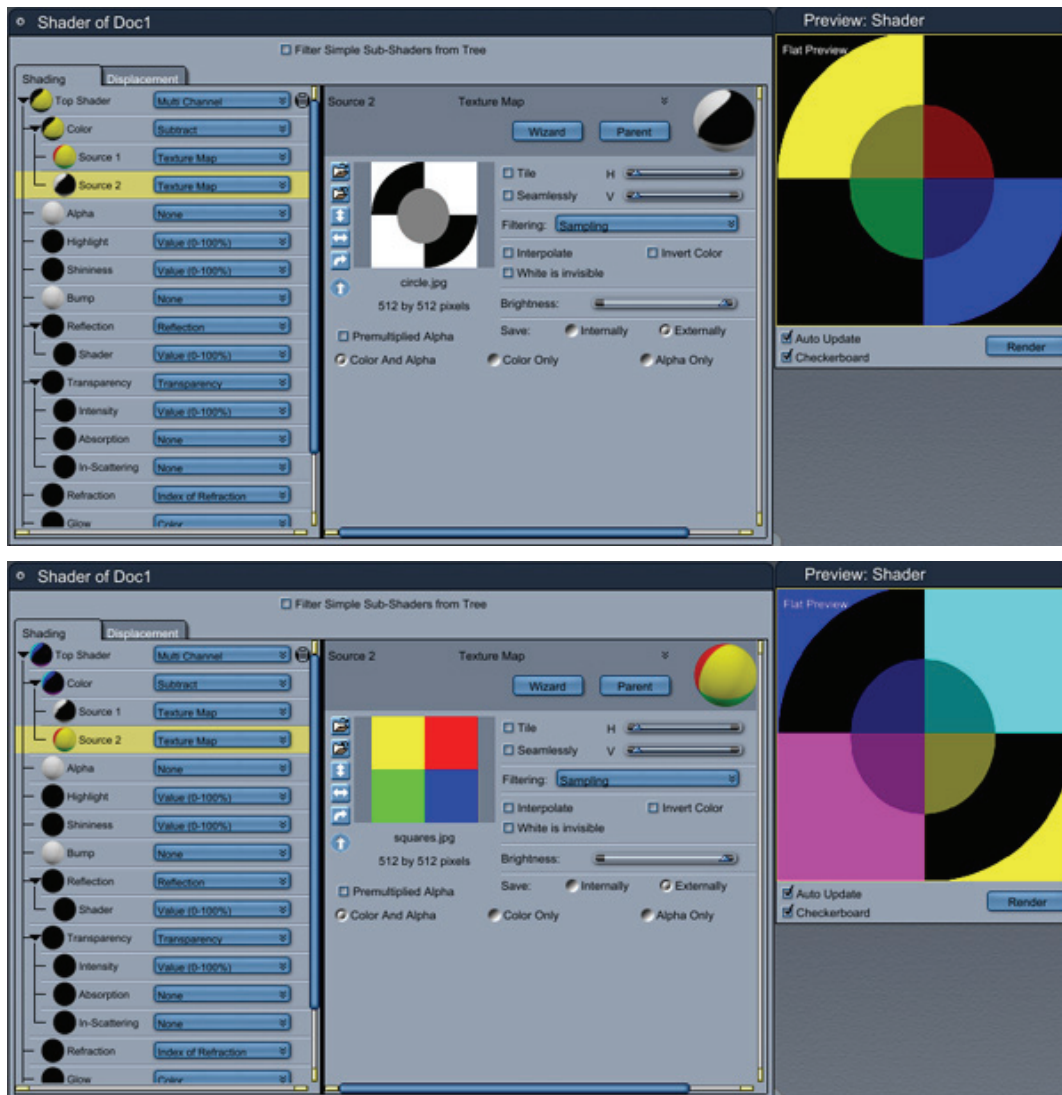
You can use the Subtract operator to invert the value of another component. Just place the component you want to invert in the Source 2 branch and a Value component set to 100 in the Source 1 branch. Try this technique on a complex shader if you want to invert its colors.

Functions

In general, you'll use Functions almost exclusively with the Mixer operator. You can, however, use them by themselves. A function generates a value between 0 and 100 for each point on an object or layer.

When you place a function on the Mixer operator, the operator uses the values generated by the function to mix the components on the Source 1 and Source 2 branches. Where the value equals 0, Carrara uses the Source 1 component; where the value equals 100, Carrara uses the Source 2 component. Intermediate values produce a blending of the two components.

When you place a function in a channel by itself, it assigns a value directly to each point on the object or Layer. In a non-color channel, Carrara uses each value "as is." In any of the channels designed for color input (Color, Highlight, Reflection, Transparency or Glow), Carrara converts each value to a shade of gray, with 0 translating to white and 100 to black.

Pattern Functions

Bricks

The Bricks function produces a pattern like a wall of bricks. It assigns a value of 100 to the bricks and a value of 0 to the mortar. Bricks is a 2D function.
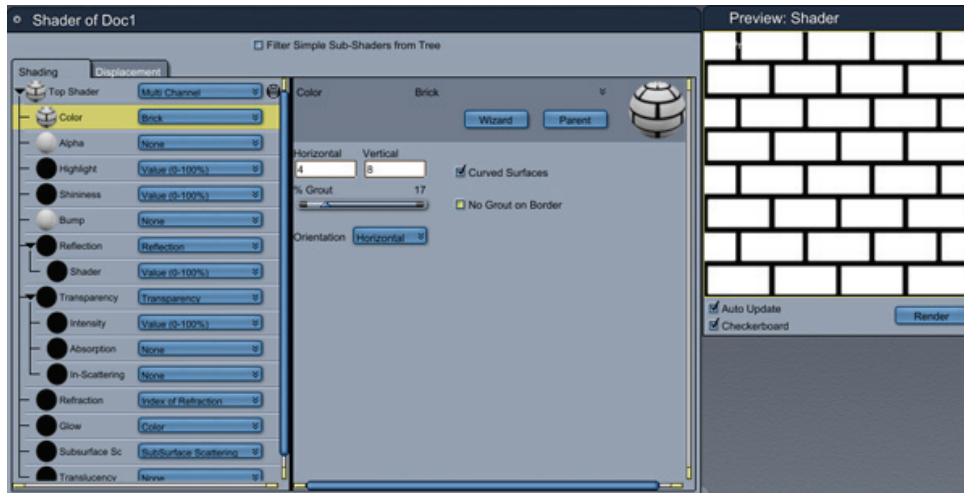
To create bricks:

1. Use the **Horizontal** and **Vertical** control the number of bricks.

2. **Grout** controls the percentage of each row that is grout.

3. **No Grout on Border** leaves the grout off of the edges.

4. **Curved Surfaces** smoothes the transition between brick and mortar.

5. **Orientation** changes the direction of the brick pattern.

Caution

The Caution function produces a pattern of diagonal stripes. It alternates a value of 100 and a value of 0 to the bands. Caution is a 2D function.
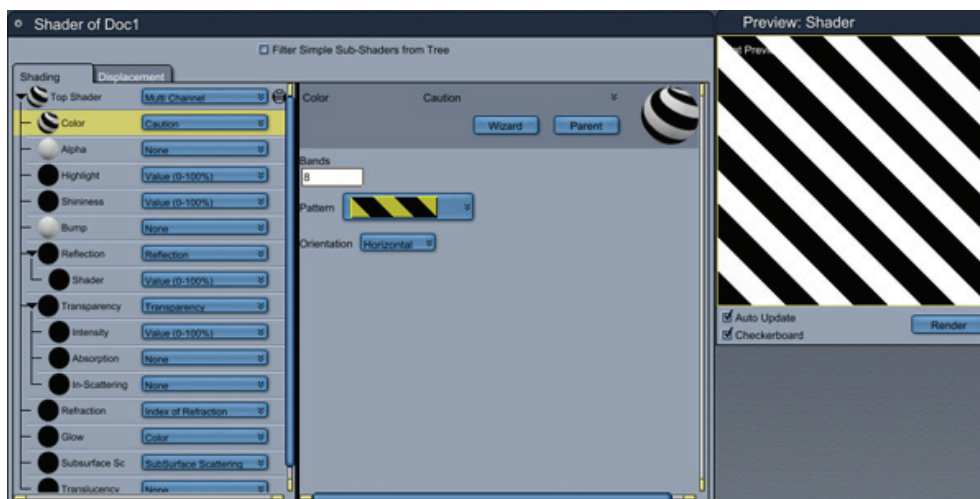
To create a caution band:

1. Bands control the number of stripes.

2. Pattern controls the pattern of the stripes.

3. **Orientation** changes the direction of the pattern.

Checkers (and Stripes)

The Checkers function produces a checkerboard or striped pattern. Every point is assigned a value of exactly 0 or 100, so no blending occurs. Checkers is a 2D function.
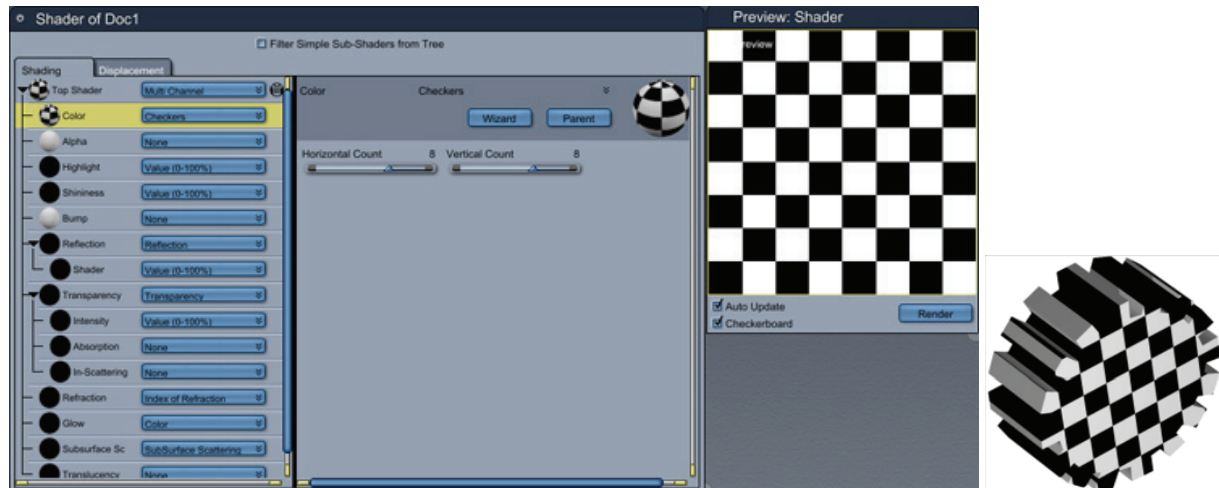
To set Checkers options:

1. Drag the Squares H slider to set the number of horizontal squares.

2. Drag the Squares V slider to set the number of vertical squares.

By setting one of the sliders to 0, you get stripes:either horizontal or vertical, depending on which slider is at 0.

You can use Checkers to create tile effects, checkerboards, striped objects such as wall papers, beach balls, or umbrellas.

To create stripes:

• Drag one of the sliders to 0.

  The stripes will be either horizontal or vertical, depending on which slider is set to 0.

Formula

The Formula function lets you design a mixing function with a mathematical formula.

For more information, refer to "Using Formulas in Carrara."
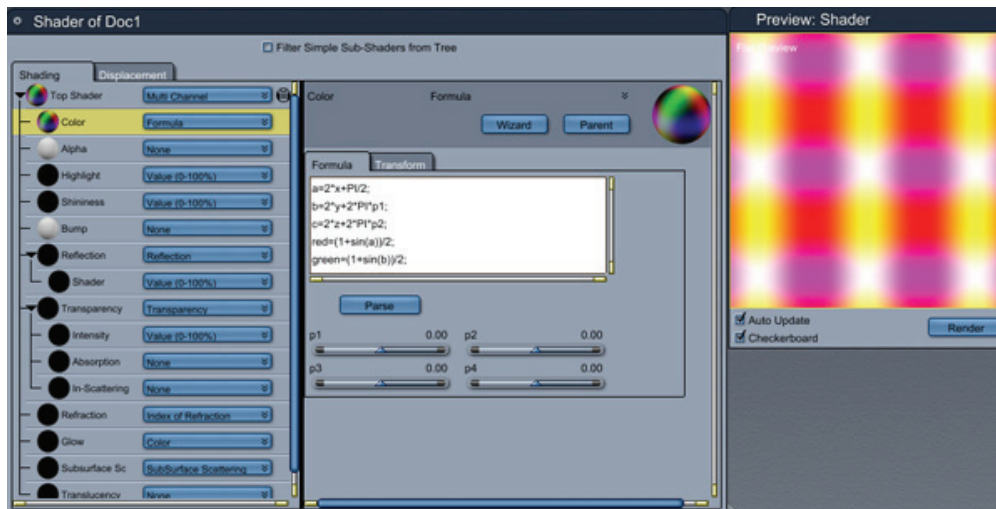
Gradient

The Gradient function produces a gradual blend from one color to another. Gradient is useful in the Color and Transparency channels. Gradient is a 2D function.
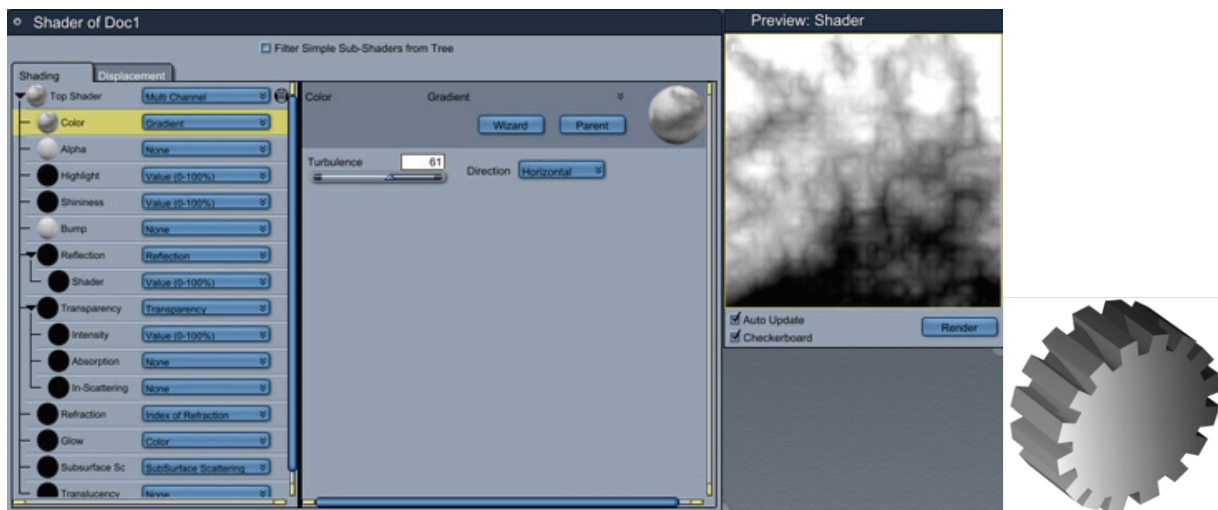
To set Gradient options:

1. Choose **Direction > Horizontal** or **Vertical** to set the direction of the blend.

2. Adjust the Turbulence slider to mix up the colors as they change.

    A low setting results in a uniform blend. A high setting increases irregularity in the blend.

Julia Set

The Julia Set pattern
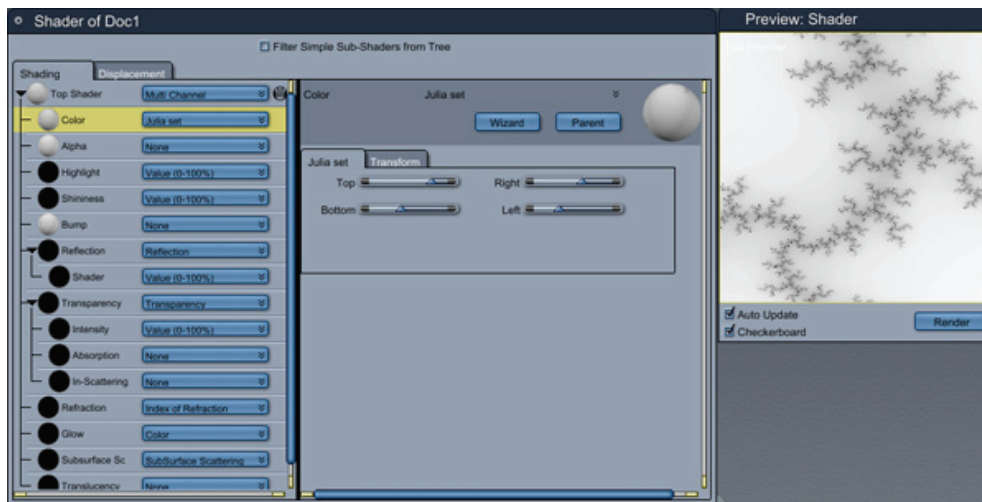
Mandelbrot Set

The Mandelbrot set pattern.
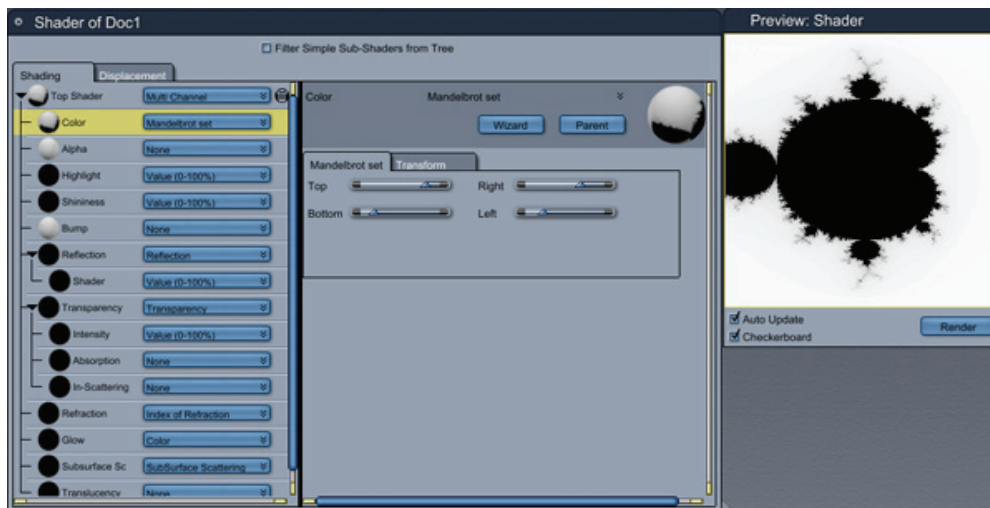
Plank Shader

This shader was designed primarily as a supplement to the wood shader. It has many of the same parameters. Here is the suggested procedure for using this extension:
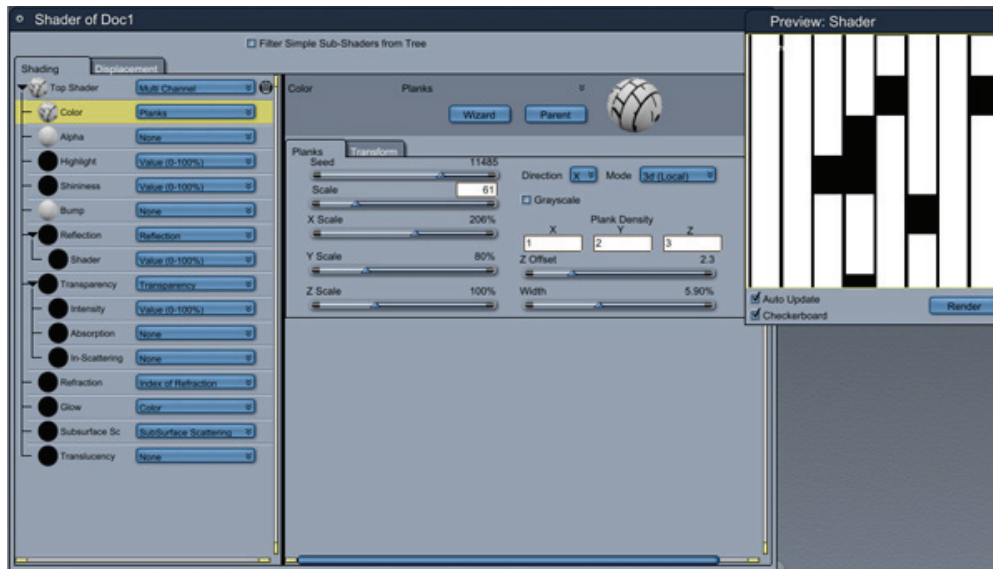
To use the plank shader:

1. Use Lumber Yard first to get your wood looking just how want it to.

2. Move your mixer from the color channel to the highlight channel. Change the color channel to a new mixer.

3. Move wood to the second slot of your new mixer.

4. Press Control and drag the lumber yard shader from its spot in the old mixer to the mix spot in the new mixer.

5. Change the lumber yard in the new mixer to 'planks'.

6. This will set all the parameters in the plank shader identical to your wood.

7. Copy into the bump channel, if desired.

Options

- **Direction**: Z, the default, provides a view as if one cut a tree down and attempted to count the rings. X and Y provide a view as if one cut a tree into planks along each respective axis.

- **Mapping**: Choose between three - 3D Local, 3D Global, 2D UV. 3D local is the default value. The difference between global and local is fairly simple; each object has its own local coordinate system, choosing that option will cause the shader to look identical even when animated. If global mode is selected, the appearance will differ based on the object's global location. UV mapping uses UV coordinates, this can often result in distortion; nevertheless, it is still useful, especially when used on an object you plan to morph.

- **Seed**: This value "seeds" the random number generator.

- **Scaling**: Scales the shader.

- **Order and Persistence**: Please refer to the introduction to noise introduction in the section covering Noise Factory. See "Noise."

- **Perturbation**: This parameter sets the amplitude of the turbulence. Higher values lead to a more knotty looking surface.

- **Grain Edge**: The default value for this parameter, 1, leaves a definite edge between each band of wood. Decreasing this value makes for successively more even blending of bands.

- **Threshold Controls**: These controls behave identically to the threshold controls in Noise Factory. See "Noise Factory."

- **Plank Controls**: Controls the density of planks (if desired). The default values of zero provide no paneling. Increasing the density increases the number of bands in each plank. To increase the amount of planks visible, scale the texture accordingly.

- **Z Offset**: This parameter creates random variation in the Z length of any given plank. The higher the value, the greater the average offset.

- **Value Variation**: This parameter controls the maximum amount by which the intensity varies from plank to plank.

Psychedelic

The Psychedelic shader function creates unusual textures, including swirls of color reminiscent of the psychedelic Pop Art of the 1960s. When animated, the Psychedelic function can create hypnotic or trippy effects. Psychedelic is a 3D function.

This shader should be used with a mixer if you want to give color to your shader (see "Add (+)").
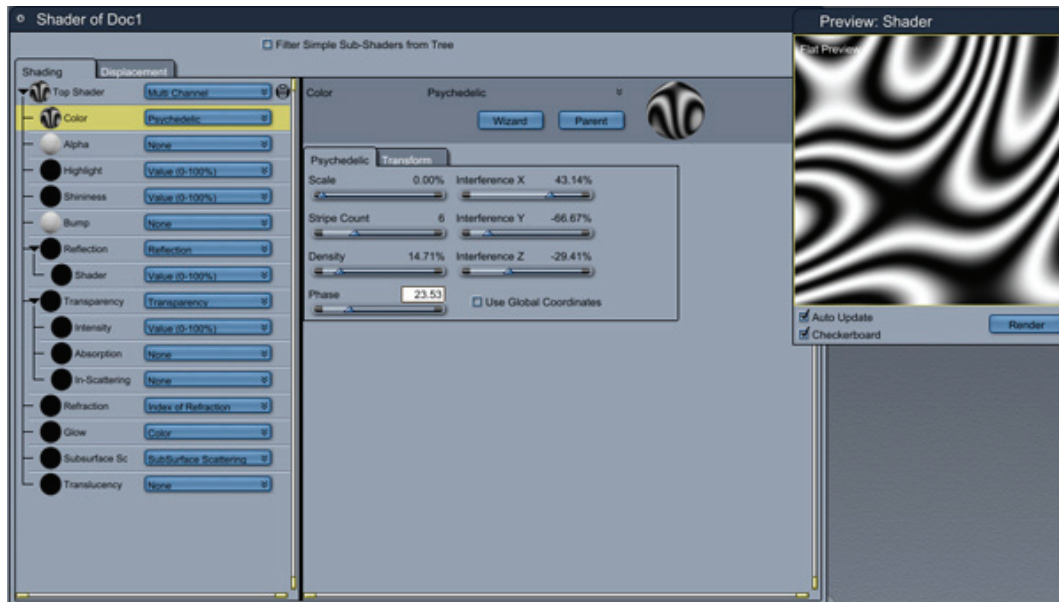
To set Psychedelic options:

1. Adjust the three Interference sliders to control the amount of interference applied to each plane. The sliders range from -1.00 to 1.00. Each slider controls the stripes applied on the specific plane. A setting of 0 results in no interference on that plane. Using all three planes, you can create more interesting design elements.

2. Enable the Use Global Coordinates check box to use the Global Coordinates for the Psychedelic function.

   - By default, the psychedelic function uses the individual Object's coordinate system. The shading is constant as you move and rotate the object.

   - When Use Global Coordinates is enabled, the shading remains static in global space when the object moves. The object behaves as a "window" on the shading. This is like moving your hand through the dappled shadows under a tree. Instead of shadows, though, the psychedelic shading passes across the moving object.

3. Drag the Scaling slider to control the size of the design elements.

4. Adjust the Number of Stripes slider to control the number of stripes in each element.

5. Adjust the Density slider to control the density of the pattern.

6. Adjust the Phase slider to set the current position in the psychedelic cycle.

   - The Phase slider ranges from 0.00 to 1.00. The two extreme settings display the cycle at the same point.

   - Use the flat preview to see how the shader changes when you adjust the Phase.

NOTE! Phase settings are particularly useful in animations. Set key events to animate the Phase from 0 to one using the Oscillate: Saw/Loop tweener.

Scanlines

The Scanlines function produces scan lines.
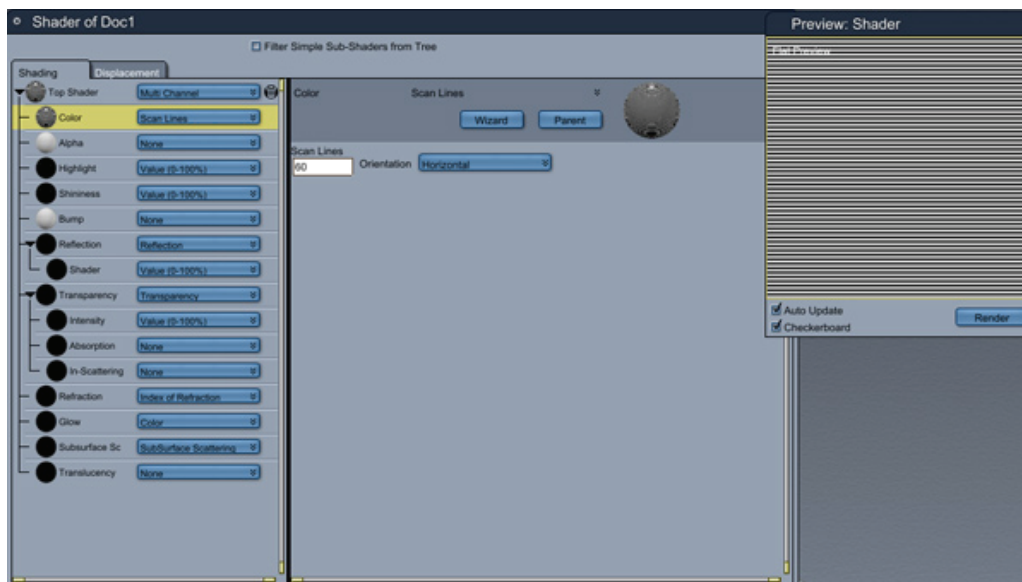
To use scanlines:

1. **Scan Lines** controls the number of lines.

2. **Orientation** controls the direction of the lines.

Spherical Gradient

This shader is most useful when designing explosions using particles. When applying this shader to a particle system, be sure to check the 'Global' option. In addition to setting the global option, be sure to set the displacement values identical to the center of your particle system.
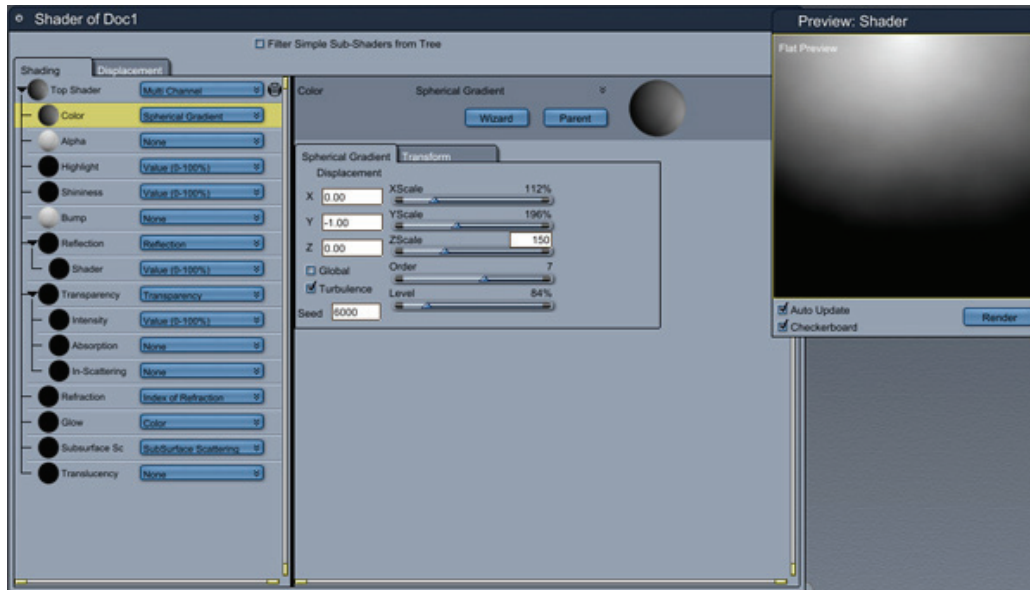
Options

- **Seed** : This value "seeds" the random number generator.

- **Scaling**: Experiments with different combinations of x, y, and z scales to observe the various effects.

- **Displacement**: These values set the center of the sphere.

- **Order and Persistence**: Please refer to the introduction to noise present in the section covering Noise Factory. See "Noise Factory."

- **Global**: When this box is checked, the shader will use global coordinates, meaning the appearance is dependent on the location of object.

- **Turbulence**: This box enables turbulence.

- **Level**: This parameter sets the amplitude of the turbulence.

Squares

The Squares function produces a pattern of random grayscale squares. Squares is a 2D function.
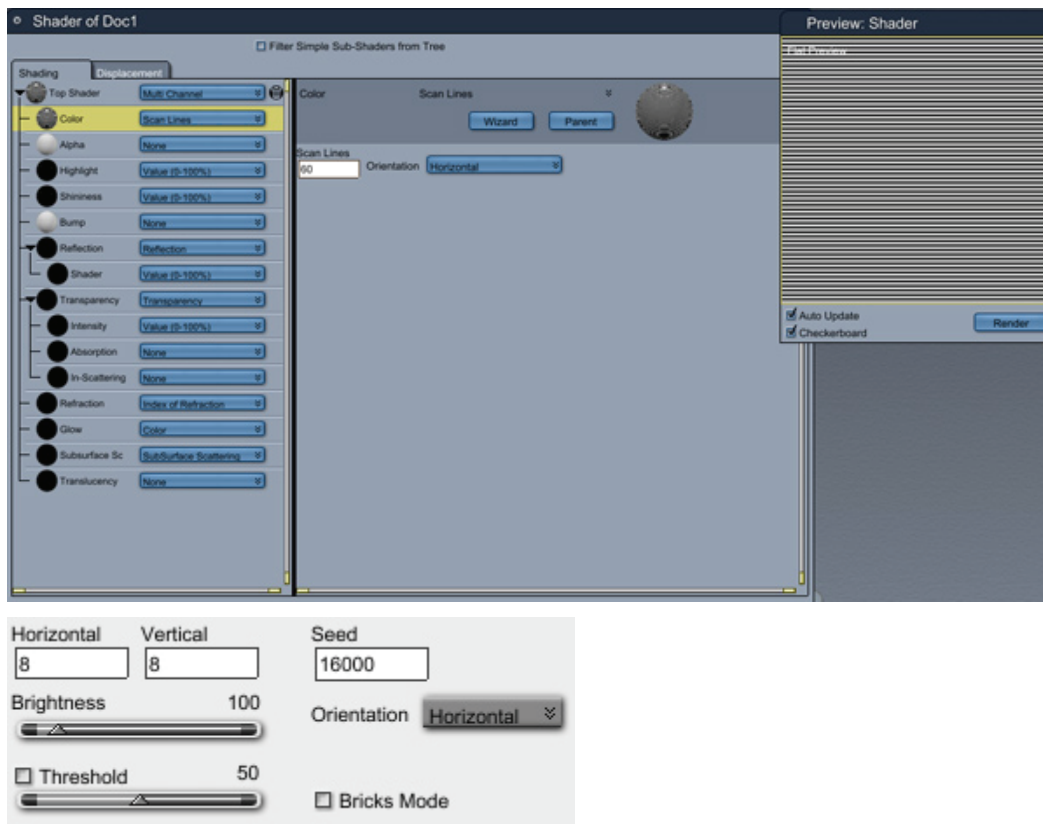
To use squares:

- **Horizontal** and **Vertical** control the number of squares.

- **Brightness** controls how light or dark the squares are.

- **Threshold** makes the square black or white depending on if the grayscale value is greater or lower than the threshold value.

- The **Seed** value provides the starting point for the grayscale values.

- **Bricks Mode** changes the pattern of the squares to be offset like bricks.

Tips and Tricks

- Random mixtures of different square settings make great spaceship hulls. Put a squares in the Mix operator with **Threshold** on and in the right and left channels put Squares with different **Horizontal** and **Vertical** settings.

- Use the Multiply function with the Tile shader in the Glow channel to make lit portholes or windows at night.

Tile

The Tile function produces a pattern like a floor of tiles. It assigns a value of 100 to the tile and a value of 0 to the grout. Tile is a 2D function.
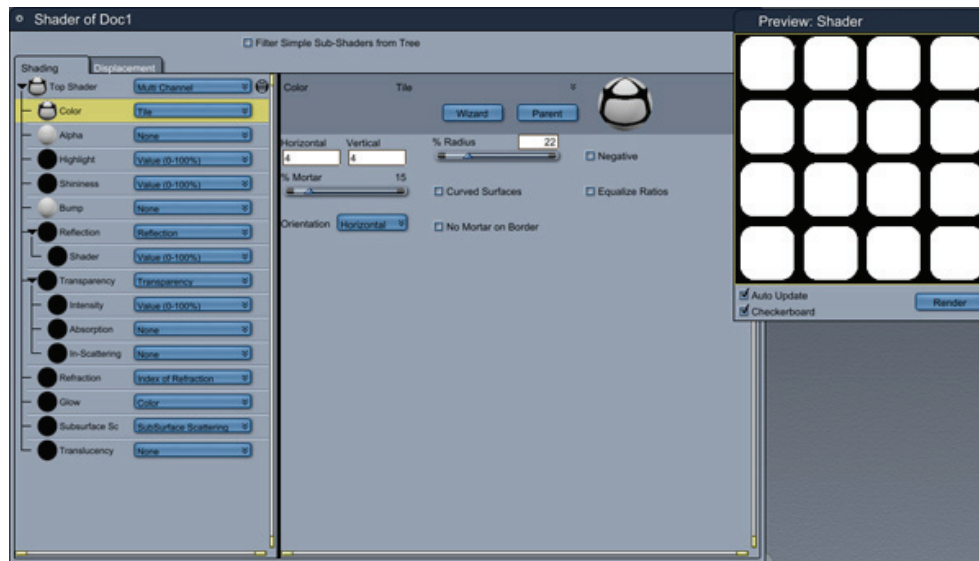
To use Tile:

- **Horizontal** and **Vertical** control the number of tiles.

- **Mortar** controls the percentage of each row that is grout.

- Radius controls amount of curve on the corners and edges of tiles.

- **No Mortar on Border** leaves the grout off of the edges.

- **Curved Surfaces** smoothes the transition between tile and grout.

- Try **Equalize Ratios** if your grout lines don't render the same size horizontally and vertically.

- Negative inverts the output.

- **Orientation** changes the direction of the tile pattern.

Tips and Tricks

- Use Tile with a large Grout setting in the transparency channel to add air vents to machines.

- No tile is perfect, so combine with Noise in the Bump channel to create more believable tiles.

Wires

The Wires function produces a grid of lines. As with the Checkers function, Wires assigns a value of either 0 or 100 to each point on an object or Layer. Specifically, it assigns a value of 100 to the lines of the grid and a value of 0 to the spaces. Wires is a 2D function.
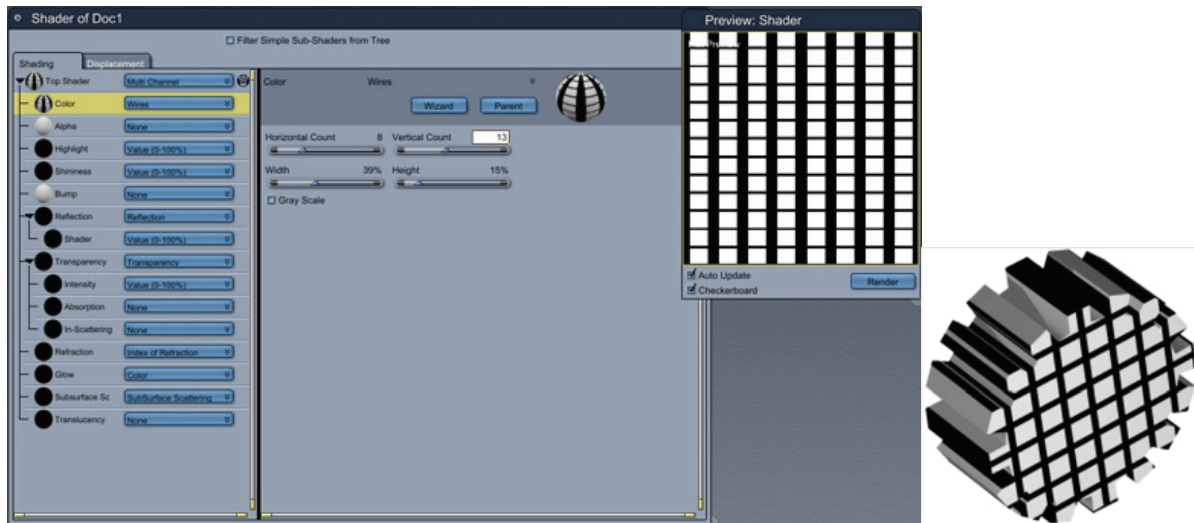
To set Wires options:

1. Drag the Horizontal count slider to set the number of lateral wires.

2. Drag the Height slider to set the size of the lateral wires. The thickness of the horizontal/vertical wires is expressed as a percentage of the height/width of the entire object or Layer. You can determine the thickness of a single wire by dividing the percentage by the number of wires. For example, if ten horizontal wires represent 50% of a Layer's height, the thickness of each wire is 5% of the height.

3. Drag the Vertical count slider to set the number of longitudinal wires.

4. Drag the Width slider to set the size of the longitudinal wires.

5. Enable the Gray Scale option if you want to smooth the transitions between the spaces and the wires.

    - The grayscale option smoothes the transitions between wires and spaces.

    - This option is especially useful in the Bump channel, where gradual transitions produce striking results. You can create textures for objects to create, for example, a pumpkin or corrugated sheet metal. You can also use it along with the Checkers function to create grouted tile.

Natural Functions

Cellular

The Cellular function creates a surface that looks like a network of cells. Cellular is a 2D function. You can use it effectively in the Color and Bump channels. You can experiment with it in the other channels for other effects. Use Cellular to create organic surfaces such as alien skins, leather, or exotic plant textures.
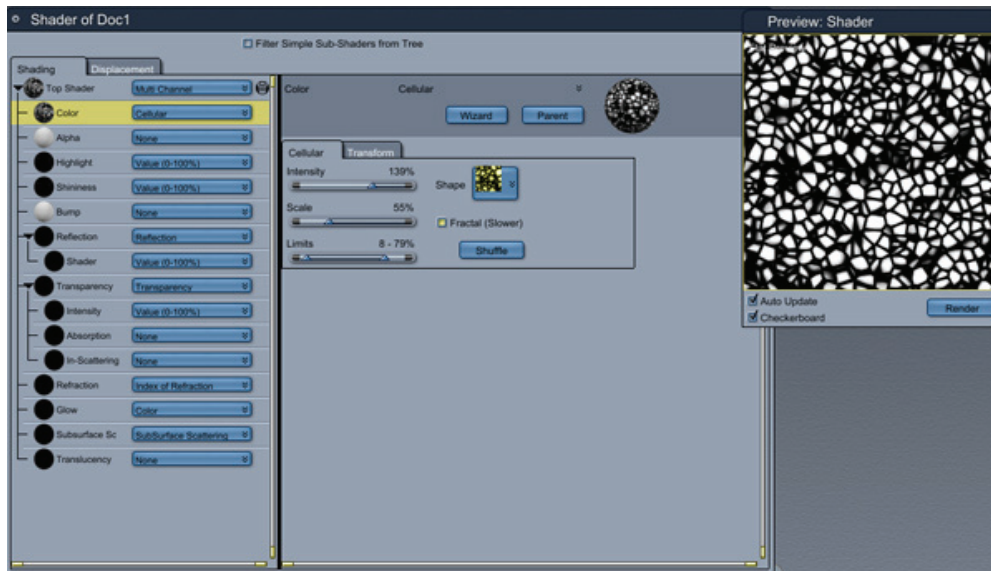
To set Cellular options:

1. Use the Shape menu to select the basic shape of the cells.

2. Drag the Intensity slider to adjust the contrast between the two colors. The slider ranges from 0% to 200%. Negative settings invert the image.

3. Drag the Scale slider to change the size of the cell shape. The slider ranges from 1% to 200%.

4. Drag the markers on the Upper and Lower Limits slider to controls the range of values created by the function. The range determines the balance between the two (light and dark) components. The left marker controls the lower limit. The right marker controls the upper limit.

5. Enable the Fractal Version to use fractals instead of more regular shapes for the cells.

   - Using the Fractal Version can significantly increase the time required to redraw and to render your image.

Fire

The Fire shader simulates a flame. You can adjust the effect from a steady candle flame to a turbulent wood fire. You can also animate this shader.
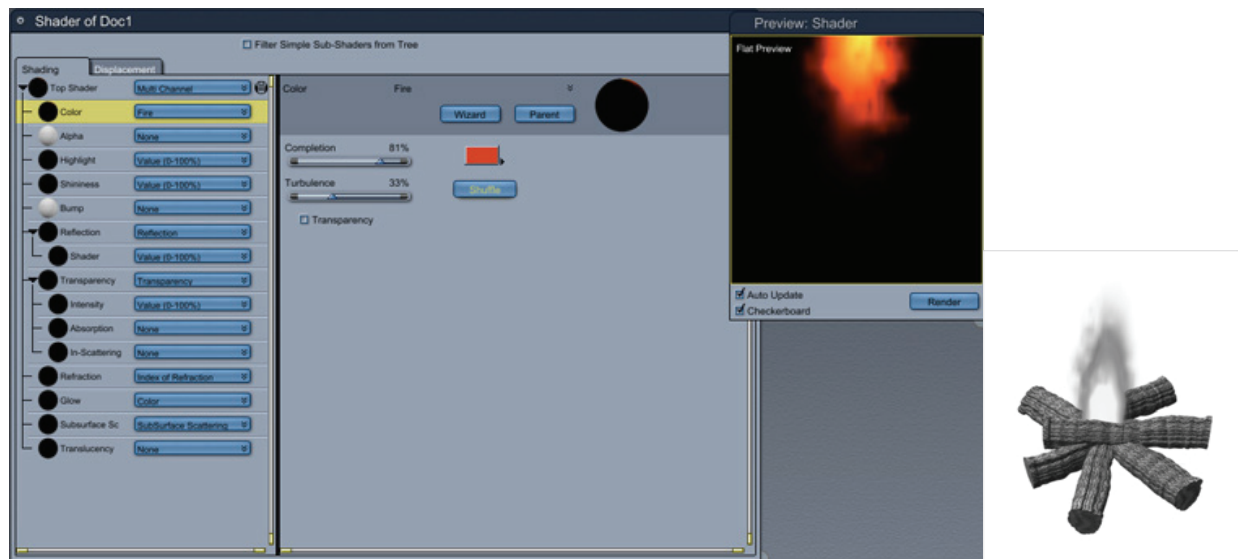
TIP! To create a good fire shader, you should put the same fire component in the color and glow channels.

To set Fire options:

1. Set the Color parameter to control the color of the fire. Typically, you'll choose an orange color.

2. Set the Turbulence parameter to change the turbulence of the fire. At 0%, it looks like a candle. At higher values, it flickers like a flame in the wind.

3. Set the Completion slider to define when the flame starts to move and when it stops. Typically, set this value to 0% for the first frame of your animation and to 100% for the last frame. If you use the Fire shader in multiple channels, the Completion value should be the same in every channel.

4. Use the Shuffle button to randomize the turbulence.

5. Use the Transparency button when this shader is placed in the Transparency channel.

Lumber Yard

Lumber Yard can be used to create amazingly realistic 3D procedural wood. It has a clear advantage over texture maps. 3D objects will appear to actually be cut out of a block of wood.
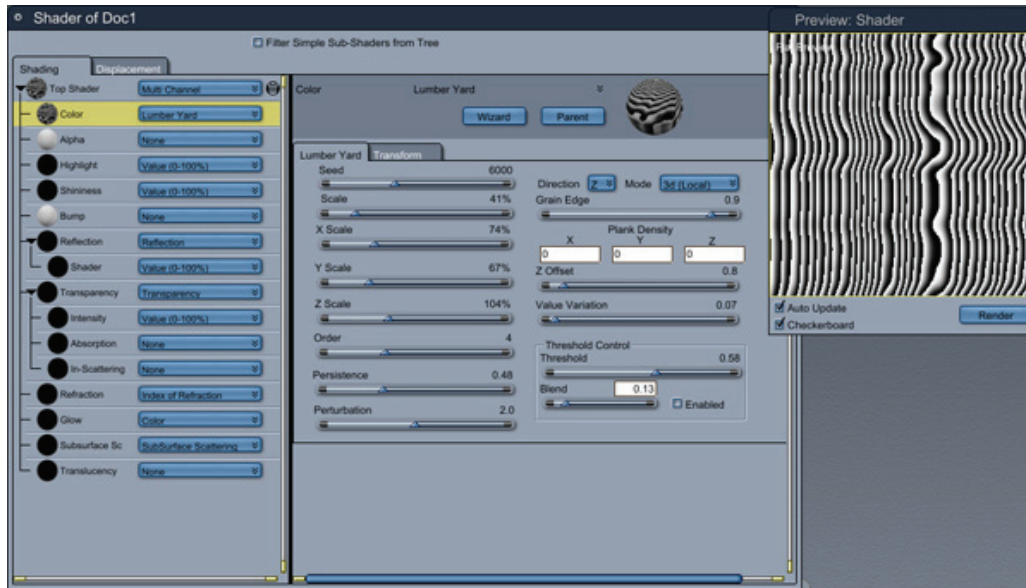
To set Lumber Yard options:

- **Direction**: Z, the default, provides a view as if one cut a tree down and attempted to count the rings. X and Y provide a view as if one cut a tree into planks along each respective axis.

- **Mapping**: Choose between three - 3D Local, 3D Global, 2D UV. 3D local is the default value. The difference between global and local is fairly simple; each object has its own local coordinate system, choosing that option will cause the shader to look identical even when animated. If global mode is selected, the appearance will differ based on the object's global location. UV mapping uses UV coordinates, this can often result in distortion; nevertheless, it is still useful, especially when used on an object you plan to morph.

- **Seed**: This value "seeds" the random number generator.

- **Scaling**: All four scaling values are useful, even in the 2D version of this extension. Experiment with different combinations of x, y, and z scales to observe the various effects.

- **Order and Persistence**: Please refer to the introduction to noise in "Noise."

- **Perturbation**: This parameter sets the amplitude of the turbulence. Higher values lead to a more knotty looking surface.

- **Grain Edge**: The default value for this parameter, 1, creates sharp edges between each band of wood. Decreasing this value smooth the transition between the bands.

- **Threshold Controls**: These controls behave identically to the threshold controls in Noise Factory. See "Noise Factory."

- **Plank Controls**: Controls the density of planks (if desired). The default values of zero provide no paneling. Increasing the density increases the number of bands in each plank. To increase the amount of planks visible, scale the texture accordingly.

- **Z Offset**: This parameter creates random variation in the Z length of any given plank. The higher the value, the greater the average offset.

- **Value Variation**: This parameter controls the maximum amount by which the intensity varies from plank to plank.

Marble

The Marble function produces a marble pattern. It assigns a value of 100 to the veins and a value of 0 to the spaces between the veins. Like Spots and Wood, Marble is a 3D function.

To create realistic looking marble, use the Marble function in a layered shader to create layers of marbled depths throughout your object.

To set Marble options:

1. Adjust the Global Scale slider to set the size of the marble pattern in relation to the object.

2. Adjust the Vein count slider to set vein spacing. High values produce dense vein patterns; low values produce sparse vein patterns.

3. Adjust the Perturbation slider to control the size of the "waves" in the veins.

4. Adjust the Undulation slider to control the frequency of the waves in the pattern.

5. Adjust the Vein Blending slider to determine whether the transitions between veins and spaces are abrupt or gradual.

6. Use the Direction pop-up menu to set the orientation of the marble pattern in relation to the object.

7. Click the Shuffle button to randomize the marble pattern.

Objects shaded with 3D functions like Spots, Wood, and Marble appear to be carved or sculpted from solid blocks of material, but this appearance is only skin-deep:all the objects you create in Carrara, these objects are still "hollow." To illustrate this point, if you make a marble object partially transparent, you won't see veins running through the inside of the object.

Snow

The Snow shader simulates snow. When you apply snow to an object, it creates an effect according the surfaces of your object.

The snow component allows you to apply snow to positive shapes, negative shapes, or both types of shapes. For best results, you should use Snow with the mixer or multiply operators.
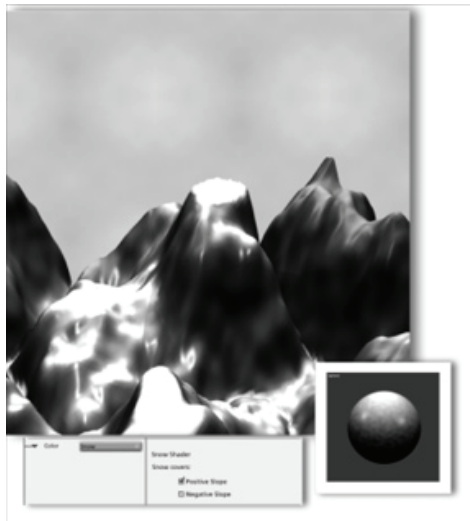
To set Snow options:

- Click the checkboxes to apply snow to positive, negative, or both types of shapes.

Spots

The **Spots** function produces a random pattern of spots. The spots are irregular, more like leopard spots than polka dots. You can use spots to create skin and building textures. Spots is a 3D function.
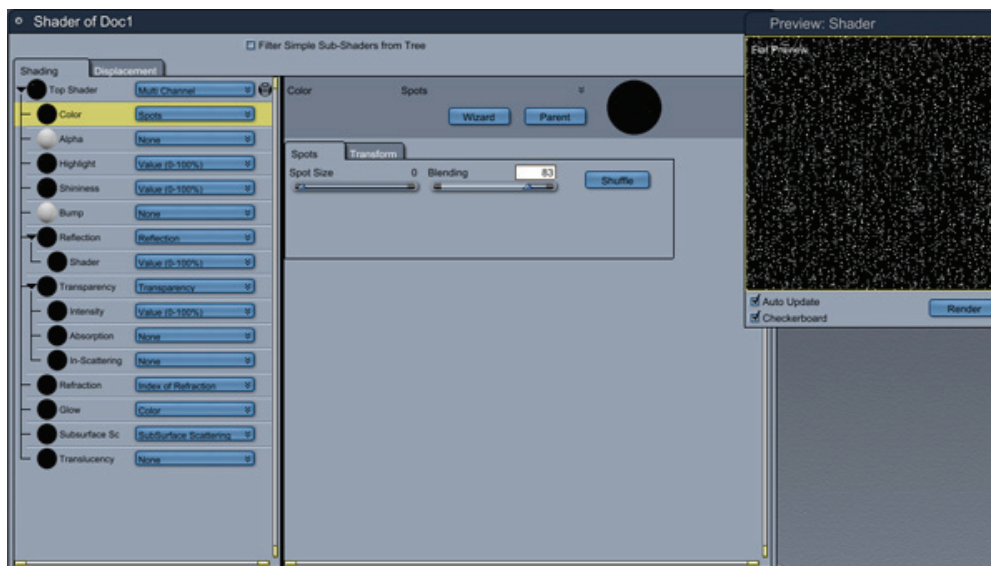
To set Spots options:

1. Adjust the Size slider to control the size of the spots.

2. Adjust the Blending slider to control how the spots blend together.

   o When Blending is set to 0%, the function produces equal numbers of zero-value and 100-value spots, with gradual blending.

   o When Blending is set to 50%, the function produces equal numbers of zero-value and 100-value spots, with abrupt transitions.

3. Click the Shuffle button to randomize the spot pattern.

Water: Ripples

The Ripples component simulates ripples in water. By adjusting the parameters of concentric waves, you can simulate disturbances on the surface of a calm pool of water. You can animate this shader. It looks best when placed in the Bump channel.
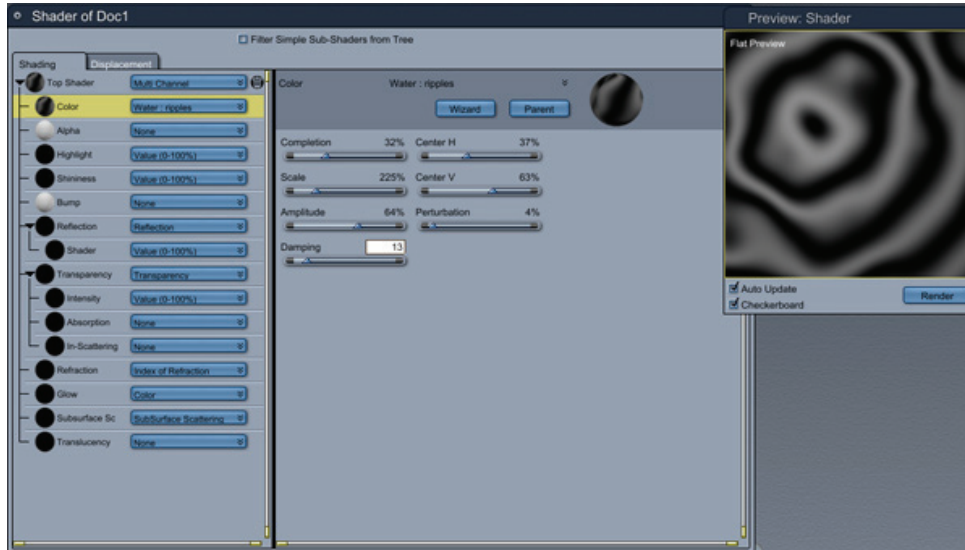
To set Ripple options:

1. Use the Time parameter to set the appearance over time when creating an animation. Set the value to 0% in the first frame and to 100% in the last frame.

2. Set the Scale parameter to control the spacing of the ripples. Low values create small, closely spaced ripples; high values create fewer, widely spaced ripples.

3. Set the Amplitude parameter to control the height of the ripples. When applied to the Bump channel, higher values will create a greater degree of depth.

4. Set the Damping parameter to create a fading effect in the ripple. Higher values cause the ripples to appear to fade as they move away from the center.

5. Set the Center H and Center V sliders to control the horizontal and vertical position of the center of the ripples.

6. Use the Perturbation slider to control the amount of distortion in the ripple. This parameter is dependent on the size of the object to which the shader is applied. To obtain good results on small objects, use very small values (2% to 9%).

To create a body of water:

1. In the Assemble room, add a Plane object.

2. Scale the plane as desired.

3. Create a Multi-Channel shader in the Shader Editor Tree and apply it to the object.

4. Set the color to blue.

5. Set the transparency to 30%.

6. Apply the Ripple shader to the Bump channel and set parameters as desired.

To create a shader with more than one ripple effect, use the Plus operator (+) and set different Center H and Center V values for each ripple.

Water: Waves

The Wave shader simulates water environments such as oceans and lakes. By adjusting the parameters, you can simulate a wide range of wave effects ranging from a calm sea to a turbulent ocean. You can also animate this shader. It works best when you apply it to the Bump channel.
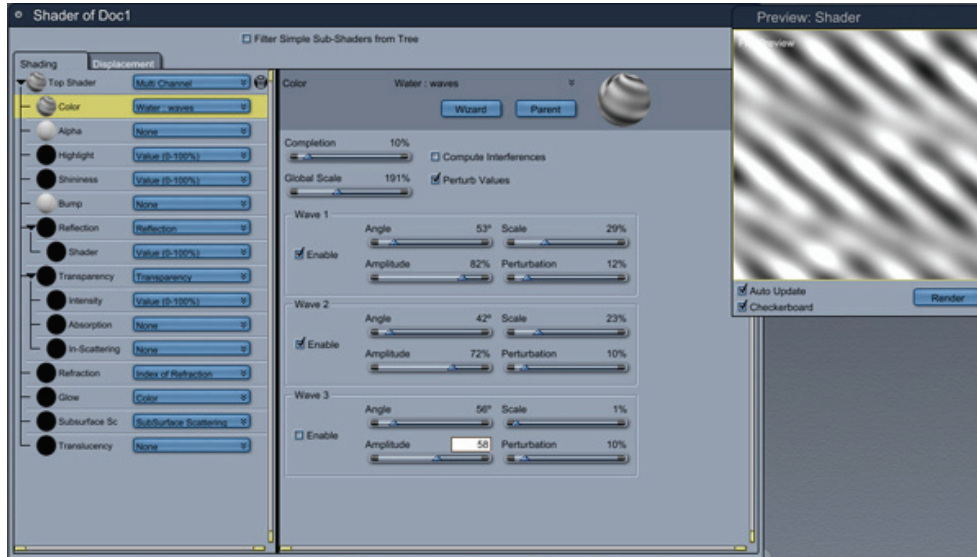
To set Wave options:

1. Use the Time parameter to set the appearance over time when creating an animation. Set the value to 0% in the first frame and to 100% in the last frame.

2. Set the Scale parameter to control the spacing of the waves. Low values create small, closely spaced waves; high values create fewer, widely spaced waves.

3. Set Compute interferences to control the interactions between waves. When enabled, waves can eliminate each other. When disabled waves can be added on top of each other.

4. Set the Perturb value to add extra depth to the waves. When disabled, waves appear flatter, with less variation. When enabled, the waves appear to have more depth and color gradations.

5. Click the Add checkboxes to add additional waves to your shader.

For individual waves, you can set the following options:

*   **Angle** controls the orientation of the wave.

*   **Amplitude** controls the height of the wave. This is most effective when you apply it to the Bump channel.

*   **Scale** controls the frequency of ripples in individual waves.

*   **Perturbation** controls the distortion of the wave. Higher values create more agitated waves.

Scale and **Perturbation** are dependent on the size of the surface on which they are applied. Use lower values when you apply them to smaller objects.

To create a body of water:

1. In the Assemble room, add a Plane object.

2. Scale the plane as desired.

3. Create a Multi-Channel shader in the Shader Editor Tree and apply it to the object.

- Set the color to blue.

- Set the transparency to 30%.

- Apply the Waves shader to the Bump channel and set parameters as desired.

You can also use the Waves shader to create a denim texture by mixing blue and white in the Color channel. You can also create Sand by mixing brown and yellow, setting a high perturbation value, and turning on Perturb values.

Wood

The **Wood** function produces a wood grain pattern. It assigns a value of 100 to the veins and a value of 0 to the spaces between the veins. Wood is a 3D function.
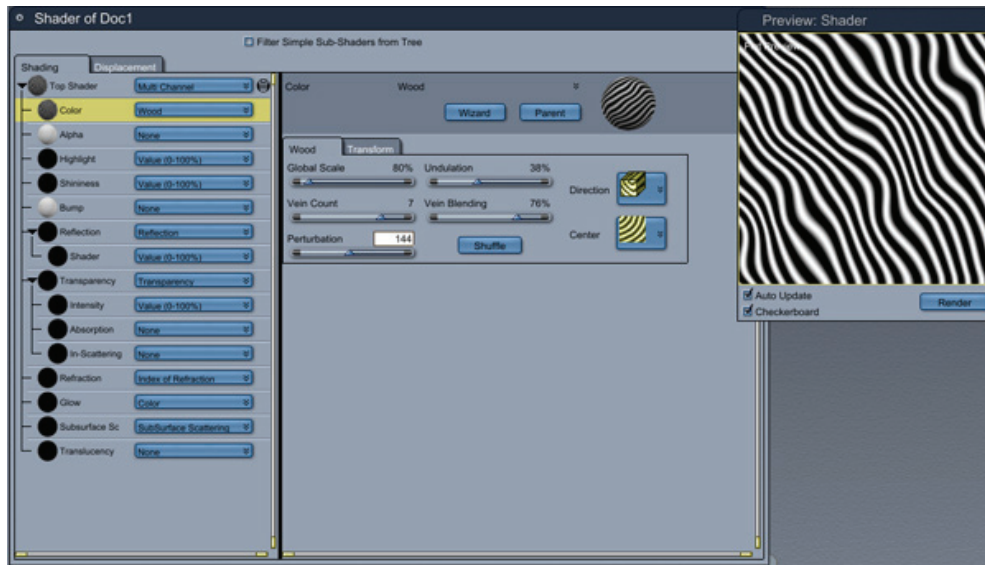
To create different wood textures, use the Wood function as a sub-shader in a Layers List shader to create more realistic types of wood.

To set Wood options:

1. Adjust the Global Scale slider to set the size of the wood grain pattern in relation to the object.

2. Adjust the Vein count slider to set vein spacing. High values produce dense vein patterns; low values produce sparse vein patterns.

3. Adjust the Perturbation slider to control the size of the "waves" in the veins.

4. Adjust the Undulation slider to control the frequency of the waves in the grain.

5. Adjust the Vein Blending slider to determine whether the transitions between veins and spaces are abrupt or gradual.

6. Use the Direction pop-up menu to set the orientation of the wood grain pattern in relation to the object.

7. Use the Center pop-up menu to set whether the wood grain pattern is taken from the center of the tree, where the curvature of the veins is quite pronounced or from a point farther from the center, where the curvature is more gradual. There are three settings to choose from.

8. Click the Shuffle button to randomize the wood grain pattern.

Environment Functions

Environment

The environment shader can be used to shade landscapes. The black and white image generated can be modified with the altitude, the slope and the orientation of a geometry.

3 tabs of settings are available to modify this shader.

Altitude
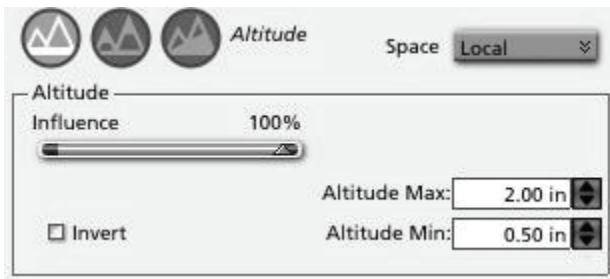
- **Influence**: the influence of the altitude can be set between 0 and 100%. An influence of 0% means that the altitude parameter won't modify the shader at all.

- **Altitude Min**: the altitude from which the grayscale will start. If under this altitude setting, the shader will be all white.

- **Altitude Max**: the altitude to which the grayscale will go. If over this altitude setting, the shader will be all black.

- The **Invert** check box inverts the grayscale values.

Slope
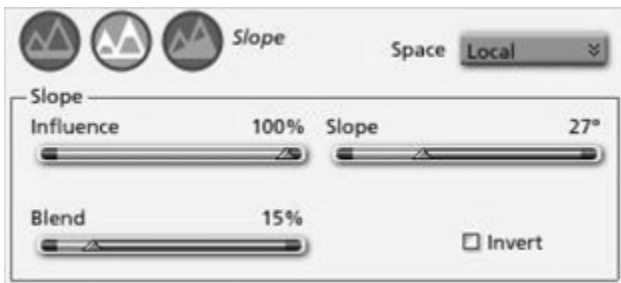
- **Influence**: the influence of the slope can be set between 0 and 100%. An influence of 0% means that the slope parameter won't modify the shader at all.

- **Slope**: set the slope value in degrees here.

- **Blend**: blends around the Slope value.

- The **Invert** check box inverts the grayscale values.

Orientation
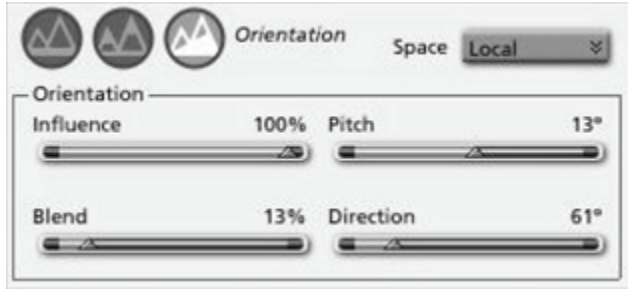
- **Influence**: the influence of the Orientation can be set between 0 and 100%. An influence of 0% means that the Orientation parameter won't modify the shader at all.

- **Blend**: blends around the Orientation value.

- **Pitch**: angle of the orientation axis with the z axis. Modifies the up/down position of the orientation axis.

- **Direction**: the direction in the ground plane.

Slope

The Slope function creates a pattern based on the slope of the surface. It assigns a value of 0 to everything below the slope and a value of 100 to everything above. Slope is a 3D function.
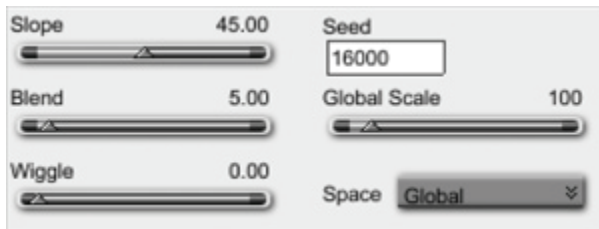
Use the Slope function to create interesting effects.

To use slope:

1. The **Seed** value provides the starting point for the **Wiggle** function.
2. The Global **Scale** slider stretches the **Wiggle** function.
3. **Slope** sets the dividing line between light and dark.
4. **Blend** adjusts the distance over which the Slope change occurs.
5. **Wiggle** changes the amount of variation in the dividing line.
6. **Space** selects from between Global and the object's Local coordinates.

Elevation

The Elevation function creates a pattern based on height. It assigns a value of 0 to everything below the height and a value of 100 to everything above. Elevation is a 3D function.

To create an elevation:

1. The **Seed** value provides the starting point for the **Wiggle** function.

2. The **Scale** slider stretches the **Wiggle** function.

3. **Height** sets the dividing line between light and dark.

4. **Blend** adjusts the distance over which the Height change occurs.

5. **Wiggle** changes the amount of variation in the dividing line.

6. **Orientation** changes the axis that drives **Height**.

7. **Space** selects from between Global and the object's Local coordinates.

Tips and Tricks

- Elevation isn't just for outdoor scenes. Use it for oozing barrels, flames, a sharpened pencil, or anything else where the color changes based on distance along an axis.

- A good rule of thumb for the **Space** option is, if it moves set it on Local, if not set it on Global.

Noise Functions

Fractal Noise

Fractal noise makes a random noise pattern between black and white. The fractal nature of this shader gives more detail and subtlety to the pattern.
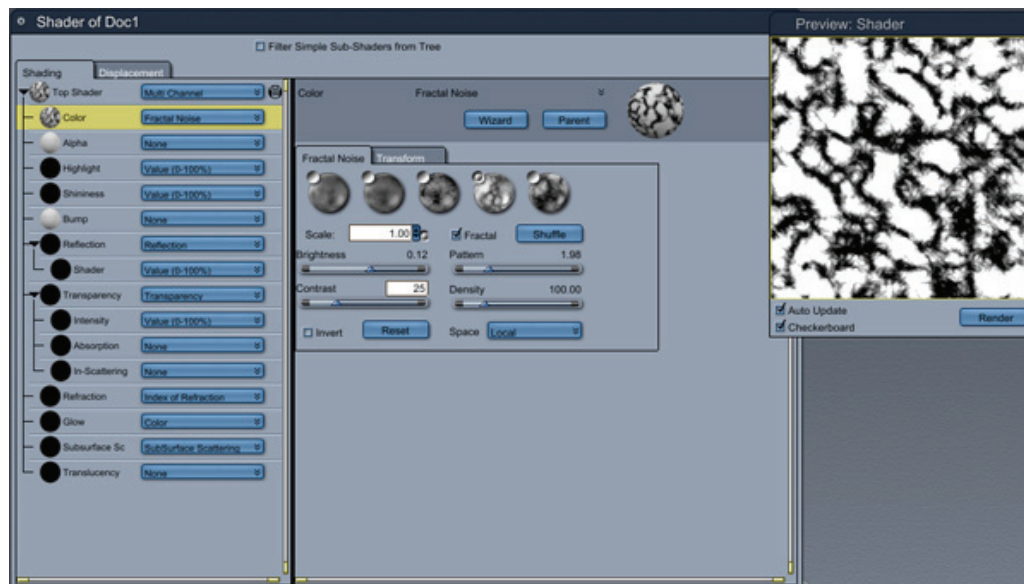
Five noise presets are available. They can be selected by clicking on the corresponding images.

The following parameters can then be adjusted:

- **Scale**: modifies the global scale of the texture.

- **Brightness**: specifies the average value of the noise from a black value (-1) to a white value (1).

- **Contrast**: sets the scale of the oscillations of the noise around the average value.

- The **Pattern** and **Density** parameters modify the way the fractal computation is done and are only available if the Fractal option is checked.

- The **Invert** check box inverts the grayscale values.

- **Reset** restores the default values.

- **Shuffle** makes the random generator build a new set of values.

- **Noise** can be set in a local space or a global space.

Noise

The Noise function produces a noise pattern.
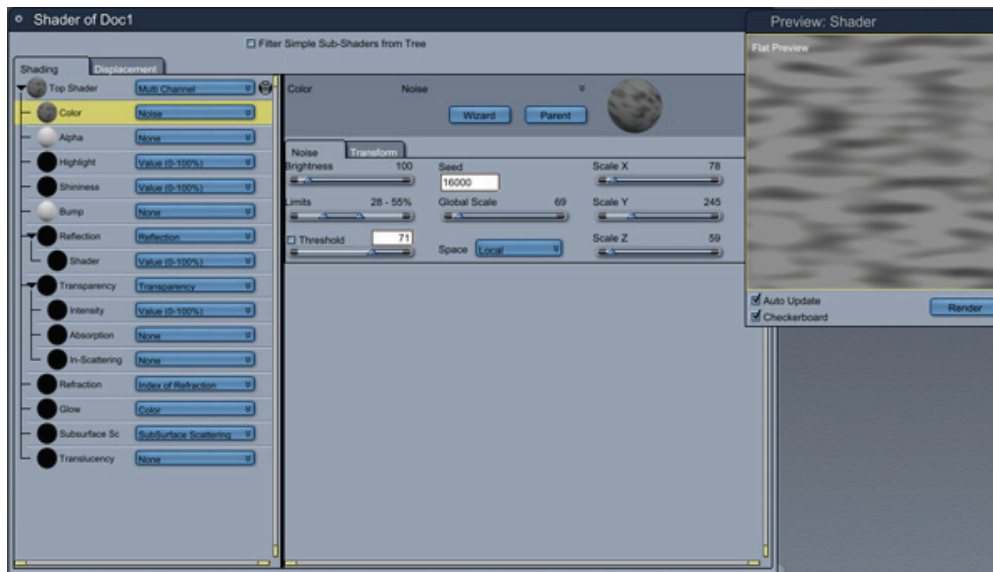
To use noise:

1. The **Seed** value provides the starting point for the Turbulence function.

2. Brightness controls how light or dark pattern is.

3. Global Scale makes the pattern larger or smaller.

4. Scale X makes the pattern larger or smaller along the X axis. Scale Y makes the pattern larger or smaller along the Y axis. Scale Z makes the pattern larger or smaller along the Z axis.

5. **Space** selects from between Global, Local, and UV coordinates.

6. Threshold makes the pattern black or white depending on if the grayscale value is greater or lower than the threshold value.

7. Limits constrain the high and low values.

Noise Factory

Noise is simply a field of random numbers. A type of noise that is more interesting is referred to as Perlin noise, or turbulence. Generally speaking, these words are interchangeable.
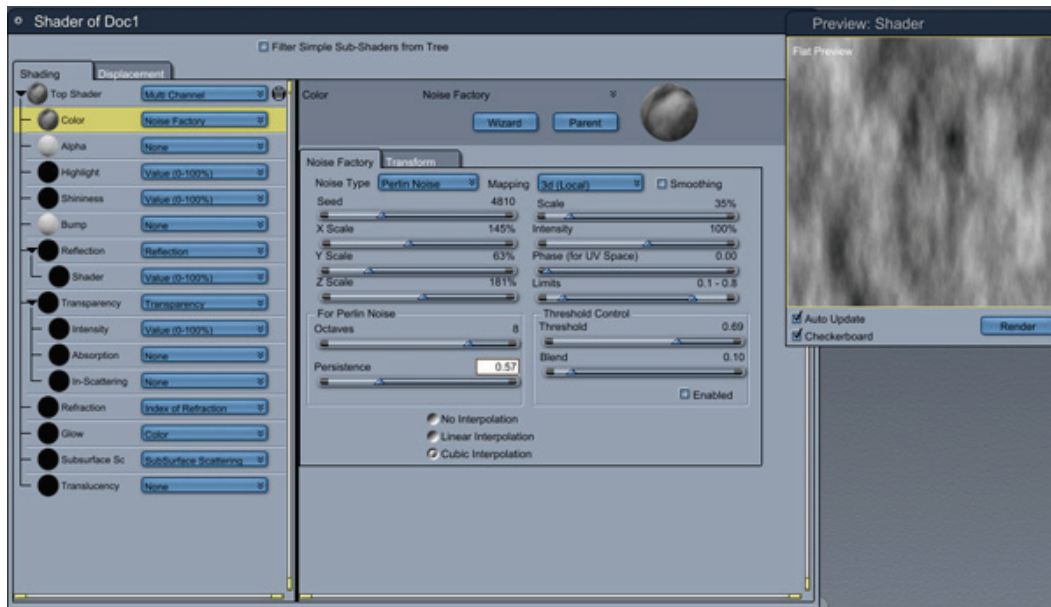
Perlin noise is a technique used to impose self-similarity on noise. Perlin noise algorithms are used to generate, among other things, clouds, marble, wood, and jitter effects. A universal parameter of Perlin noise is the octave parameter. Each successive octave of noise has a frequency twice that of the previous octave. Therefore, each time you increase the octave parameter, the Perlin Noise becomes more detailed and intricate. Some shader extensions have an order parameter instead of an octave parameter. When order is a parameter, the extension is using so-called "reverse" Perlin noise. In this case, a low order gives a very noisy result, and each successive increase imposes more order on the noise. Noise Factory uses the traditional Octaves, and Lumber Yard uses order. Another parameter that is common to Perlin noise, is persistence. This parameter defines the rate at which the amplitude decreases for each successive level of noise/order. Higher values of persistence lead to greater levels of noise for both regular and reverse Perlin noise.

Options

- **Noise Type**: Choose between three - Basic Noise, "Linoleum" Pattern, and Perlin Noise. Experiment with the linoleum pattern to understand what it does.

- **Mapping**: Choose between three - 3D Local, 3D Global, 2D UV. 3D local is the default value. The difference between global and local is fairly simple; each object has its own local coordinate system, choosing that option will cause the shader to look identical even when animated. If global mode is selected, the appearance will differ based on the object's global location. UV mapping uses UV coordinates, this can often result in distortion; nevertheless, it is still useful, especially when used on an object you plan to morph.

- **Smoothing**: When checked the shader averages the value of each pixel with its neighbors.

- **Seed**: This value "seeds" the random number generator. It is important to note that if Noise Factory is used twice, and all the values remain the same, both instances will be identical.

- **Scaling**: These parameters control scaling.

- **Intensity**: Controls the brightness of the noise.

- **Phase**: Can be used to animate the texture. This really only works in 2D mode, in 3D mode it will appear as if the texture is simply raising vertically. This can be useful for glowing effects, and so on.

- **Limits**: Upper and lower limits. Any values that fall below the lower limit will be set to that number, any reaching above the upper limit will be set to that limit.

- **Perlin Noise Parameters**: Read the introduction to Perlin noise to learn more about its parameters.

- **Threshold controls**: Very similar to limits. The threshold is a middle value, that when enabled, any numbers falling below it will be set to 0, any above it to 1. The blending value blends values close to the threshold.

- **Interpolation Settings**: Three options - none, linear, and cubic. When None is used, any values that occupy more than one pixel are not interpolated, giving them a harsh, pixelated look. This can be desirable in certain situations. Linear and cubic are fairly similar, one uses a simple linear function to perform interpolation, and the other uses a cubic function. Cubic, generally speaking, is more CPU intensive and looks better, though at a distance, it is nearly impossible to distinguish linear from cubic.

Turbulence
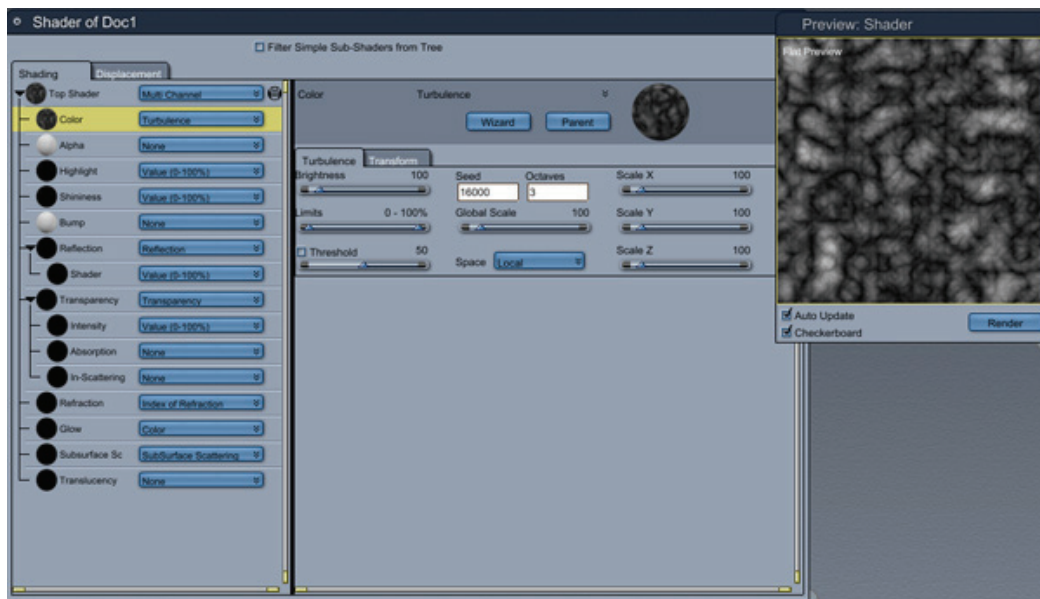
To use turbulence:

- The **Seed** value provides the starting point for the Turbulence function.

- **Octaves** controls the number of layers in Turbulence, each layer is finer than the last.

- **Brightness** controls how light or dark pattern is.

- **Global Scale** makes the pattern larger or smaller.

- **Scale X** makes the pattern larger or smaller along the X axis.

- **Scale Y** makes the pattern larger or smaller along the Y axis.

- **Scale Z** makes the pattern larger or smaller along the Z axis.

- **Space** selects from between Global, Local, and UV coordinates.

- **Threshold** makes the pattern black or white depending on if the grayscale value is greater or lower than the threshold value.

- **Limits** constrain the high and low values.

Hair (Standard, Pro)

For information on hair shaders, see "Dynamic Hair and Fur."

Layer Components

You can create layer components with the Layers tools or you can add them in the Shader Tree Editor, then size and position them numerically. Each layer is displayed in a Layers List shader and can have a sub-shader applied to it, as well as an optional opacity mask sub-shader.
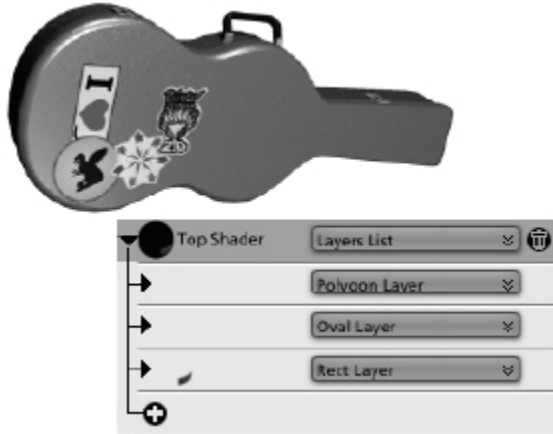
- **Oval**: The Oval layer creates a round or oval layer.

- **Rectangle**: The Rectangle layer creates a rectangular or square layer.

- **Polygon**: The Polygon layer creates a layer with multiple sides.

- **CoverAll**: The CoverAll layer creates a layer that covers the entire UV space of the object.

You can adjust the layer order, layer position, and opacity of a layer in a Layers List shader.

To change the layer order:

1. Click a layer to select it.

2. Drag the layer to a new location within the list. The Shader Tree Editor changes to indicate where you can place the layer.

3. Release the mouse button to drop the layer into the new location. The preview updates to show your change.

To adjust the position of a layer:

1. Click a layer to select it.

2. Drag it into a new location in the Preview window, or type the numeric values for the new location in the Shader Tree Editor. The preview updates to show your change.

To adjust the layer opacity:

1. Click Opacity Mask and choose Value from the pop-up menu.

2. Drag the slider or enter a number to choose a new value. The preview updates to show your change.

To specify a different compositing method:

- Click Opacity Mask and choose one of the other shaders from the pop-up menu.

Using Other Components as Functions

You can actually use any component as a function in the Mixer operator, or in the Blender channel. This allows you to mix components in ways not supported by Carrara's basic functions.

When you use a component as a function, Carrara automatically converts its output to values so that the Mixer operator or Blender channel can use it to mix the components on the Source 1 and Source 2 branches.

Value and Texture map components are especially useful in place of functions.

- You can use a Value component to blend the components on the Source 1 and Source 2 branches of the Mixer operator in a given ratio. A value of 50 blends the two components evenly. Lower values favor the Source 1 component; higher values favor the Source 2 component.

- You can use a black and white or grayscale texture map to mix the Source 1 and Source 2 components in a custom pattern. For each white pixel in the image, the Mixer operator uses the Source 1 component. For each black pixel, the Mixer operator uses the Source 2 component. Intermediate shades produce a blending of the two components.

You can also use a shader or reference shader as a Blender component.

3D Painting (Pro)

In Carrara 7 Pro, 3D Painting was added as part of the Assembly room. A texture can be edited, selected, or assigned to an object for various channels and shading domains will also be taken into account. Only simple shaders made of textures can be recognized by and edited in the 3D Paint tool.

The 3D Paint tool supports layers, image stamping, tablet support, and the ability to save a brush for later use.

Load or Create New Texture

1. Create or build the object to be painted.

2. Click on the **3DPaint tool**.

If the object does not have a texture active, a message dialog will come up letting you know that one needs to be added. Click on **OK** to close the dialog. If you do not wish this message to come up each time you start painting a new object, click in the checkbox next to **Skip from now on** before clicking on **OK**.
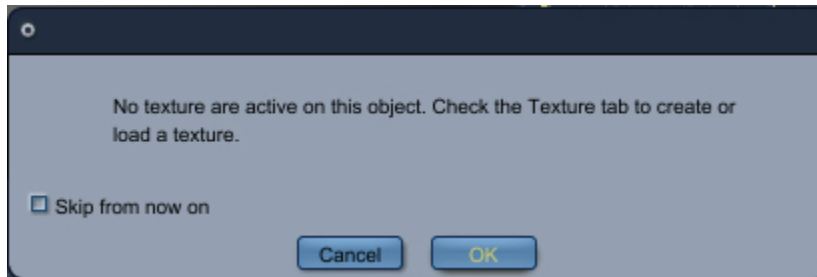
3. The **Properties Tray** for the tool will activate with the **Textures** tab active.
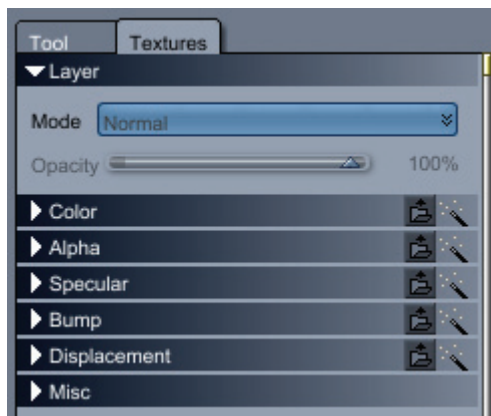
If you click on any of the arrows next to the channels, it will show that there are no textures assigned to the channel.

4. Left-click on the **Open Texture** icon to bring up the file open dialog where you can locate and select a texture for that channel.

OR

5. To create a new texture, click on the **New Texture** icon.
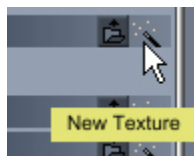
A message dialog will come up cautioning you that the instance shader will be changed. Click on **OK** to continue or **Cancel** to stop the change. If you do not wish this message to come up each time you create a new texture, click in the checkbox next to **Skip from now on** before clicking on **OK** or **Cancel**.
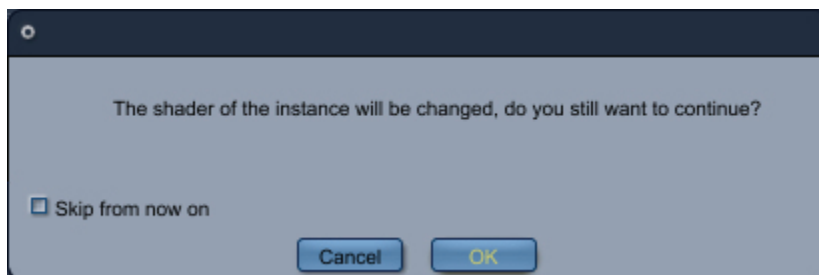
6. If loading a texture, the **Load Texture panel** will come up.
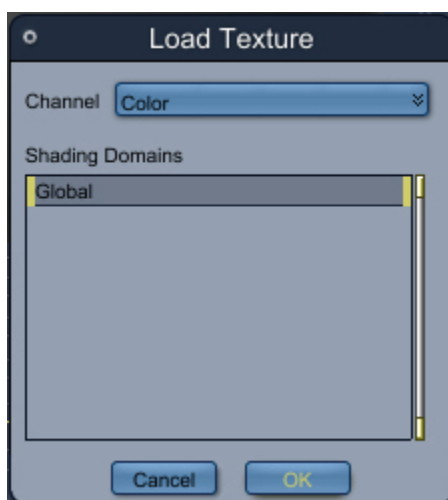
Select the needed channels from the **Channel menu** then click on **OK** to load the texture. Repeat as needed.
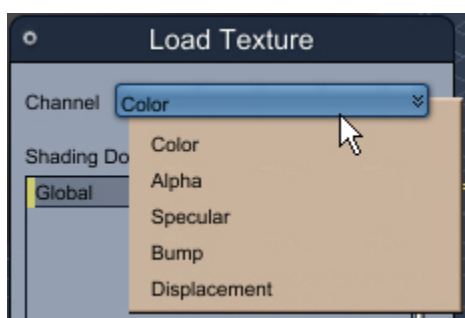
7. If creating a texture, the **Create Texture panel** will come up with the current channel selected.
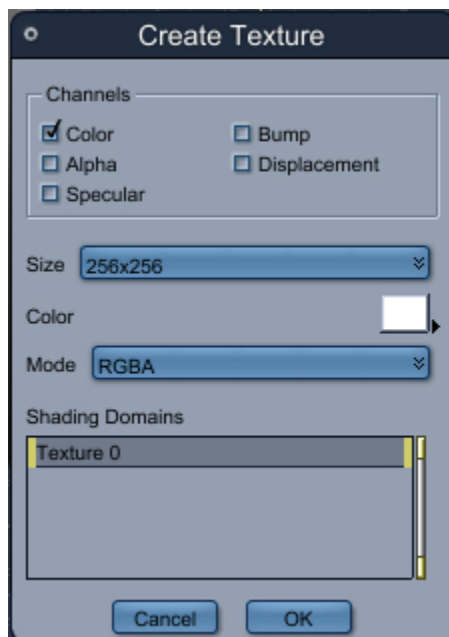
Select which channels to create a new texture for by clicking in the checkbox next to each channel.
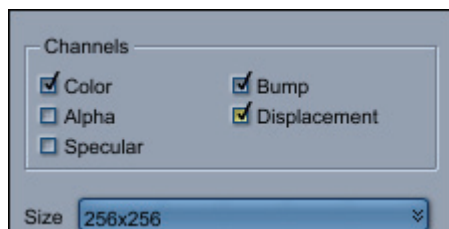
8. From the **Size menu**, select the dimensions of the texture(s).
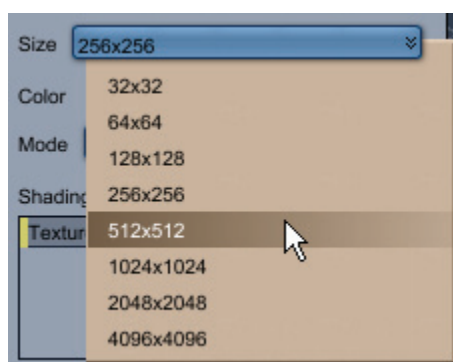
9. From the **Color chip**, left-click on the chip to bring up the **Color Picker** or on the arrow next to the chip to bring up the gradient color picker. Select a color for the image background from the applicable color picker.
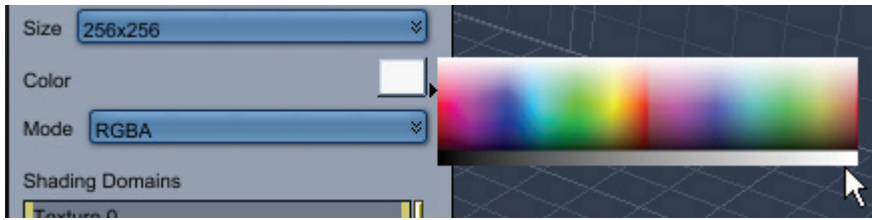
10. Select the color mode from the **Mode menu**, either RGB or RGBA (RGB with alpha):

11. In the **Shading Domains** section, if there is more than one domain available, left-click on the one to use for this texture.

12. Click on **OK** to accept the setting for the new texture. The **Save File dialog** will come up. Select a file type from the file type menu, enter a file name, locate the save directory, and click on **Save**. Repeat for each texture for each channel selected, i.e., one for color, one for bump, one for displacement, etc. As each file comes up for saving, the file type will revert to .gif format, so if you need a different format, make sure you change it for every texture.
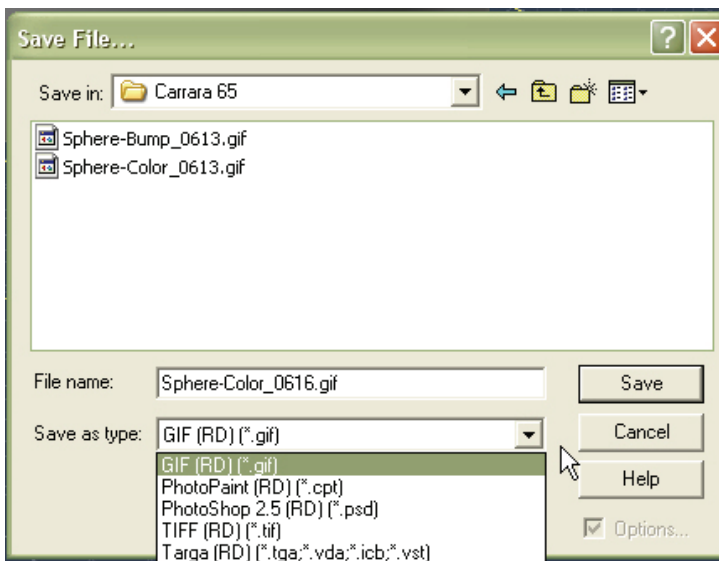
TIP! If the texture you will be creating will contain layers, select PhotoShop as your file type.

13. The texture will be applied to the object.
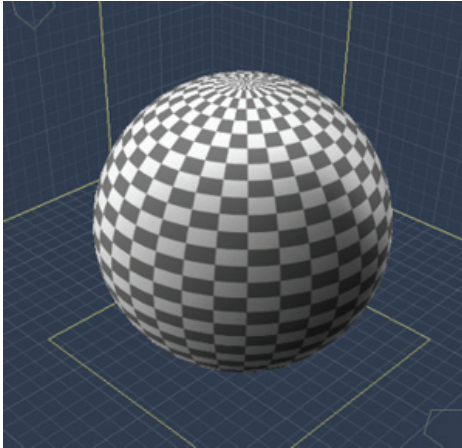
and the Texture tab will be updated to show the new or loaded textures for the selected channels. Each channel group can have multiple layers which overlay or underlay other layers. See "Layer Properties and Options" for more on working with layers.
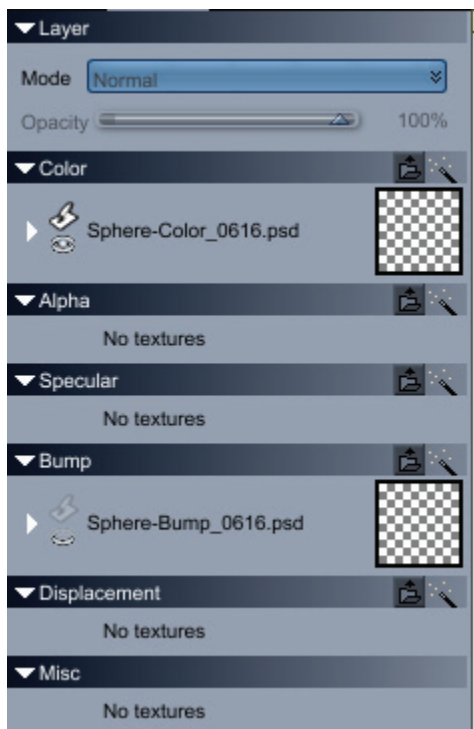
14. Click on the arrow next to the layer group you will be working with then select the actual layer to paint on. Make sure that when you need to paint on a different layer or a layer in a different channel group, that you select that layer before you begin painting.

NOTE! When you begin to paint and nothing happens, change to the Textures tab and make sure the correct layer is selected.

Paint Tools

When the 3D Paint mode is active, the 3D Paint toolbar is activated with six different tools available. The image below shows the available tools, from left to right: Paint Brush, Air Brush, Stamp, Line, Erase Brush, and Eyedropper.

Each tool is described below along with its available options.

Paint Brush

The **Paint Brush** is the default 3D Paint tool and is the tool automatically activated when the 3D Paint mode is selected.

Symmetry and Color

1.   Click on the **Tool tab** to activate it. This tab contains three sections: **3D Paint**, **Color**, and **Tool**.
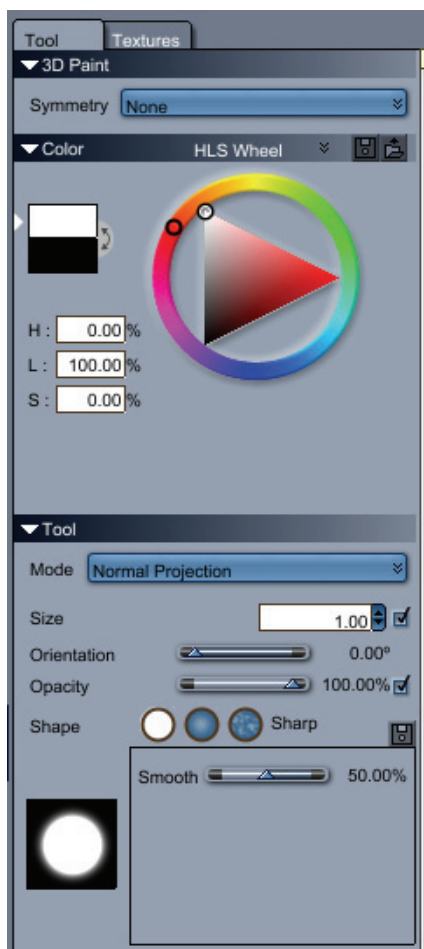
2.   In the **3D Paint section**, select whether to use symmetry on a specific axis or not (None). This will determine whether the painting on the object will only be where you paint or if the texture will be mirrored on the selected axis.
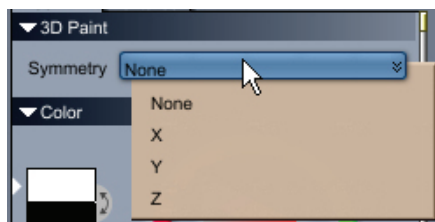
3. Select a color to be used as the paint color in the **Color section**. See "Color Pickers" for more on choosing colors.

TIP! If you will be using an image on the brush or as a stamp and need the actual colors of the image to show through, set the brush color to white, otherwise, the image will take on the color of the brush.

Tool Options

In the **Tool section,** choose the options for the brush mode, size, orientation, opacity, shape, and smoothness.

- **Mode**: From the Mode menu, select either **Normal Projection** or **Fast UV**.
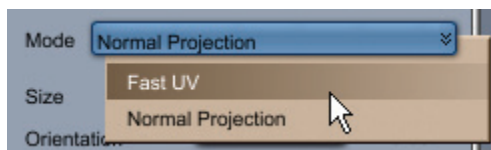
- **Size**: Set the brush size by entering a number in the field or clicking on the up/down arrows. If using a tablet, click in the checkbox next to **Link to tablet pressure** to link the brush size to your tablet. This option will use the selected brush size as the starting point for the brush size, then adjust as needed based on the current pressure applied to the tablet.

The brush size can also be changed on the fly by pressing the **Ctrl** key, then left-clicking on the object and dragging left to decrease the brush size or right to increase the brush size.

The image below shows examples of the brush size at 0.5, 1.0, and 2.0.
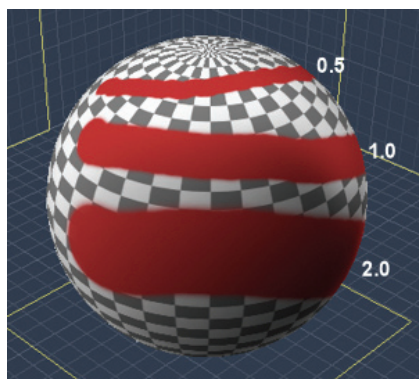
- **Orientation**: Select an orientation for the brush by dragging this slider left or right, or by left-clicking in the value field and entering the degrees manually. This option is only applicable when an image is being used on the brush.
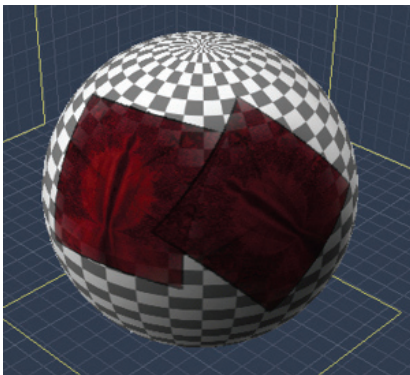
- **Opacity**: Set the opacity for the brush by dragging this slider left or right, or by left-clicking in the value field and entering the percentage manually. If using a tablet, click in the checkbox next to **Link to tablet pressure** to link the brush opacity to your tablet. This option will use the selected opacity setting as the starting point for the brush, then adjust as needed based on the current pressure applied to the tablet.

The first image below shows the opacity at 100%, 50%, and 25%, while the second image shows how painting over the areas increases the amount of paint applied as each brush stroke overlays the stroke below.
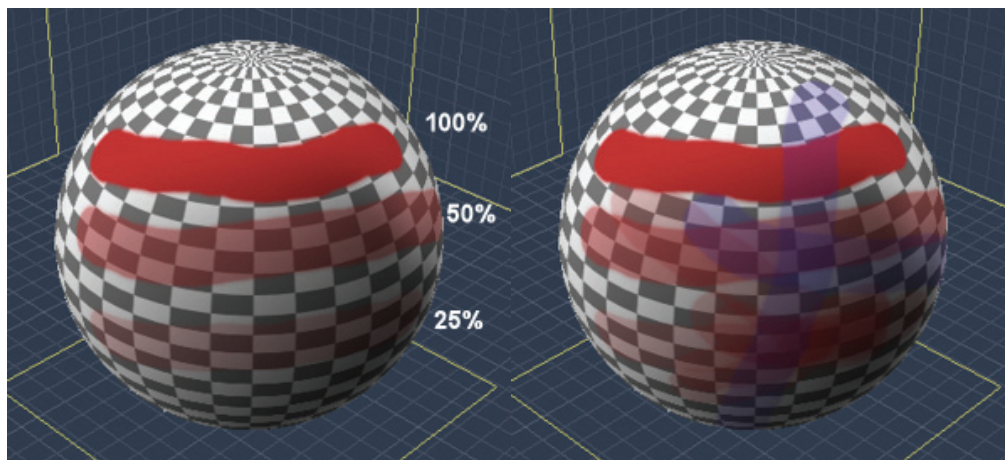
- **Shape**: The shape of the brush refers to whether it is sharp, smooth, or uses an image.
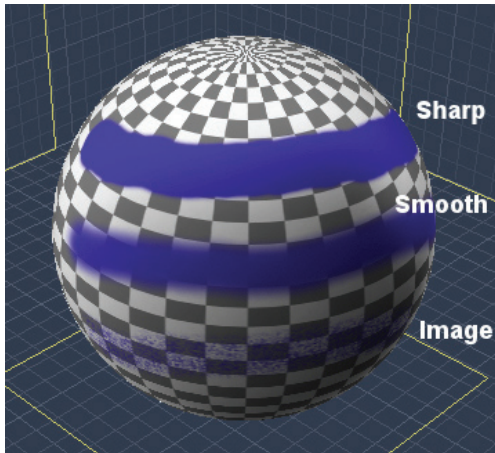
If the brush shape selected is **Image**, the **Image properties** will activate. See "Image Shape" for more on using an image on the brush.

- **Smooth**: Set how smooth the brush is by dragging this slider left and right, or by left-clicking in the value field and entering the value manually. This option is only applicable when the brush is set to the Sharp or Smooth shapes. The default for the Sharp shape is around 27% while the default for the Smooth shape is 50%. Set the smoothness lower for an even sharper brush or higher for a smoother one.

Image Shape

When using the Image shape brush, an options palette is activated with various options available for selecting an image and setting brush properties.
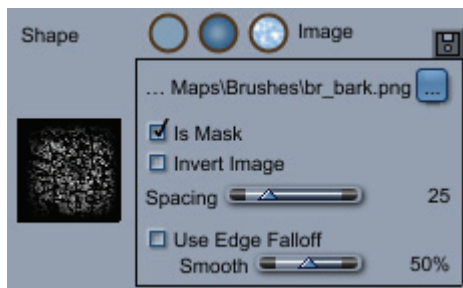
- **Select File**: Click on the **Browse** button to open the **Open File** dialog. Select a file type from the file type field, locate the file, select it, and click on **Open**. The selected image will show up in the thumbnail to the left of the brush options.
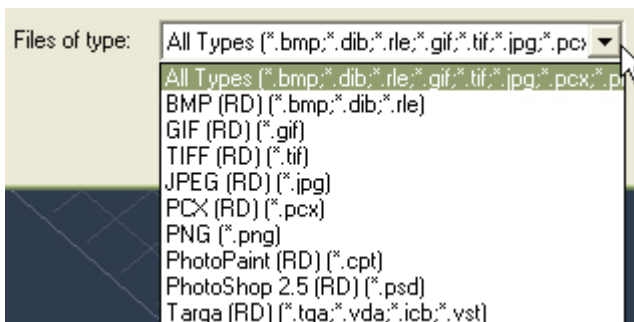
- **Is Mask**:

  - If the **Is Mask** option is **selected**, the loaded image will be used as a mask with the current brush color used as the "stamp" color. The image below shows an apple image

that has a black background so the background is opaque while the apple is partially opaque.
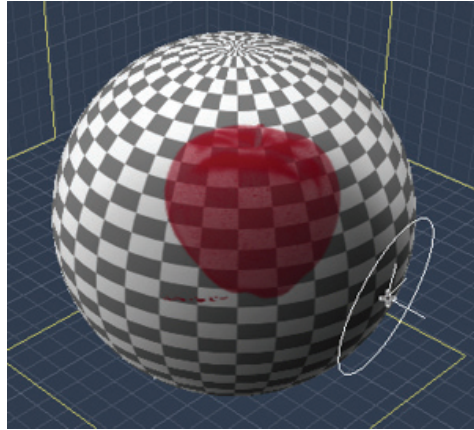
- If the **Is Mask** option is **deselected**, the loaded image will be "stamped" as is on to the object.
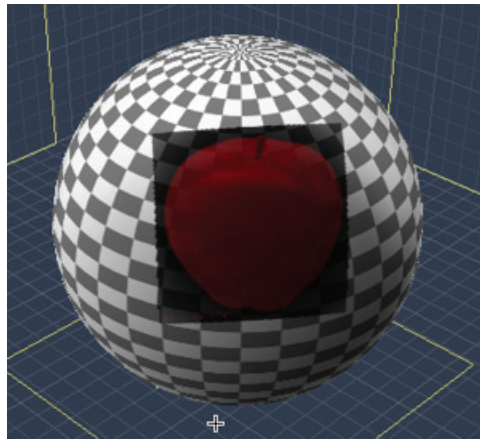
- **Invert Image**: When selected, this option will create a negative image of the loaded image.
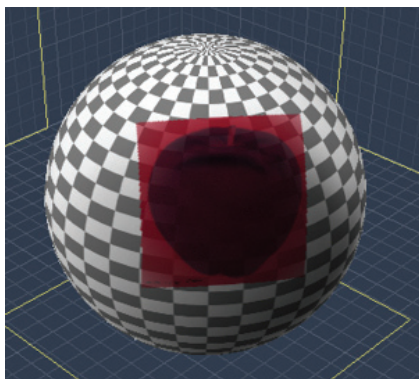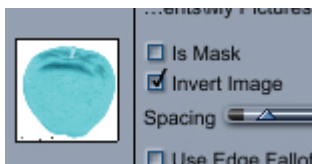
- **Spacing**: When an image is loaded as a brush, the spacing is set to 25 by default. This allows the image to be painted (sprayed) on to the object with the image overlapping as you paint. Adjusting the setting higher spaces the images further apart. Adjusting the setting lower will space the images closer together.
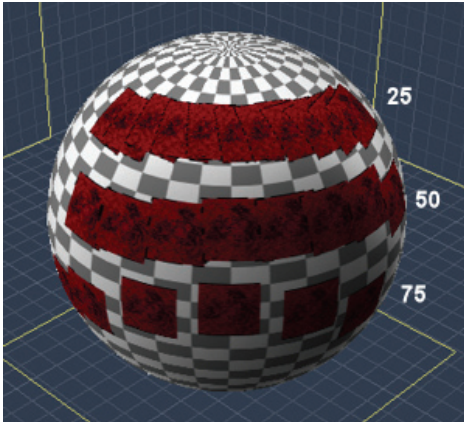
- **Use Edge Falloff**: Enables or disables edge falloff for the loaded image.

    - When deselected, the entire image is used as the brush.
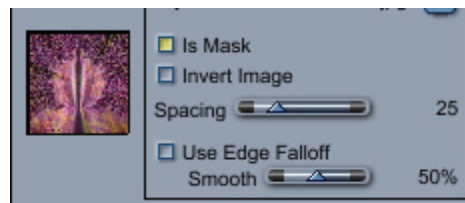
    - When selected, the edge falloff is applied based on the amount of smoothness selected from the Smooth slider.
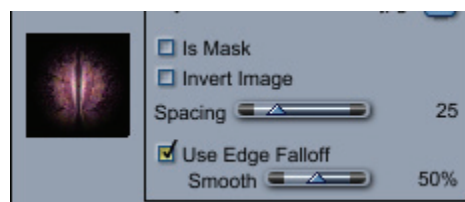
    Examples:
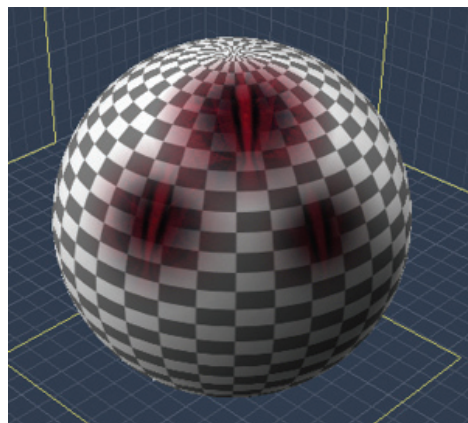
- **Orientation**: The orientation option is available under the Brush Size option. Drag the slider or enter a percentage in the value field to rotate the brush image. Only applicable if an image is loaded as a brush.
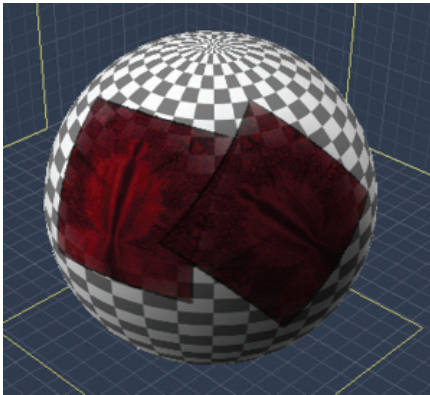
Air Brush

The **Air Brush tool** is very similar to the Paint Brush with Size, Orientation, and Opacity options, however, it adds two additional settings, Pressure and Rate, that can be linked to tablet pressure. The Air Brush is a softer version of the Paint Brush and works much the same as similar tools in many paint programs.
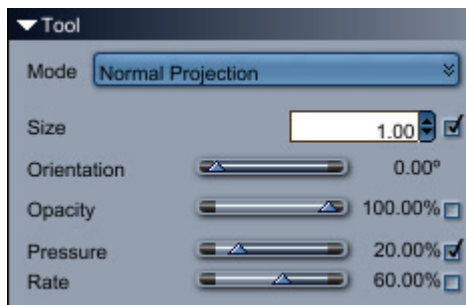
- **Pressure**: This setting determines how much "pressure" is applied to the Air Brush tool. A lower setting will result in a softer spray while a higher setting will result in a harder spray.

- **Rate**: This setting controls how quickly the paint is applied to the object as the user paints. So, at a low setting the paint will be very light such that more of the original color will still be visible while at a high setting, more paint will be applied.
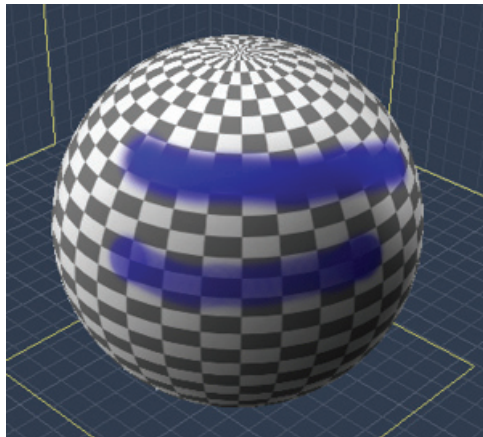
Stamp

The Stamp tool is similar to using an image as a brush except, instead of spraying the image as you paint, it stamps the image once each time you left-click on the object.
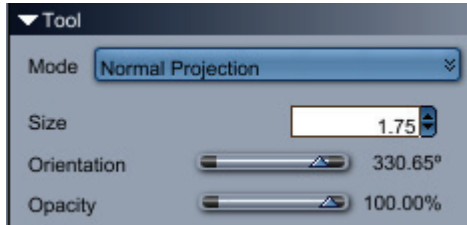
To stamp an image on to an object:

1. Select the **Stamp tool**.

2. Set the stamp size, orientation, and opacity.

3. Select the Image shape option.

4. Load an image and set options as shown in "Image Shape."

5. If the stamped image needs to have the exact colors as on the original image, set the brush color to white and make sure Is Mask and Invert Image are deselected.

6. Left-click on the object to stamp the image on to the object.

7. If the stamped image is too transparent, left-click on the same spot again to overlay another copy of the image. Repeat as needed.

To resize the stamp on the fly, left-click on the object and drag the mouse left to decrease the stamp size or right to increase it.

To rotate the stamp on the fly, left-click on the object and rotate the mouse left or right. You'll see the brush axes rotate as you do this. Let up on the mouse button to stamp the image at the current orientation.

Line

The **Line tool** provides options for the size, orientation, and opacity which are used when creating the line on the object.
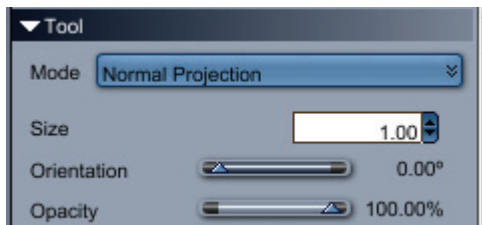
To add a line to the object texture:

1. Select the **Line tool**.

2. Set the line size, orientation, and opacity.

3. Left-click and hold on the object where you want the line to start.

4. Without letting up on the mouse button, drag the line where you want the line to end. A white line will extend from where you started to where the brush tool is currently located.
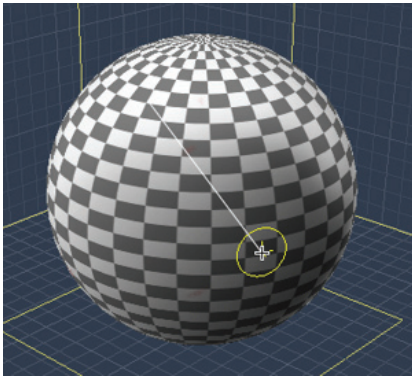
5. Let up on the mouse button.
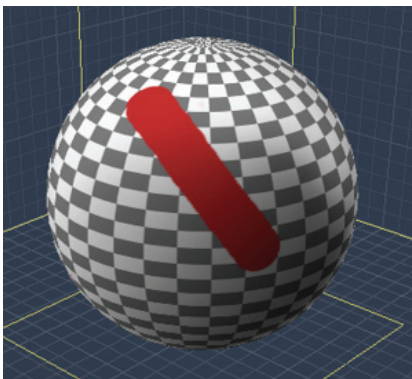
6. Repeat as needed.
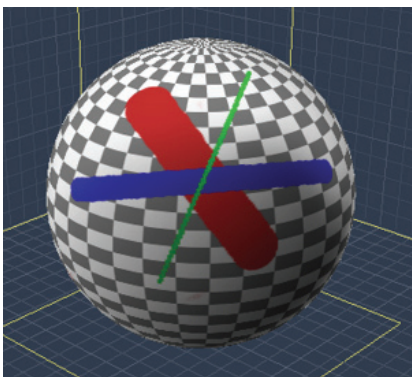
Eraser

The **Erase Brush** works like any other eraser and removed sections of the texture from the object based on brush size and opacity.
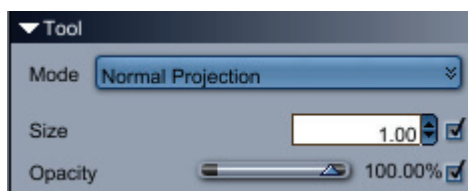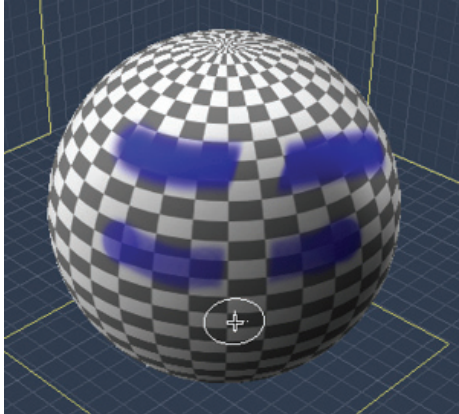
You can adjust the size and opacity of the eraser and link the pressure to your tablet, plus, if you use an image as the eraser, you can erase parts of the texture to create some cool effects.
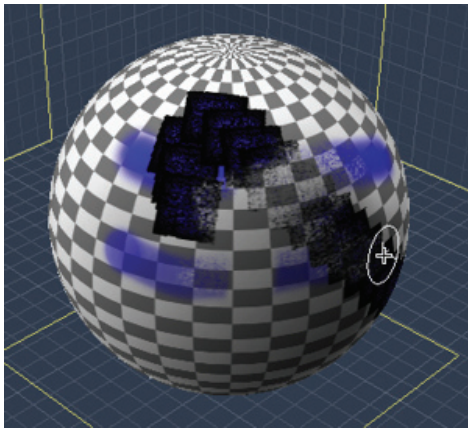
Eyedropper

The **Eyedropper tool** provides a means of selecting color from a section of the object's texture and replace the current brush color with the new color. Simply select the tool, move the mouse over the object texture, and left-click on top of the new color.
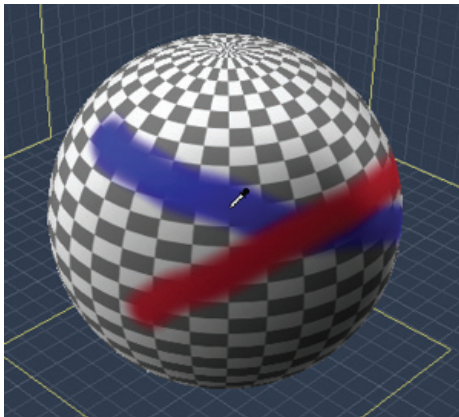
Layer Properties and Options

Each texture channel is created with a default background layer but additional layers can be added or removed as needed.

To paint on a specific layer, left-click on the layer in the listing to activate it.

As you paint on the object, the **Background layer** will update to show the painted texture.
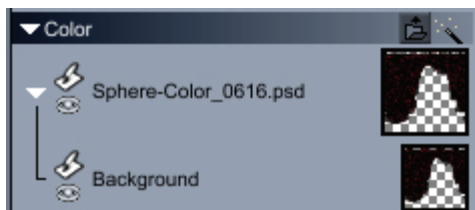
Right-click on the main texture thumbnail to access its context menu.
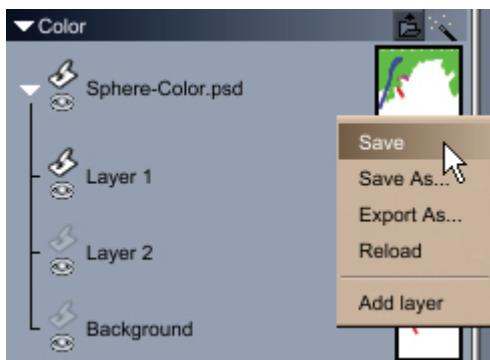
- **Save:** Saves the texture and any changes to file. No confirmation.

- **Save As**: Brings up a default save dialog to save the texture to a new file and the image file reference used by the texture is changed to use this new file. Available options shown below:
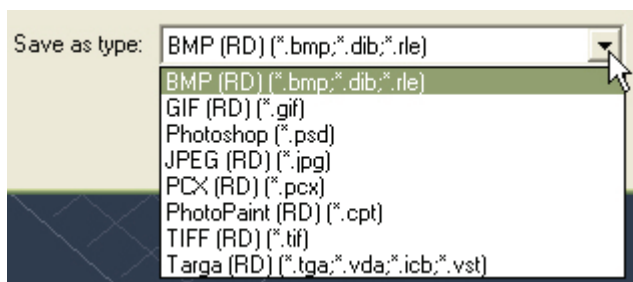
TIP! To save a texture with multiple layers, select the PhotoShop file type.

- **Export As**: Saves the texture to a new image file without changing the reference in the shader (so it will still point to the old file).

- **Reload**: Reloads the original blank texture. A confirmation dialog will come up to make sure you want to do this as it cannot be undone.

- **Add layer**: Adds a new sub-layer at the top of the layer tree.
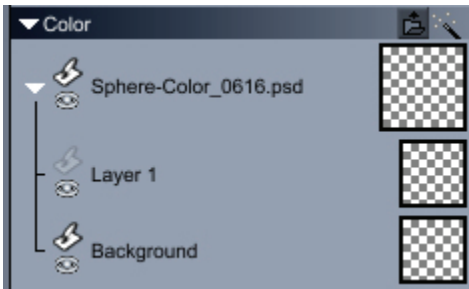
- Right-click on the Background or another layer to access the layer context menu.
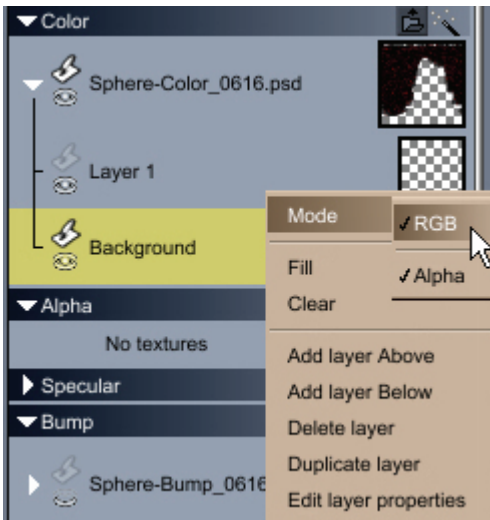
  - **Mode**: Shows the current RGB mode of the layer, either RGB or RGB with Alpha. If the layer does not need an alpha transparency, deselect the Alpha option.

  - **Fill**: When selected, fills the entire layer with the current brush color. You will not be asked to confirm the operation.

  - **Clear**: When selected, clears the layer. You will not be asked to confirm the operation.

  - **Add layer Above**: When selected, adds a layer above the current layer.

  - **Add layer Below**: When selected, adds a layer below the current layer.

  - **Delete layer**: When selected, deletes the current layer. You cannot delete the original Background layer and this option will be grayed out.

  - **Duplicate layer**: When selected, duplicates the current layer and places the new layer above it but with the same name.

  - **Edit layer properties**: When selected, brings up the **Layer Properties** dialog where the name of the layer can be changed.
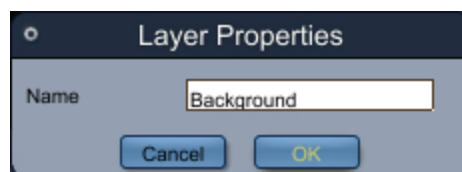
The image below shows a layer added before and after the background layer with the result shown in the second image. Note how the yellow in Layer 1 overlays the Background layer while the blue in Layer 2 underlays it.
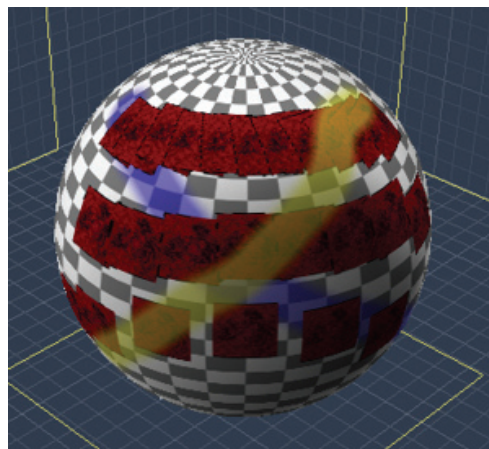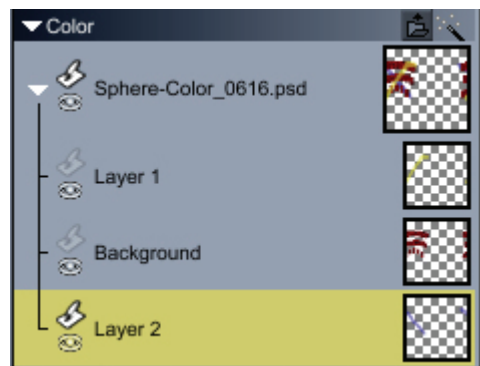
Layer Options

- **Active Layer**: An active layer will be highlighted in yellow and the 3D Paint icon on the left will be solid.

- **Inactive Layer**: An inactive layer will be the default color (blue) and the 3D Paint icon will be grayed out.
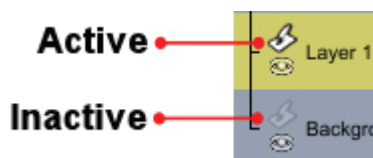
- **Hide/Unhide**: To hide a layer, left-click on the eye icon under the 3D Paint icon for the layer. The "eye" will close and any textures applied to the layer will be hidden in the viewport. Left-click on the eye icon again to unhide the layer.
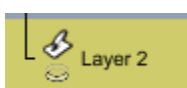
- When a layer is added, the **Layer section** activates and the **Mode** and **Opacity** can be modified. When the Background layer is selected, these options are grayed out.

- **Mode**: Provides options for setting the layer mode. These are similar to those found in paint programs and work similarly to them.
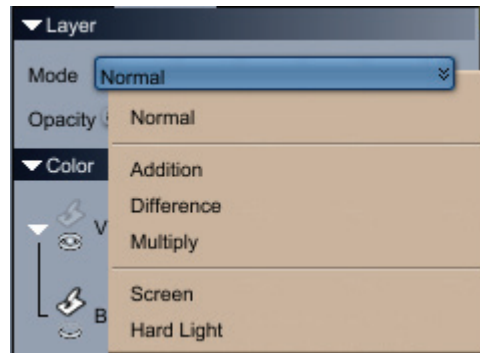
- **Opacity**: Set the layer opacity by dragging left or right on the slider, or by left-clicking in the value field and entering the percentage manually.

Animating

This chapter explains general concepts about the motion of Carrara objects:

In this chapter:

- For a list of the steps involved in creating an animation, see "What is Animation?."

- For useful animation principles to keep in mind as you create your animation see "Animation Principles."

- Refer to "Key Frame Animation" for an overview of using key frames to create animation in Carrara.

In addition, this chapter introduces the main animation tools in Carrara and briefly describes how to use those tools. The main animation features are:

- "Animation Features of the Sequencer Tray."

- "Storyboarding."

- Finally, there is a brief overview of animation features found in other chapters:

- Refer to "Additional Animation Features."

- For specific information on applying animation features, see "Animation Techniques."

- For specific information on character animation, see "Character Animation." (Standard, Pro)

What is Animation?

When you animate with Carrara, you add a fourth dimension:time:to your 3D illustrations. To create an animation, you change the arrangement of your scene at different key points in time. **Carrara** then fills in the gaps to complete the animation.

You can animate almost everything in your scene. For example, you can animate:

- the motion of objects, lights, and cameras
- object size, shape, and shading attributes
- object modifier settings
- camera and light parameters
- ambient lighting, background, backdrop, and atmospheric effects.

Animatable Properties

Each characteristic of an object or effect that can be animated is called a **property**. A typical object or effect has many animatable properties. Some properties, like those that define position and orientation, are common to all objects. Other properties are specific to certain types of objects.

- Lights have special properties like brightness, falloff and gels.
- Scene effects, like backgrounds, backdrops, and atmospheres, have their own special properties.
- The properties of an individual object depend on several factors, including what type of object it is, and what types of components are used in its shader.
- Environmental primitives (like Fountain, Cloud and Fire) have special properties that determine their shape.
- When you apply a modifier to an object, you add special animatable properties.

Steps to Creating an Animation

Animation is an extension of creating an illustration. For example, to create a scene you set up scene properties. To create an animation, you change those properties over time. When you play the animation, the changing properties create action.

In Carrara, you create an animation by moving to different points in time and making changes in the scene to define the state of the action at that moment. These changes are called **key frames**. Carrara automatically fills in the transitions between key frames to create the illusion of motion. This can be done through the Storyboard room or through the animation features in the Sequencer tray, which are described in this chapter, or by using one of the motion methods introduced in "Animation Techniques."

To create an animation:

1. Create objects using the basic geometric primitives and the modelers.
2. Build the scene by arranging objects and setting lights and cameras.
3. Animate the scene by making changes to the properties of the elements in your scene at different points in time.
4. Render the final animation.

Animation Principles

The following sections include some helpful animation principles and concepts to help you use Carrara. This is by no means an exhaustive explanation of computer animation, but some issues you should take into account when you create your animation.

Simplifying Your Scenes

In general, the 3D scenes you create for animation need not be as complex as a typical illustration scene. The viewer's eye tends to be drawn toward motion and foreground elements. Static objects and background elements are scanned only casually. .

By reducing unnecessary detail, you can dramatically reduce rendering times and keep the size of your scene files manageable. When preparing a scene for animation, keep the following guidelines in mind:

- Refine your animation. Keep the objects as simple as possible. Detail is usually lost in an animation.

- Use fewer objects, and limit the number of reflective and transparent objects.

- Use the minimum number of lights required to achieve an effect. Additional light slow down the rendering.

- If a complex model remains in the background for the duration of the animation, try substituting a simpler version.

- Avoid using a modeled background. Use an image or a procedural one instead.

- If your camera view remains unchanged for an entire scene, consider rendering a still image with just the scene's background elements. Then use this image as a backdrop and animate only the foreground elements. This technique requires some planning to make sure that shadows and transparent objects don't give the trick away.

Motion and Timing Principles

Motion and Timing Principles

As an animator, the most important skill you can master is the ability to portray motion convincingly, whether it is intended to be realistic or exaggerated and cartoonish. No matter how good a 3D modeler you are, the timing of the events in your animation are of paramount importance.

Many of the principles of timing and motion developed by cel animators apply to 3D animation with **Carrara** as well. Many of these principles apply especially to character animation, but most are useful for any subject matter. For more information, you can refer to one of the many excellent books on cartoon animation. .

Squash and Stretch

Squash and stretch are animator's terms for the exaggerated redistribution of an object's mass as it moves or shifts positions. Squash and stretch portray the qualities of elasticity and weight in a character or an object.

Think of a bouncing rubber ball. As it falls it stretches; as soon as it hits the ground it is squashed. If the ball failed to change shape, the audience would interpret it as a solid, rigid mass.

You can accomplish squash and stretch in Carrara by animating an object's shape using the Vertex modeler, the Spline modeler, the Metaball modeler, or the Stretch modifier.

This moon's bouncing motion is exaggerated by deforming its shape so that it stretches as it descends and squashes when it hits the surface.
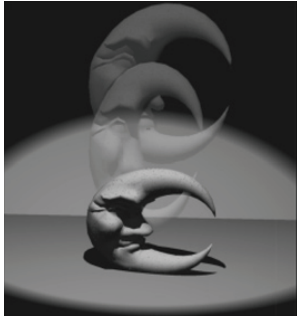
Lag and Overlap

When an object moves from one point to another, not everything has to move at once. For true-to-life movement, action that is secondary to the main activity can lag and overlap. For example, when you animate a character in pursuit of a bus, different parts of the body move at different speeds. The character's head may lead, followed by his torso, and finally by his arms and legs. In Carrara, the Inverse Kinematics modifier can be used to experiment with poses that convey this type of motion.

A flag waving back and forth is another example of lag and overlap. It lags at one end of the arc and overlaps when moving in the opposite direction. You can use the sweep path and the envelope in Carrara's Spline modeler to create lagging and overlapping effects.

Arc vs. Straight-Line Movement

Character motion appears more realistic if it follows an arc or curved path instead of a straight line. Most objects affected by gravity also follow curved, rather than straight trajectories.

Carrara's Bézier tweener is optimized for arc-type movements. Try experimenting with its Tighten In and Tighten Out settings to produce narrower or wider arcs.

Motion paths are also great for this effect. Refer to "Motion Paths" to learn more about creating and using motion paths. (Standard, Pro)

Secondary Motion

Secondary motion adds realism and credibility to a scene. A character turning his head to stare at something in disbelief shouldn't just turn his head; his jaw should drop and his eyes should blink as well. The viewer focuses on the main action, but registers the secondary motion as supporting it. Carrara's

Time Line gives you enough control to manage even the finest details of your animation, so you can add this kind of secondary detail.

Anticipation and Follow-through

In anticipation of a major action, an animated character often makes a small preliminary action in the opposite direction. For example, a character about to move screen right might first make a slight movement screen left, then strike a pose before moving screen right.
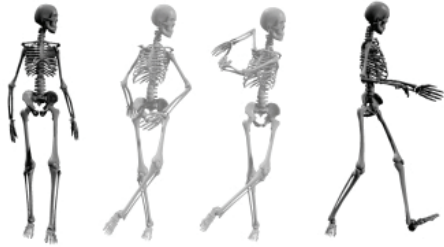
Follow-through is the continuance of motion after a major action is completed. A baseball bat does not stop moving after hitting the ball; rather, it continues along its arc of motion. Anticipation and follow-through make actions more believable.

Exaggeration

Exaggerating an action emphasizes it, making it more prominent. For example, if intrigue is called for, have a character sneak instead of walk.

If you want your animation to resemble footage from a hand-held camera, give your camera an exaggerated bobbing motion. Virtually any type of action can be exaggerated to get an idea across.

Timing

Timing is as important in animation as it is in any dramatic form. Consider the difference between an abrupt stop and a gradual slowdown. Each conveys a completely different impression. In general, a motion that continues at the same pace lacks interest and seems unreal.

If you are trying to animate realistic character action, act out the sequence yourself, timing how long each pose is held and how long each action takes with a stopwatch.

Timing is one of the most difficult aspects of animation to master. The key frames you define at different points on the Time Line must be synchronized with those that came before and those that follow. Fortunately, you can use the interactive nature of computer animation to fine-tune your timing. Test your animation frequently by previewing it in the 3D view or by dragging the Current Time bar back and forth between key frames. Adjust the event's position until you're satisfied with the timing.

Compression

If you are not pressed for hard disk space, it usually makes sense to render your animation without compression. This ensures that you'll have a high quality copy of the animation to work with.

Working from your uncompressed original, you can save copies, experimenting with various compression settings until you are satisfied with both the image quality and playback rate.

An animation compressed multiple times degrades significantly, so you should render without compression if you intend to do any post processing in another application.

See "Rendering" for information on compression.

Key Frame Animation

In Carrara, you animate objects by moving to different points in time, then making changes in the scene to define the state of the action at that time. These states are called key frames.

Key frames are tied to real-world time increments, seconds and minutes. You can determine how many key frames you want to run for your animation (from 8 to 60 frames per second) so that you can run a fast draft, or create more seamless movement. This means you can create key frames at specific points in time without worrying about the number of frames in your final rendered animation.

You can edit a series of key frames in the Storyboard room or using the Time Line in the Sequencer tray from any other room.

Multiple key frames in the Storyboard room are useful when you want to see the effect of changes on objects over several frames.

Viewing a single frame in the Assemble room or a Modeling room is more useful for doing detailed work on individual frames as you animate.

Carrara provides additional methods for creating key frames. Refer to "Animation Techniques" to learn about the other methods.

Tweeners

Carrara uses formulas called **tweeners** (from "be **tween"**) to calculate the state of each object between key frames. By specifying which tweener to use for each transition, you can control the rate of change between key frames in your animation.

Carrara includes many different types of tweeners: linear (for a constant rate of change), Bézier (for smooth motion paths and greater control over acceleration and deceleration), discrete (for instantaneous change), and oscillate (for alternating back and forth between key frames).

Animation Features of the Sequencer Tray

All of the grouping, linking, and organization tools you use in the Sequencer tray to produce a 3D illustration are the same tools you use to produce an animation.

In addition, the Sequencer tray also contains most of the primary tools you need to create an animation. It contains:

- The Hierarchy list
- Frame Rate Selector
- Time Information
- Render Range
- The Scrubber
- Key Frame and Tweener Markers
- Animation Preview Controls

The Sequencer tray is available in the Assemble, Model, Storyboard, and Texture rooms. Refer to "[Using the Sequencer Tray](#)" for general information on using the Sequencer tray. This chapter focuses on the areas in the Sequencer tray that are specific to animation.

Hierarchy List

The Hierarchy list of the Sequencer tray displays a textual representation of the scene's hierarchical structure that includes objects (such as cubes or spheres), cameras, lights, links, groups.

The Hierarchy is non-animatable. This means that you can't add or remove objects at specific points in time. Nor can you change groups or links over time.

In other words, the values of the properties on the Sequencer may change over time, but **the properties themselves must be constant**. For example, a spot light can't become a distant light over time; a bitmap background can't become a bi-gradient.

To use the Hierarchy:

- Click an object's name to select it.

- Click an object's name and drag it onto another object name to link it to that object. Unlink by dragging it to any other level in the Hierarchy.

- Select a child object by expanding its parent object in the list, then clicking the child object.

- Click on Time Line expansion triangles to reveal individual Time Lines.

Collapsing and Expanding the Hierarchy

The Hierarchy list can be expanded to show the animatable properties of each object contained in it. This allows you to animate each property individually.

It can also be collapsed to show fewer properties.

To expand or collapse the Hierarchy:

- Click the triangle to the left of the property.

- Click the plus (+) or minus (-) sign to the left of the child property.
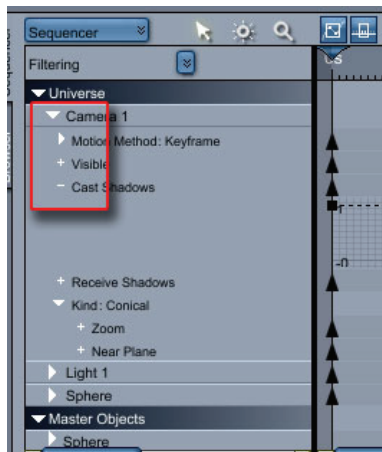
Creating Groups

Groups of tracks can be grouped together which can then be collapsed or expanded. This helps to optimize the screen space so you can focus on your animation.

To create a group:

1. Select the tracks to be grouped together.

2. Right-click on the hierarchy listing and choose **Group** from the menu or press **Ctrl+D**.

The group will be created with the selected tracks as part of the group.

3. Add additional tracks as needed by selecting the track and dragging it on to the group.

4. Remove tracks by selecting the track and dragging it out of the group.

To ungroup a group:

1. Select the group.

2. Right-click on the group and choose **Ungroup** from the menu or press **Ctrl+U**. The group will be removed and the tracks that made up the group will be placed back into the hierarchy as before.

Displaying Specific Time Lines in the Hierarchy List

Large scenes may have more time lines in the Hierarchy than you want to work with at one time. To minimize the number of time lines displayed in the Sequencer, you can select buttons on the Sequencer tray to show only certain categories:
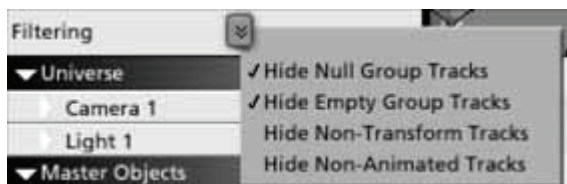
Select this button in the Sequencer to specify which properties are displayed.

- **Hide Non-Animated Tracks** sets the Sequencer to display only animatable properties that have key frames. (This excludes animatable properties that are constant throughout the animation.)

- **Hide Non-Transform Tracks** limits the Sequencer to display only motion properties. (This excludes animatable properties that are constant throughout the animation.)

- **Hide Null Group Tracks** sets the Sequencer to exclude animatable properties that have not been applied to the object.

- **Hide Empty Group Tracks** sets the Sequencer to exclude properties that can't be animated.

Frame Rate Selector

The frame rate selector lets you choose between 1 and 60 frames per second (fps). Typical frame rates are 15 or 18 fps for multimedia (CD-ROMs), 24 fps for film, and 30 fps for NTSC video. You'll probably want to use a lower frame rate for previewing, then increase the rate for final rendering.

The Frame rate you select will appear on the Time Line axis.

To change the frames per second:

1. Click the **Sequencer tray: fps button (frames per second).**
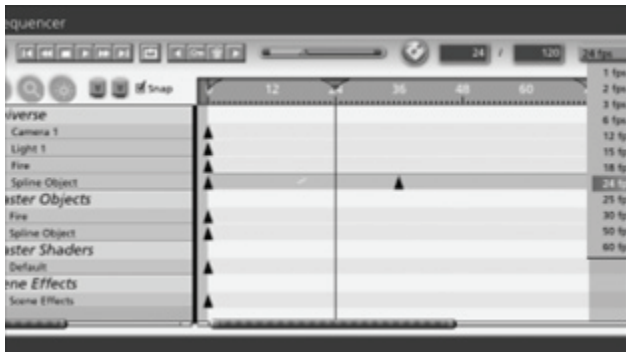
2. Select frames per second from 1 to 60.

Time Information

The Time Line is located to the right of the Hierarchy in the Sequencer tray, and provides a linear visual representation of the key frames that make up an animation. It allows you to manipulate key frames and move them to different points in time using points on the line.

The Time Line displays a time track for each item (object, effect, or property) currently shown in the Hierarchy. Key frame markers on these tracks represent key frames in the animation. For more information on key frames, refer to "Creating Explicit Key Frames."

Allocated Time

The allocated time is roughly the entire length of the animation you're working on. The non-allocated time is displayed as a dark shaded area on the right side of the sequencer. When you move the Current Time bar past the end of the allocated time, it is automatically adjusted. If you then need to set the allocated time to a smaller value, you can use the Allocated Time control.

Allocated Time control

Allocated Time
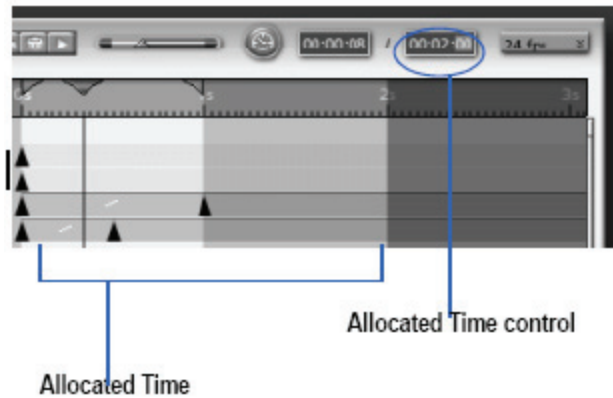
Elements of the Time Line

A key frame marker on a particular time track indicates a key frame that modifies the associated object, effect, or property.

You can create, copy, delete, or move key frame markers along the Time Ruler by dragging them.

The Time Ruler

The Time Ruler extends across the top of the Time Line. The ruler marks indicate time increments (minutes, seconds, and frames).

The Current Time Bar

The Current Time bar or Current Frame bar is a slider with a triangular handle that points to the current time or frame on the time/frame axis. You can switch between working with time or working with frames.

As you slide the bar along the time axis, the time or frame updates in the time scale/frame rate indicator.
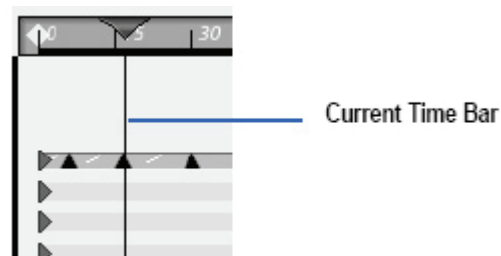
Current Time Bar

The Current Time Bar in the Time Line window indicates the current time in the Time Ruler.

To change the current time or frame with the time bar:

- Drag the Current Time bar to a different point along the Time Ruler. The time or frame is reflected in the Sequencer.

Time Scale and Frame Rate

The time scale is the amount of time between each mark on the Time Ruler. You can change the scale of the axis to see more or less time displayed.

The frame rate indicates the number of frames displayed in each second, when previewing and rendering.

Carrara has a display time/display frames toggle button to swap the time ruler between the two types of information. The frame rate also determines the number of tics (or marks) between major time divisions on the Time Ruler.
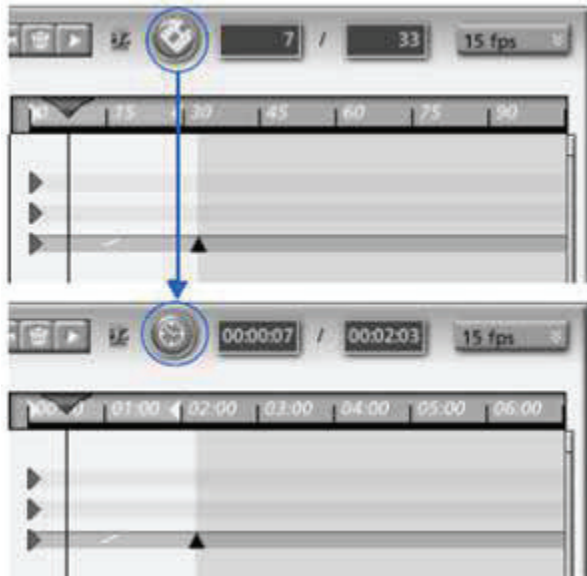
Snap

When Snap is turned on, the Current Time bar and key frame markers will snap to individual frame boundaries when you drag them.

To control snapping to frame boundaries:

- Click the Snap checkbox at the top of the Sequencer tray.
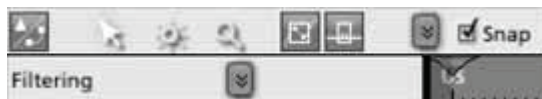
The Render Range

The **render range** depicts the portion of the animation to be previewed or rendered. It is represented by the non-shaded area of the Time Line. Two yellow arrows in the Time Ruler also indicate the beginning and the end of the render range.
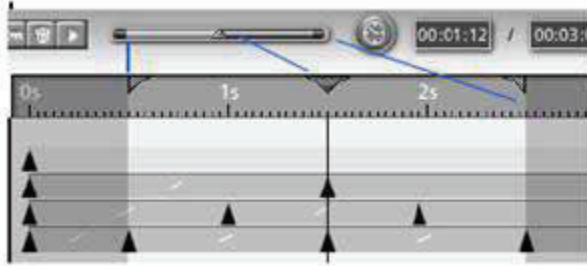
The render range must extend past all key frames if you want to preview or render the entire animation. The animation stops at the right boundary of the render range. If your animation does not preview or render to the end, verify that the render range extends beyond the last key frame.

In theory, the maximum render range is one frame short of one hour (the entire length of the Time Ruler). In practice, an animation that long would exceed the capacity of most hardware configurations. Lengthy animations should be produced in several short clips, which can then be edited together in a video-editing application.

To change the render range start and end times:

1. Drag the right edge of the render range to set the end time.

2. Drag the left edge of the render range to set the start time of the animation.

Scrubber

The Scrubber is the bar above the Time Ruler. It shows the current time, and 100% of the render range.

To use the scrubber:

*   Move the Current Time marker to preview the animation.

Key Frames and Tweeners in the Sequencer Tray

Key frames are stored animation values along a track. The Time Line displays a time track for each element (group, object, effect, property category, or individual property) currently shown in the Sequencer. Key frames and tweeners are indicated with markers on the Time Line for each object in the scene.

Key Frames and Tweener Markers on the Time Line

A key frame marker on the tracks of the Time Line may represent a single key frame or several coincident key frames, each relating to different properties of the same object (the object on the Sequencer to the left of the Time Line).

Tweeners appear between key frames. The location of a key frame marker along its track indicates the time at which the key frame occurs.

Key Frames and the Hierarchy

A key frame modifies a single property. However, whenever two or more coincident key frames modify properties of the same object or effect, they are represented on higher levels by a single key frame marker. So, when you move, copy, or delete a key frame marker that represents several coincident key frames, all of the key frames it represents are affected.

To manipulate the key frames separately, expand the Hierarchy to the individual property level, where the two key frames are represented by different key frame markers on separate tracks.

Following is a key frame marker on a collapsed Hierarchy.
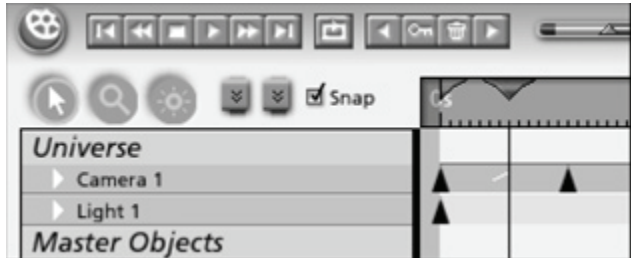
When you expand the Hierarchy, you see that there are actually several individual key frames represented.
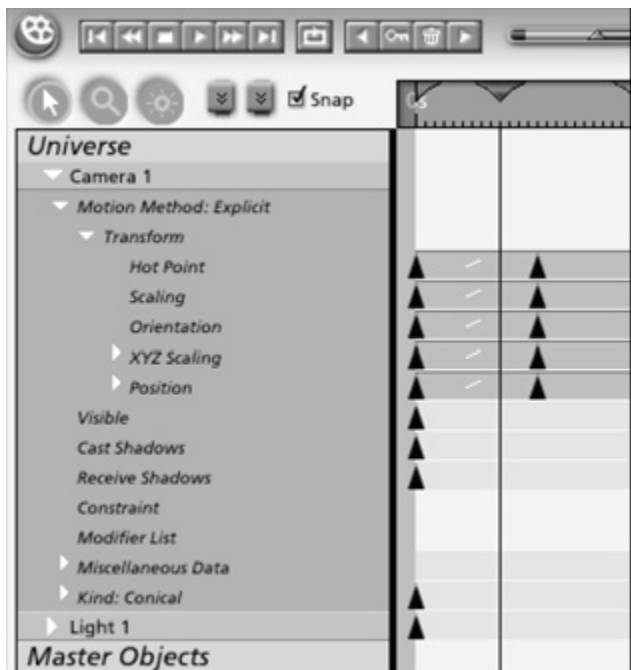
Overlapping Key Frame Markers

Sometimes two or more key frame markers appear on the same time track at nearly, but not exactly the same place (they may represent key frames modifying the same property, or key frames modifying different properties of the same object or effect.)

Depending on the current time scale (how closely you have zoomed in on the view of the Time Line area), these markers may overlap.
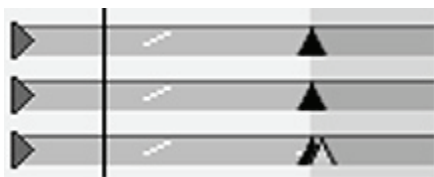
This is what two overlapping key frame markers look like on the Time Line.

Zooming in usually makes it easier to distinguish between overlapping key frames. You can zoom in with the Zoom tool.

To zoom in and out in the Sequencer, click the Zoom tool in the Sequencer tray and click the location where you want to zoom in.

To zoom out of the Sequencer, hold Alt, then click while the Zoom tool is selected.

Animation Preview Controls

The animation preview controls consists of VCR-like buttons for previewing and moving through your animation within the Assemble room, moving forward or back one frame, and moving to the beginning or end of the render range.

The render range is the yellow area of the Time Ruler, which determines the starting and ending points for previewing or rendering an animation.

You can use the animation preview controls to preview your animation at the current display quality. Textured preview quality will slow down animations.

Following are the buttons used with the animation preview controls.

	Rewind	Jump directly to the first frame of the render range.
	Previous Frame	Step back one frame.
	Stop	Halt play.
	Play	Preview the animation forward from the current time.
	Next Frame	Advance one frame.
	Fast Forward	Jump directly to the last frame of the render range.
	Loop	Loop when you play. Click again to disable looping.

Animation Preview Tips

When you preview an animation in the Assemble room, Carrara does not skip any frames. If it can't draw frames fast enough to maintain the current frame rate, the preview takes longer to play.

Reducing the frame rate produces choppier motion, but the preview will play closer to real time. Lowering the current display quality to wireframe or box will speed up the preview without sacrificing smoothness of motion.

In some cases, you may want to render a low-resolution animation for preview purposes, rather than preview the animation in the Assemble room.

Storyboarding

A **storyboard** is a series of drawn images showing the key actions in an animation. The storyboard helps you quickly work out the viewpoint, framing, and composition of the animation.

Because several frames of your animation are visible in sequence, it is easy to make changes. Any change you make in a frame of your animation causes the Storyboard to update the entire animation.

Features of the Storyboard Room

You can access the Storyboard room from any other room by clicking the Storyboard room button to modify an existing scene in the Storyboard room, or to start a new scene file using this room.
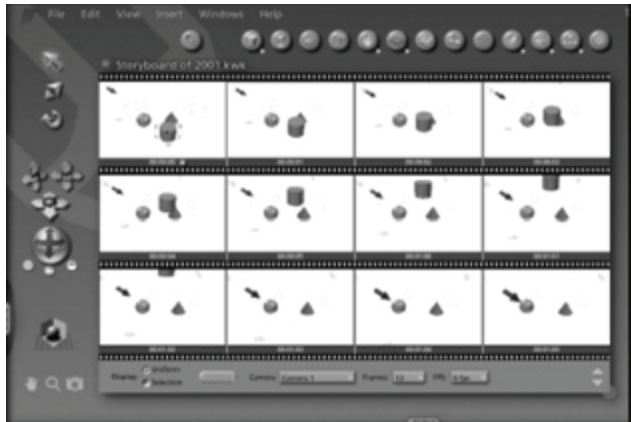
The image initially appears identically in all of the storyboard frames, with the first frame as a key frame.

To make changes to a frame, you select it and use editing or animation tools. If you wish to use features that are available in the Assemble or Modeling rooms, you can move to those rooms with the selected frame as the active image. You can then edit the frame, then return to the Storyboard room to see the effects of your changes on the animation.

In addition to the features of the Sequencer tray, you can press Update to update the key frames shown in the storyboard or press Up or Down to scroll through the frames. When you press Update while Selection is selected, the key frames for the selected item in the Sequencer are updated in the Storyboard.

Working with Key Frames and Tweeners

When you first switch to the Storyboard room with a new document, the objects in the scene appear the same in all of the frames. Your first frame is automatically a key frame. As you modify frames on the Storyboard, those frames also become key frames. All of the other frames are interpolated.

Carrara identifies each frame containing key frames with a dot next to the time/frame number so you can tell them from the other frames.
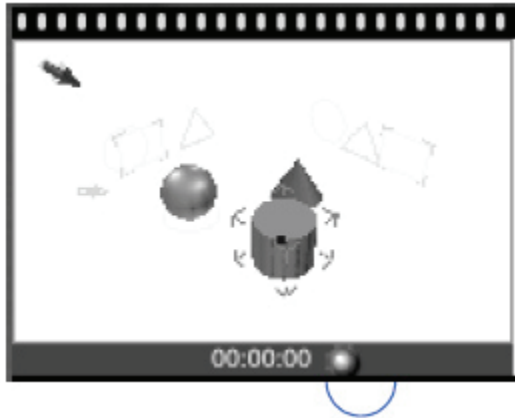
The number at the bottom of the frame, to the left of the key frame indicator, shows the time position of this frame in relationship to the frames per second selected. (For example, if you have selected 6 frames per second, your first frame will say 00, your second frame 01, your third 02 up to your 7th frame, which will say 1.00 to indicate the completion of a second of time).

To work on key frames:

- Select Uniform to show frames with equal increments of time according to the storyboard frames per second indicator.

- Select Selection to show only the key frames of the selection.

Changing the Number of Key Frames

You can view from 8 to 28 frames at any one time.

To change the number of visible key frames:

1. Select Storyboard room.
2. Click the Frames button at the bottom of the Storyboard window.
3. Select a number of frames from the list.
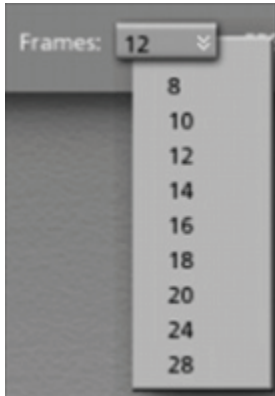
If you reduce the number of frames (for example, if you have been working with 20 frames and decide to only work with 10) only the first 10 of the 20 appear. Use the Up and Down button to scroll the storyboard and view other frames.

If you expand the number of frames (for example, if you have been working with 8 and decide to work with 28), the additional frames are appended to your original frames.

Changing Frames per Second Displayed

You can adjust the frames per second for the storyboard, from 1 to 60 fps. See "Time Scale and Frame Rate."

To change the frames per second:

1. Click the fps button (frames per second) located at the bottom of the Storyboard window.
2. Select a frames per second value from 1 to 60.

Changing the Camera

You can change the camera for all of the frames in the Storyboard room.

To change the camera in the Storyboard room:

- Select **the Storyboard room: Toolbar: Camera**. All of the frames change to the selected camera.

Creating Explicit Key Frames

In Carrara, key frames are tied to real-world time increments:minutes and seconds:rather than to individual animation frames. This means you can create key frames at specific points in time without worrying about the number of frames in your final rendered animation.

The following image shows what Key Frame indicators look like in the Storyboard room and Sequencer Tray
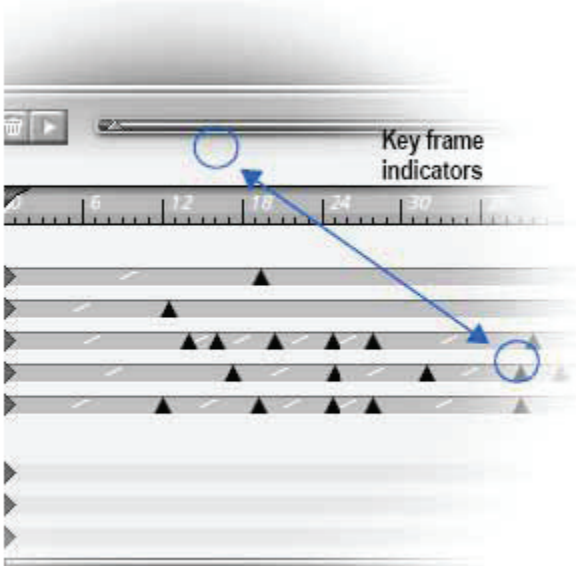
You might work at a low **frame rate** (6 frames per second is the default) to preview your animation on screen. Then increase the frame rate to render your final animation. .

Once you have defined a key frame, you can remove it, copy it, or move it along the Time Line to fine-tune the timing of your animation.

Adding Key Frames

Key frames are changes to the properties of objects and effects in your scene at specific points in time. You define key frames to set the changes that create your animation.

After creating key frames, you may want to modify the rule (tweener) Carrara uses to calculate the transitional states between keys. For more information, see "Applying the Tweener".

Typically, you create a key frame at the beginning and end of a particular action.

To add key frames:

1. Select an object (or effect) in the Assemble room or a frame in the Storyboard room.

2. Either click the object, or open the Sequencer tray and click the Time Line for that object.

3. Drag the frame bar to the time position on the time axis where you want to add a key frame on that Time Line.

4. Use the standard Carrara tools, editors, trays, and dialogs to modify the properties of the selected object. Carrara places a key frame marker on the appropriate track in the Time Line for that object.

5. Continue creating key frames at various points along the Time Axis.

Carrara automatically calculates the transitional states between key frames using tweeners. The transitions before and after the new key frame use the tweener options established for the previous key frame.

Editing the Time Line

After you've created several key frames, you may want to change the timing of your entire animation, or synchronize individual events within the animation.

You can edit the Time Line to adjust the timing and content of your animation. You can add key frame markers, delete them, or move them along the Time Axis.

Editing Key Frames

You can delete key frames, change their timing, duplicate them while dragging, stretch a series over time or copy them. You can also use a series of key frame actions, such as snap, reverse, repeat, and mirror.

Deleting Key Frame Markers

To delete a key frame marker, you select a key frame marker then select **Edit > Delete**.

To delete a key frame marker:

1. Open the Sequencer tray.

2. From the Time Line, select one or more key frame markers.

3. Click the desired key frame marker.

    *   Hold down the Shift key and click any additional markers you want to select.

    *   You may also drag a marquee around one or several markers to select them.

4. Choose **Edit > Delete,** or press Delete. Carrara recalculates the tweeners.

Changing the Timing of Key Frames

To change the timing of key frames, you move key frame markers horizontally along the time tracks to different points along the Time Axis.
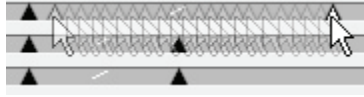
To move one or more selected key frames:

- Drag the key frame marker along the Time Axis to the desired point. If you've selected several markers, drag one of them and they all will move. The time intervals between the selected key frames remain the same.

- Carrara lets you move a key frame marker past neighboring key frame markers on the same time track.

- To restrict a key frame marker to stay between the previous and next key frame markers, hold down the Shift key as you drag the key frame marker.

Duplicating Key Frames

You can duplicate a key frame and move it to another location on the same time track by holding down a modifier key while you drag a key frame marker. You may use this to return an object or effect to the same state at any point during an animation.

To duplicate key frames:

1. Open the Sequencer tray.

2. Hold down the Option/Alt key, and drag one or more key frame markers along their time track(s) on the Time Line.

Nudging Key Frame Markers

You can move key frame markers in increments of one pixel each.

To nudge selected key frame markers:

1. Open the Sequencer tray.

2. Select one or more key frame markers on the Time Line.

3. Press the left or right arrow key to move the marker one screen pixel to the left or right. You can also hold the Shift key down and press the arrow keys to move one frame at a time.

When you duplicate or Copy and Paste an object, you duplicate all of its animatable properties. Likewise, when you copy and paste objects into other scenes, they carry their key frames.

Stretching a Series of Key Frames over Time

You can stretch a series of key frames over time, while maintaining the proportional timing between the events.

This is useful when you're satisfied with the relative animation of a particular sequence, but you want the entire sequence to speed up or slow down.

To compress or expand the timing between a series of key frames:

Stretch or compress a series of key frames by dragging the key frame markers.

1. Open the Sequencer tray.

2. Select three or more key frame markers on the Time Line.

3. Hold down the Command/Ctrl key and drag one of the selected key frame markers.

   The entire series of key frames will be stretched or compressed, retaining the proportional timing between the key frames.

Key Frame Actions

Carrara provides some special features for duplicating and modifying a series of key frames. You can snap, reverse, repeat, or mirror a key frame sequence. You can also adjust the positioning of the events so that they coincide with the start of the next frame.
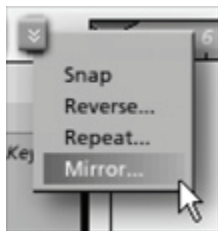
To use a Key Frame Action:

1. Open the Sequencer tray.

2. On the Time Line, adjust the display to show the desired sequence of key frames. You may need to change the time scale or expand the properties of an object.

3. Select each marker in the sequence. You may drag a marquee or press Shift+click.

4. Select the desired action from the action selections.
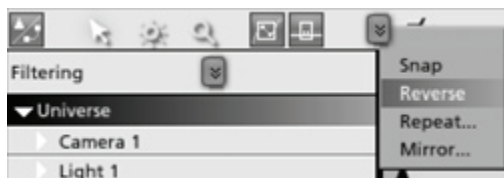
- **Snap** adjusts the positioning of the selected events so that they coincide with the start of the next frame.

- **Reverse** flips the order of the events so that the last becomes first and vice versa.

- **Repeat** duplicates the sequence and places the duplicate after the selected sequence.

- **Mirror** duplicates the sequence, reverses the duplicate, and places it after the selected sequence.
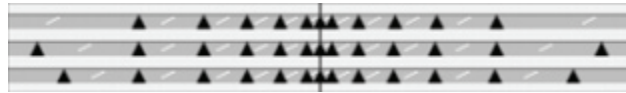
Using Tweeners

You can use tweeners to create interesting and subtle changes in the transitions between key frames. Tweeners save you time by automatically creating movements and changes that would be extremely difficult with key frames alone. Tweeners make your animation motions, and other changes, more natural and sophisticated.

What is a Tweener?

The time period between any two key frames is considered a transition. In the Time Line, you see the transition as the "gap" between key frame markers. Carrara uses a formula (the "tweener") to create the transitional states so that the object or effect changes smoothly between the two key frames.

You won't always want a smooth, linear transition. Sometimes you want an abrupt change of state. Other times you want a transition that starts slowly and accelerates to finish in a rush. You might even want a transition that quickly alternates between the before and after states, like a fluorescent light flickering on.

All of these transitions are possible by setting your options for the tweener used between each pair of key frames.

A tweener requires a beginning and an ending key frame, although you can place the ending key frame far out in time.

Applying the Tweener

Carrara automatically applies a tweener in every "gap" between key frames on the Time Line. The default tweener is linear, which produces an even rate of change between the two states.

You can achieve a different sort of transition by applying a different tweener.
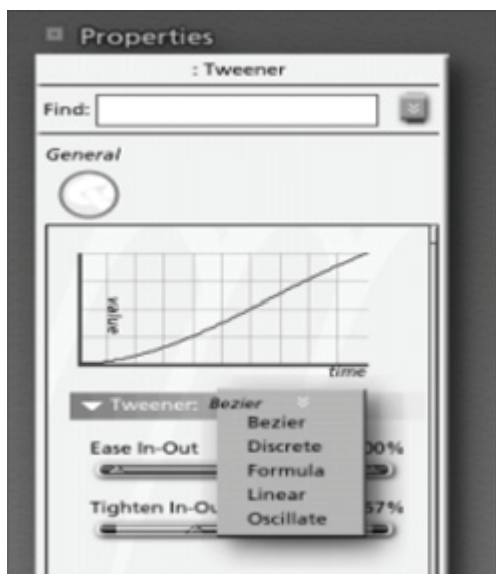
To apply a tweener:

1. Open the Sequencer tray.

2. On the Time Line, set up beginning and ending key frames.

3. Click the time track segment between the two key frame markers.

4. Select a tweener type from the list.

5. Set the appropriate controls for the tweener. The controls for each type of tweener are described in the sections that follow. As you move the handles on a slider, the graph reflects the new settings.

   Notice the small graphic on each segment of the time track. This graphic indicates the kind of tweener that is applied.

Bézier Tweener

The Bézier tweener creates smooth motion paths. For best results, use it on an object's transformation track.

The Bézier tweener has two controls that you can adjust:Ease-In/Out and Tighten-In/Out.
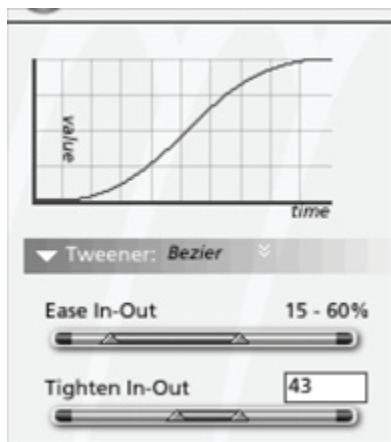
To set Bézier tweener options:

1. Open the Sequencer Tray.

2. On the Time Line, click the time track segment between the two key frame markers. The Transition Options display in the Properties tray.

3. Select a Bézier from the list. The Bézier tweener controls appear. The two handles on the Ease-In/Ease-Out slider on the Linear tweener to modify the rate of change at the beginning and end of the transition.

4. Drag the left handle to the right to start slowly.

5. Drag the right handle to the left to finish slowly. The Tighten In/Out slider allows you to adjust the trajectory of the transition:the "path" the values take as they change from one key frame to the next.

6. Drag the left handle to change the initial trajectory.

7. Drag the right handle to change the finishing trajectory. Values near the extremes (0% and 100%) result in tighter trajectories (closer to linear). Values near 50% result in looser (less direct) trajectories. The default settings (33% in and 67% out) are optimized for natural, smooth motion.
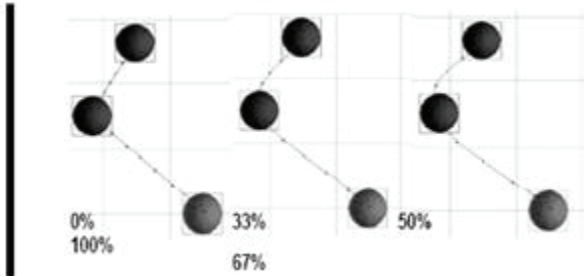
Discrete Tweener

The Discrete tweener maintains the values from the key frame that begins the transition until a specific time in the transition is reached. Then the values abruptly change to those of the key frame that ends the transition. Use the Discrete tweener when you want a property to change immediately, like switching on a light.

To set discrete tweener options:

1. Open the Sequencer Tray.

2. On the Time Line, click the time track segment between the two key frame markers. The Transition Options display in the Properties tray.

3. Select Discrete from the list.
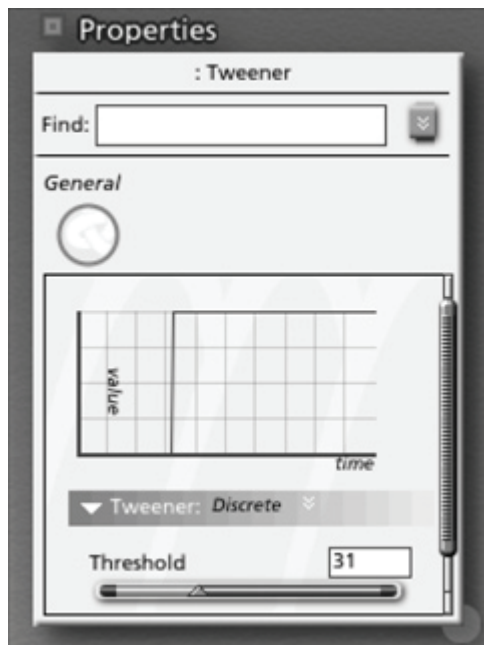
4. Adjust the Threshold slider to specify the exact point during the transition where you want the change to occur.

    The threshold value is a percentage of the transition's entire duration. For example, if you want the discrete change to occur halfway between the key frames, set the threshold to 50%. If you want the change to occur at the time of the first or last key frame, set the threshold to 0% or 100%, respectively.

Formula Tweener (Animation)

Formula creates a transition curve or pattern from a formula. You can devise your own formula to create a custom tweener.

To use a formula for the tweener:

1. Open the Sequencer tray.

2. From the Time Line, click the time track segment between the two key frame markers. The Transition Options display in the Properties tray.

3. Select a Formula from the list. The Formula controls appear.
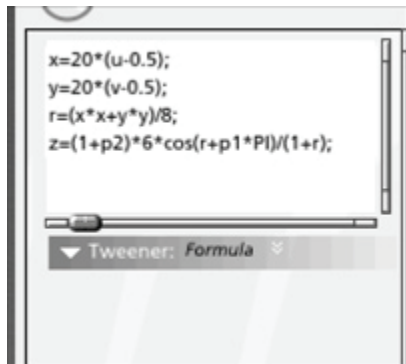
4. Set the Formula controls to create a tweener of the shape you want. Enter a formula into the editing panel on the top, or edit the existing one.

    - **Parse** Click this button to test and compile the formula. If there are any errors, Carrara gives you an error report.

    - **Parameter** Use these sliders to increase and decrease the value of parameters.

    - **More** Click this button to display the Formula Editor.

Linear Tweener

The Linear tweener creates gradual, direct transitions between actions.

When the graph line is straight, from the bottom left to the top right corner, the transition proceeds at an even rate from start to finish. This is the default, but you can change it.

To set Linear tweener options:

1. Open the Sequencer tray.

2. On the Time Line, click the time track segment between the two key frame markers. The Transition Options display in the Properties tray.

3. Select Linear from the list.

4. Adjust the two handles on the Ease-In/Ease-Out slider on the Linear tweener to modify the rate of change at the beginning and end of the transition.

- Drag the left handle to the right to start slowly.
- Drag the right handle to the left to finish slowly.
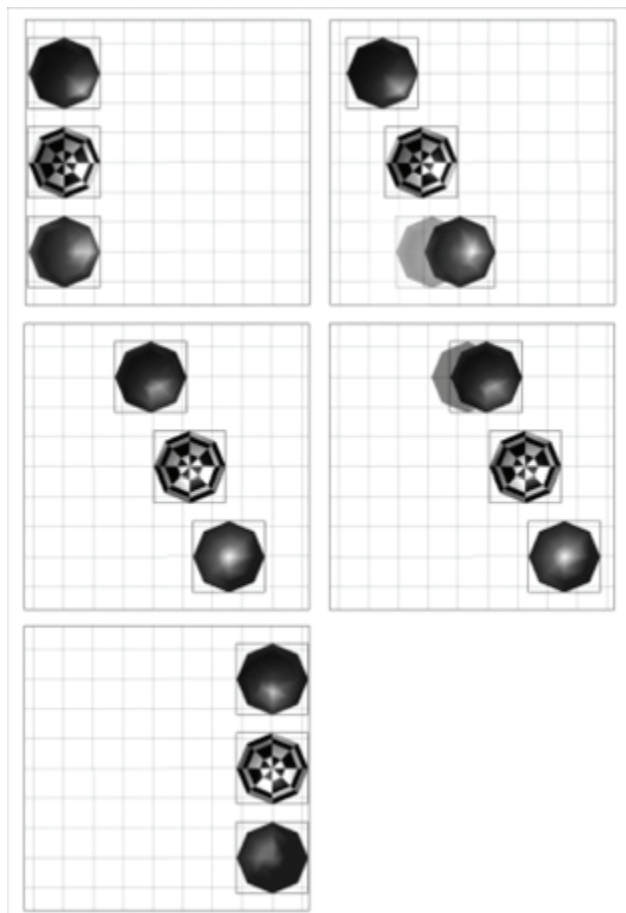
Noise Tweener

Noise creates a transition based on a random value, also referred to as noise.
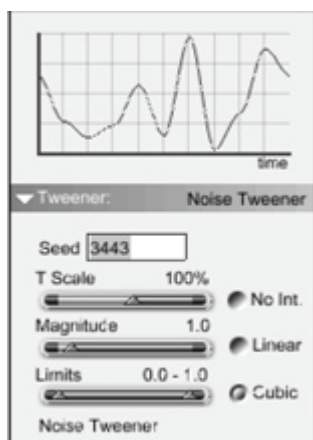
- **Seed** This value "seeds" the random number generator.

- **T Scale** Changes the frequency.

- **Magnitude** Changes the amplitude.

- **Limits** Upper and Lower limits on the amplitude.

- **Interpolation Controls** Changes the type of interpolation. None will result in sharp (instantaneous) changes, linear will smooth the transition, but changes in direction are very sharp. Cubic provides smooth transitions.

Oscillate Tweener

The Oscillate tweener switches back and forth between the start and end values several times during the transition You can choose between several wave forms to describe the alternation.
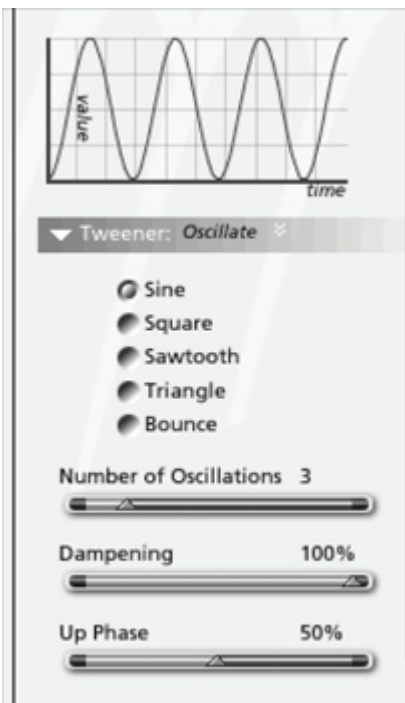
To set Oscillate tweener options:

1. Open the Sequencer tray.

2. From the Time Line, click the time track segment between the two key frame markers. The Transition Options display in the Properties tray.

3. Select Oscillate from the list.

4. Click one of the Wave Shape radio buttons to choose the desired wave shape. The graph of the wave shape shows how the switch occurs between up and down phases.

   - The Sine wave uses a sine curve to alternate the before and after values.

   - The Square wave alternates back and forth between the first and last key frames with no interpolation. One way to make a light blink at one second intervals is to turn the light on and off by setting key frames every second on the light's time track. A more efficient way is to turn a light on at Event A and then turn it off, perhaps ten seconds later, at Event B. Afterwards, apply the Oscillate tweener, and specify ten oscillations (using the Square

wave so that the switches are abrupt). The Oscillate tweener switches the light on and off ten times during the ten-second interval, using only two key frames and a single tweener.

- The Saw/Loop wave interpolates linearly from its beginning value to its ending value, then snaps back abruptly to the beginning value in each oscillation. You can use this option to replicate an action, such as hitting a gong. The character's arm is first placed in a drawn back position, moves to strike the gong, and then snaps back. You can also use Saw/Loop to play a movie from start to end in a loop.
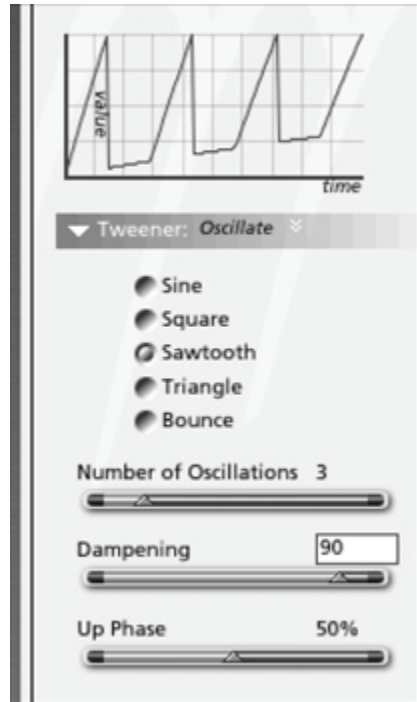
- The Triangular wave shape causes the transition to alternate back and forth abruptly.

- The Bounce wave shape causes the transition to alternate, with abrupt change in and out of the second position and gradual change in and out of the first position. This works well when the first position is a ball at height, and the second position is the ball on the floor.

5. Adjust the Nbr Oscillation slider to set the number of times the values should alternate.

6. Adjust the Damping slider to progressively diminish the amplitude of the oscillations. After each oscillation, the amplitude diminishes by the damping factor percentage.

7. Adjust the Up Phase slider to control how much of each oscillation is dedicated to the "up phase." For example, if you set the Up Phase value to 90%, the object spends 90% of the oscillation changing from A to B and only 10% changing from B to A.

Spline Tweener (Pro only)

The spline tweener is used in the graph editor where you can control the tangents. See "Graph Editor."

Graph Editor (Pro)

The graph editor gives you the ability to see, understand and edit your animation with a high level of control. You do not just see key frames but also their values and how they are interpolated during your animation.

Mini-graph Editor

In the timeline where you can edit the key frames, you can display a mini graph editor for a parameter. This mini-graph editor is only available for parameters that support it. This depends on each parameter type and also the parameter need to be animated (its timeline must contain more than one key frame).

To display it, click on the + icon just before the name of the parameter. It will open a small graph bellow the timeline where the interpolated curve will be visible.
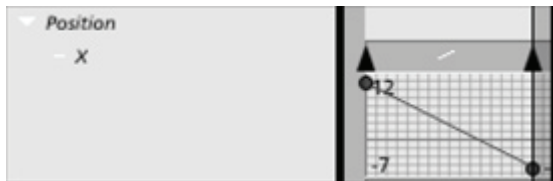
Using the Graph Editor

To switch back and forth between the graph editor and the timeline mode, use the following button:

The graph editor mode is like the mini-graph but here you can display multiple curves all together. You have more room to clearly see the details. Also, having multiple curves at the same time is useful if you want to adjust two different curves, one according the other, or to align two key frames for instance.

To display a curve in the graph editor

- Select the parameter that you want to see in the hierarchy on the left. If you want to select multiple parameters, you can select more than one while holding the shift key down.

- You can also select a group of parameters to see all children displayed in the editor. For instance, if you select the group Position, the three curves for X, Y and Z will be visible.

Editing Key Frames in the Graph Editor

To select and edit key frames, use the Pick tool. Just select the key frame and drag. Note that you can also move several key frames at the same time by selecting them while pressing the shift key at the same time or by using the marquee selection. You can also use the key modifiers (alt/option or control) to scale, duplicate, etc. For more information on this, see "Editing Key Frames."

Editing Tangents

The main purpose of the graph editor is to let you adjust the tangents that influence the interpolation between two key frames. This gives you precise control over the animation.

Tangent Presets

You can quickly setup the tangents using the predefined presets. First select the key frames (one or a multiple selection) that you want to modify and select one of the following buttons:

- The first one will create tangents to have a smooth interpolation.

- The second button means linear interpolation. This preset will remove the tangents.

- The last one creates flat tangents. This is a quick way to create tangents for a key frame and edit it manually afterward.

Half-tangents

The tangent for a key frame is composed of two half-tangents: the left and the right.

You can create or remove one of the half-tangents (left or right one) by clicking on one of the following buttons:

Half-tangents can be used in two modes:

- In linked mode, when you move, for instance, the control point of the left half-tangent, the right one will be modified as well to keep both half-tangents aligned.

- In unlinked mode, moving one control point will not modify the other one.

To change the tangent mode, you can use these buttons:

Editing Tangents Control Points

After creating tangents, select one control point using the Pick tool and drag to move it. This will modify the tangent according to the current mode (linked or unlinked).

If, while dragging, you hold the shift key, the direction of the tangent is constrained. You will only be able to modify the length of this tangent.

You can also press the alt/option key to switch between linked and unlink tangents mode while dragging the control point.

Notes on Tangents

- The tangents are only available if the tweener used is a Spline tweener. This means that if you use an Oscillate tweener you will not be able to have a tangent here. Also, using the tangent presets may change this tweener and replace it with a Spline.

- Some parameters do not support tangents. This is the case, for instance, for the rotation and the hot point, due to the fact that the way they are interpolated is not compatible with tangents. For this kind of parameter, you can only move the values in the graph editor and use another tweener to control the interpolation, like a Linear or Bézier tweener.

Additional Animation Features

Carrara also allows you to animate several other types of objects and effects, including the following:

- Modifiers (see "Modifiers").

- Shapes of spline, vertex, and metaball objects. For more information, refer to "Animating in Other Rooms."

- Characteristics of environmental primitives, to achieve effects such as flickering flames or rising fog.

You can also use animation in the following additional ways:

- Add rotoscoping to your scenes.

- Create animations using motion paths or the physical behaviors of objects.

- Use Physics to let your objects accelerate, move, and interact in ways consistent with the laws of physics. For more information, refer to "Physics." (Standard, Pro)

Animation Techniques

In "Animating," you learned about key frame animation and were introduced to the animation features in Carrara.

In this chapter:

This chapter discusses the following motion methods:

- Explicit (key frames)

- Motion paths

- Physics

In addition to these motion methods, you'll also learn about animating in other rooms.

See "Character Animation" to learn about animating using skeletons. (Standard, Pro)

All of the motion methods used by Carrara create key frames. However, in some cases you create them directly, and in other cases, Carrara calculates the movement for you. For example, you create key frames when you use the Storyboard room or the Sequencer. Yet, when you use physics, you can create key frames to set up the scene, but Carrara calculates the motion.

To choose your motion method:

1. Select the object or group you are animating.

2. Choose the **Properties tray: Motion tab: Motion panel:** and select a method from the motion method pop-up menu**.**
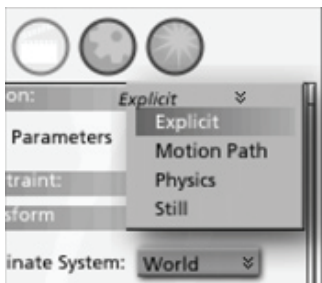
3. Select **Still** to create a non-moving object.

4. If a warning message appears telling you that you are changing motion methods, move the slider to determine the quality of the conversion.

Explicit Motion Method

In "Animating," you learned about key frames and the basic techniques for creating explicit key frames. The sections that follow introduce additional ways you can animate scenes using explicit key frames.

For more specific information on key frames, refer to "Key Frame Animation" and "Creating Explicit Key Frames."

Tip Although you can use different motion methods within a scene, you cannot use more than one motion method per object. What you can do is duplicate objects and use different motion methods on each copy. Then cloak and uncloak the objects to switch between them at the right moments.

Animating with Rotation

You generally animate complex motion by moving (translating) an object to various positions along the desired path of motion and creating key frames at each position. You can then apply the Bézier tweener to smooth the motion.

However, when animating along simple, curved paths, it is often easier to offset an object's hot point and animate its motion using rotation rather than translation. You might want to do this using a motion path.

An object always rotates around its hot point. By default, an object or group's hot point is at the center of its bounding box. However, you can move the hot point anywhere, even some distance from the object.

For example, you can point a camera at an object and move the camera's hot point to the center of the object. You could then rotate the camera around its own hot point to animate a "fly-around" of the object. This approach generally requires fewer key frames than creating a similar motion path in the usual way, and produces equal or better results.

For instructions on working with hot points, refer to "Moving the Hot Point."

Duplicating Relative Motion with Groups

Duplicating an object or effect also duplicates its animation data (its key frames and tweeners).

When key frames are duplicated in this fashion, the values of the original object's properties are copied exactly to the duplicate. In the case of motion, this does not always produce the desired results.

Suppose you have animated an object, and you decide to create a flock of identical objects, all moving in the same way. The following figure illustrates what happens when you duplicate your object and offset the duplicate from the original.

'Since the numerical coordinates of the original object at the second key frame were copied exactly to the duplicate object, the two objects end up in the same location.

You can avoid this problem by nesting the original object within a group before duplicating. When an object is animated within a group, its position is animated relative to the group, not relative to the global coordinate system. Therefore, when the group is duplicated and offset, the animated object in the duplicate group moves along its own motion path, within the group.

Rotoscoping

Rotoscoping lets you add live action or moving textures by incorporating existing animations or digitized videos (movies). You can use QuickTime or Microsoft Video AVI (Windows) files as texture maps or layers on objects, backgrounds, backdrops and light gels.

Rotoscoping is a general term that describes using video selectively inside another movie. You can add live action or moving textures to your scene by rotoscoping with animations or digitized videos (movies) in your animation. Rotoscoping adds realism and visual excitement to your animations.

You can use any movie file for rotoscoping. You can use movies in any of the following places:

- Texture map or layer on an object:including in non-color channels, like Bump.

- Background or backdrop.

- Light gel.

The movie is referenced as a separate file. Its path name is relative to the animation file in which it is used. When you move a scene file that uses rotoscoping, be sure to maintain the relative path to the movie file.

Rotoscoping is visible on objects in Textured mode, but is not displayed on backgrounds, backdrops and in gels until you render the animation.

Synchronizing Movie Frames with Animation and Looping

When you apply a movie, Carrara creates two key frames:one at the start and one at the end of the movie. You see the markers on the time track.
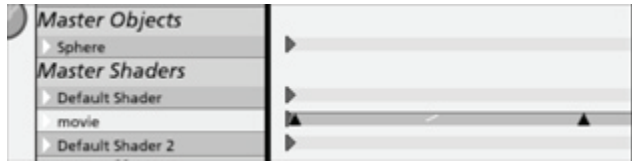
The ending event marker is placed according to the duration of the movie:not the number of frames it has. This means that the frame rate of the movie should be compatible with the rate of the animation.

It isn't necessary that the movie begin at the first frame (frame 1). You can start at a later frame by synchronizing the movie to the animation.

To synchronize a movie with the animation:

1. Set the current time bar to the moment (frame) where you want to synchronize the animation.

2. Select the object or light onto which the move is rotoscoped. (This is not necessary for a background or backdrop.)

3. Display the rotoscoped movie controls

   - For a texture map, display the Shader Tree Editor.

   - For a light, display **Properties tray: Effects tab: Gel panel**. Then choose Map from the pop-up menu.

   - For the background or backdrop, display the Properties tray, select the Scene from the Find list, then click the Effects tab and choose Background or Backdrop controls.

4. Use the animation preview controls to advance to the movie frame you want synchronized with the current frame of the animation.

To loop a movie:

1. Apply the movie at the start of the animation.

2. Drag the end-of-movie key frame marker out as far as you want the movie to continue.

3. Double-click the track between the two movie key frame markers.

4. Choose the Oscillate tweener.

5. Select the Saw/Loop Wave Shape.

6. Set the Up Phase slider to 100%. This rewinds the movie instantly after each cycle.

7. Adjust the Number of Oscillations slider to set the number of times the movie should repeat. Each up slope in the graph represents one time playing the movie.

   For precise work, you should calculate the correct number of oscillations based on the number of frames in the movie and the duration of the animation. You can adjust the end-of-movie marker, if necessary.

Motion Paths (Standard, Pro)

Motion paths let you create and edit an animation in a scene from points on a straight or curving line. This gives you a visual representation of the movement of the scene.

A motion path is a line curving through space that visually depicts both the path of the object during animation and the position of key frames.
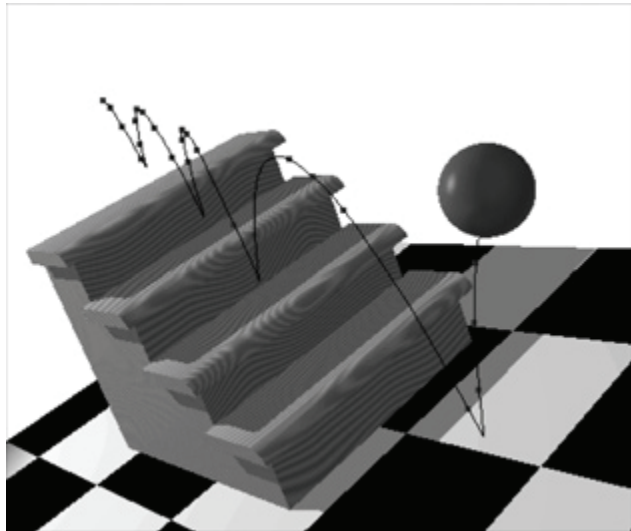
Creating Motion Paths

To move an object along a motion path, you assign a motion path method to an object, then create and edit the path.

To assign motion path animation to an object:

1. Select **Properties tray: Motion tab: Motion panel: Motion Path**.

2. A dialog appears, warning you that you are about to change the motion method (from the default of Key Frame). Set the level of conversion you want.

After you assign motion path animation, the Motion Path tools are available. These tools appear whenever you select an object with motion path animation

To draw a motion path:

1. Position and orient the object at the start of the path. If you move the object later on, Carrara automatically redraws the motion path to start at the new location.

2. Use the Working Box control to set the active plane. This is where the path will be placed.

3. Select the Extend tool.

4. Use the Extend tool to place the turning points and ends of the path on the active plane. Carrara draws the path, using Bézier curves to connect the points. You can convert these curves to corners using steps given later in this section.

5. Select the Selection tool to stop drawing the path. To see your object move along the motion path, you must place the object along the path at key points. The distance along the path is the animatable parameter.

To move the object along the path use one of the following methods:

- To drag the object, Command/Ctrl click the object and drag it along the path.

- To move the object to a specific position on the path, enter a value for Distance Along Path in the **Properties tray: Motion tab: Motion panel.** Carrara uses the unit of measurement set in File preferences (General Scene Settings).

Moving the Motion Path

After creating a motion path, you may want to move it to a different location in the scene.

To move a motion path to a different location:

1. Select the object that has a motion path.

2. Select the Move Entire Path tool in the motion path tools.

3. Click anywhere on the motion path and drag the mouse.

Modifying Motion Paths

You can edit the path and change the way the object moves along it in several ways:

- Delete points

- Change points from curves to corners and vice versa

- Directly manipulate Bézier curves

- Bank and reorient the object as it moves along the path

Editing Points

Use the Motion Path tools to add, delete, and convert points. Use the manipulation handles to reshape the Bézier curves.

To delete points from the path:

1. Select the Delete Point tool.

2. Click the points on the path that you want to delete.

To convert curve points and corner points:

1. Select the Convert Point tool.

    2.   Click corner points to convert them to curve points. Click curve points to convert them to corner points.

To directly manipulate Bézier curves:

    1.   If the manipulation handles are not visible, enable Display Handles in the **Properties tray: Motion tab: Motion panel.**

    2.   Drag the manipulation handles to reshape the path.
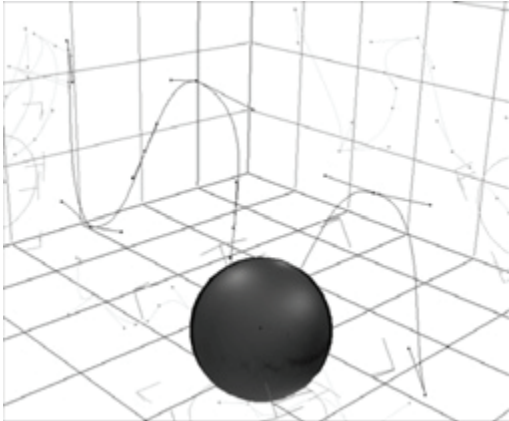
TIP! You can show or hide the projections of the motion path on the working box by checking Display Projections in the **Properties tray: Motion tab: Motion panel**. When the handles and the projections are visible, you can even drag the handles on the working box to manipulate the Bézier curve in a plane.

Banking and Aligning Objects

You can make an object's motion along a path appear more realistic by using the align and bank options.

When you align an object on the motion path, the object continually reorients itself to follow the path. For instance, a toboggan aligned on a path would always point in the direction of its motion, not the direction of its original position.

If an object is not aligned, it maintains its global orientation throughout its movement along the path.
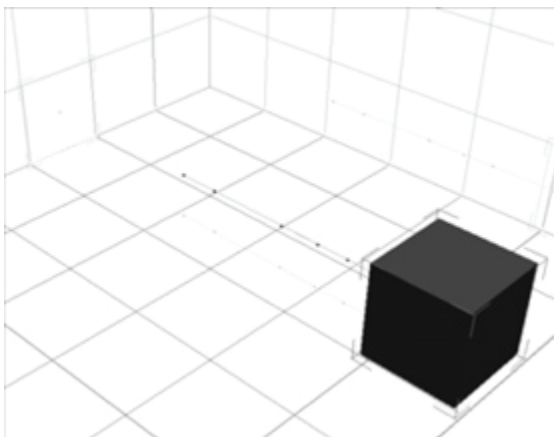
When the alignment option is set, the object changes its orientation to follow the curves of the path.
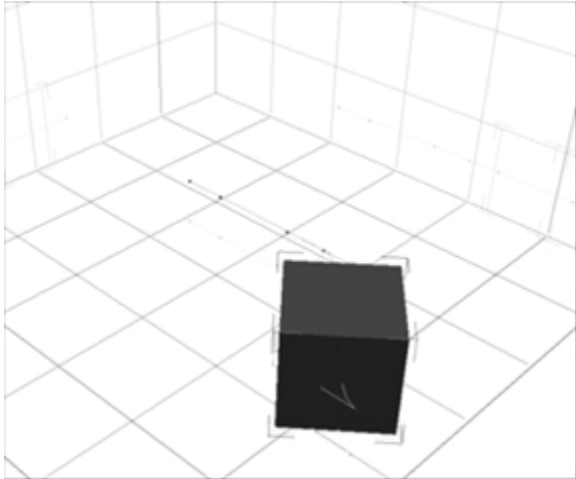
When you bank an object, it swings in and out as it turns corners and curves.

To keep an object with a motion path aligned to global coordinates:

1. Select the object.

2. In the **Properties tray: Motion tab: Motion panel,** select Not Aligned.

As the object moves along the path, its rotation around the X,Y and Z axis do not change.

To align an object to the motion path without banking it:

1. Select the object.

2. In the **Properties tray: Motion tab: Motion panel,** select Align.

To automatically bank an object as it moves along the path:

1. Select the object.

2. In the **Properties tray: Motion tab: Motion panel,** select Align and Bank.

3. Set the Maximum Bank slider to set the maximum amount the object can tilt on the path. A setting of 100% allows the object to bank 45° on each side.

Motion Path Coordinate System

When an object inside a hierarchy moves along its motion path, there are several ways to interpret the positions of the path in the scene. This is controlled by the Coordinate System radio buttons in the **Properties tray: Motion tab: Motion panel .**

- **Parent**: the motion path follows the movements of the object's parent. This was the only option available prior to Carrara 4. In the example that follows where the sphere is a child of the cube, you can see how the sphere keeps moving around the cube as it goes right.
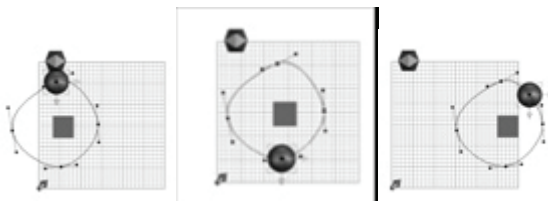
- **World**: the motion path stays at the same place regardless of the object's parent movement. In the example below, the sphere is still a child of the cube but doesn't follow it as it goes right.
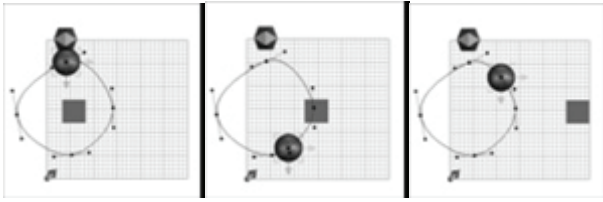
Non-Linear Animation (Standard, Pro)

Non-linear animation allows you to create smaller animation clips that you can later repeat, layer, and combine to create a more complete animation.

Creating an Animation Group

To begin with non-linear animation, you have to first create at least one animation group. In the case of a figure, an animation group is already created for you

To create a new animation group:

1. Select the object that you want to add to the animation group.

2. Choose **Animation > Create Animation Group** from the main menu.

   The new animation group appears in the Sequencer. An animation track appears, with the items contained in the animation group listed beneath them. You'll also see the Animation Group and the associated objects listed in the Instance tab in the Assemble Room.

Track and Clip Menu Commands

You can use an animation track to combine several clips that contain individual portions of the animation. For example, you can create an animation track for one character in your scene. The track can then include four individual clips that contain a walk cycle, arm movements, head turns, and facial expressions. The beauty of this approach is that you can move the clips forward or backward in the timeline without affecting the contents of other clips.

If you right-click an Animation Group, Track, or editable clip in the sequencer, a context menu appears. The command options that are available depend on whether you have an animation group, track, or clip selected.

- **Cut**: Cuts the selected item and places it into the clipboard.

- **Copy**: Copies the selected item and places it into the clipboard.

- **Paste**: Pastes the selected item into the current track.

- **Delete**: Deletes the selected item.

- **Duplicate**: Duplicates the selected item.

- **Group**: Groups two or more animation groups together.

- **Ungroup**: Removes grouping from items that have previously been grouped.

- **Add NLA Track**: Adds a new animation track below the currently selected track.

- **Add NLA Track Group**: Creates a group that NLA tracks can be grouped into. Drag tracks onto the group as needed.

- **New Editable Clip**: Click this button to open the Clip Creation dialog. You are prompted to enter a name for the clip, and whether to use the current frame as a starting point, or a custom time that you enter. You also specify the length of the clip. After making your choices, click OK to continue.

- **Edit Clip**: Allows you to animate the items that are assigned to an Editable Clip.

- **Finished Editing Clip**: Exits Edit mode and applies all changes to the editable clip.

Using NLA Tracks

Adding NLA Animation Tracks & Groups

After you create an Animation Group, you can add one or more tracks to it. Each track can hold the animation clips for any object in your scene (or multiple objects, if you prefer). Tracks are used to layer animations&ldots; for example, you can use a track to override the motion of a hand in a walk cycle.

To create a new NLA Track,

1. Scroll through the Sequencer to find the Animation Group that you want to add a track into.

2. Right-click on one of the tracks in the animation group. The menu described in "Track and Clip Menu Commands" appears.

3. Choose **Add NLA Track** from the menu. The new track appears directly beneath the track you initially selected.

To group NLA Tracks:

1. Scroll through the Sequencer to find the Animation Group that you want to add a track group into.

2. Right-click on one of the tracks in the animation group.

3. Choose **Add NLA Track Group** from the menu. The new group appears immediately below the selected track.

4. Select and drag NLA Tracks into the group or create new NLA Tracks within the group.

Animation Track Properties

After you create and select a track, you will notice the following properties in the Assemble Room, in the Properties tray: General tab: NLA Clip Track panel.
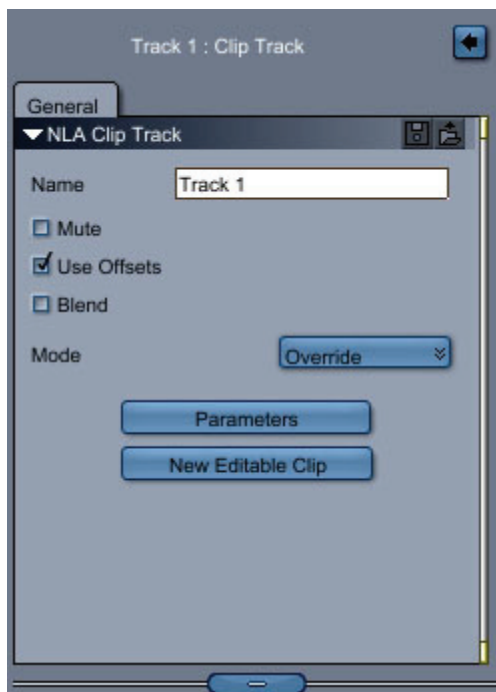
- **Name**: Assign a name for the track in this field.

- **Mute**: Check this option to turn the track off so that it has no affect on the animation.

- **Use Offsets**: Check this option to determine whether the track will use the offset between clips.

- **Blend**: Check this option to make the track blend with previous tracks in areas where there is no clip, or in the blend zone of clips.

- **Mode**: The mode decides how blending is applied. Choose **Override** if you want the tracks to override the values of the parameters that are controlled by the clip. Choose **Additive** if you want to add the values of the track to the values of the clip.

- **Parameters**: Click this button to open the Parameters Selection dialog, which displays the items that are included in the current NLA Track. Check or uncheck the items that you want to include in the animation track.

- **New Editable Clip**: Click this button to open the Clip Creation dialog. Carrara prompts you to enter a name for the clip, and to choose the starting point for the clip. You can use the current time as a starting point, or enter a custom time in minutes, seconds, and frames (MM:SS:FF). Specify the length of the clip in MM:SS:FF format. After making your choices, click OK to continue.

Using NLA Clips

Creating Editable Clips

Clips are small segments of animation that can be moved forward or backward in time and looped over a longer range of animation frames. In addition, clips can be layered with each other to combine several individual motions into a single, fluid animation. For example you can create a clip that makes your character move its arms and legs to create one segment of a walk cycle (left foot, then right foot). Stretch the clip over time to make the character walk across the screen. A second clip can make the character look right or left, and you can move the clip forward or backward in time without affecting the walk cycle. A third clip can contain facial expressions … a blink, for example, that you can reuse in several places in the animation and move wherever you want the blinks to occur.

The more you layer these clips, the more complex your animation becomes. Later, when you need to make changes to the animation, you only edit the clip that contains the portion you want to change. The other clips leave the rest of the animation intact until you edit other clips.

When you first create a clip, it's always editable because you have to create the desired actions within it. The Clip Creation dialog shown below appears when you create a new editable clip.
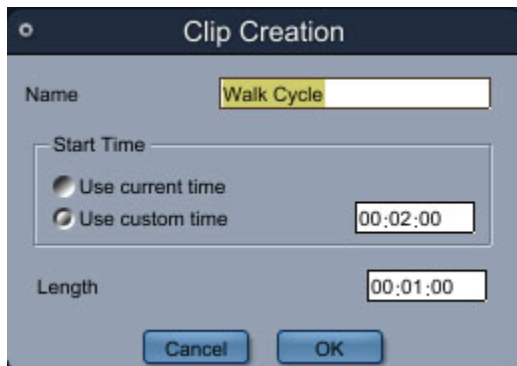
To create an editable clip:

1. Right-click in an animation track to display the context menu discussed in "Creating an Animation Group."

2. Choose **New Editable Clip**. The Clip Creation dialog appears.

3. Enter a name for the new clip in the **Name** field.

4. In the **Start Time** section, choose one of the following options:

   - **Use Current Time**: Choose this option to start the clip at the current frame.

   - **Use Custom Time**: Choose this option to start the clip at a specified frame in your animation. Time is entered in minutes, seconds, and frames (MM:SS:FF). Note that you can always drag the clip forward or backward in the timeline if you enter an incorrect start time.

5. Specify the length of the clip in minutes, seconds, and frames in the Length field.

6. Click OK to create the clip.

Adding Animation to Editable Clips

At this point, you've created an animation group that can contain one or more tracks. Each track can contain one or more editable clips. The next step is to edit the clip and add the keyframes necessary to produce the actions you want to include in the clip. For example, you can create the keyframes necessary to create one segment of a walk cycle.

To add animation to objects in an editable clip, follow these steps:

1. Right-click on the Editable Clip that you want to change. Editable clips are bounded by a red box.

2. Choose **Edit Clip** from the menu. The clip is highlighted in red in the NLA track, and is labeled **Clip (Editable)**.

3. Move the scrubber into the region of the editable clip. Add keyframes as necessary to animate the objects to complete your desired animation sequence.

4. When you are finished with the animation sequence for the clip, right-click on the Clip (Editable) clip in the NLA track, and choose **Finished Editing Clip**.

Clip Properties

When an editable clip is selected in the Sequencer, the General tab in the Assemble Room allows you to set the following properties:

- **Name**: Displays the current name for the editable clip. This field is editable.

- **Color**: Click the Color square to open a color picker. This allows you to choose a display color for the selected track. The clip displays this color in the timeline when it is not selected.

- **Time Range**: Displays the start and end times of the clip in minutes, seconds, and frames (MM:SS:FF).

- **Blending**:

- o **Auto Blend**: Choose Auto Blend to automatically compute the blend zone based on the previous clip and the next clip in the track.

- o **Start**: Enter the location in the timeline where you want the blending to start. Time is entered as MM:SS:FF.

- o **End**: Enter the location in the timeline where you want the blending to end. Time is entered as MM:SS:FF.

- **Offsets**:

  - o **Offsets Relative to Previous Clip**: The offsets control how the position is matched between the end of the previous clip and the beginning of the current clip. For example, you can use offset to match the position of the foot between two walk cycle clips so that the character starts the second walk cycle from the end of the first one.

  - o **Translation**: Enter the location or locations on the X, Y, and/or Z axes where you want translation offsets to occur.

  - o **Rotation**: Enter the degrees of rotation along the X, Y, and/or Z axes where you want rotation offsets to occur.

  - o **Match Previous Position**: Click this button to open the Match Previous Clip dialog shown in the following figure.
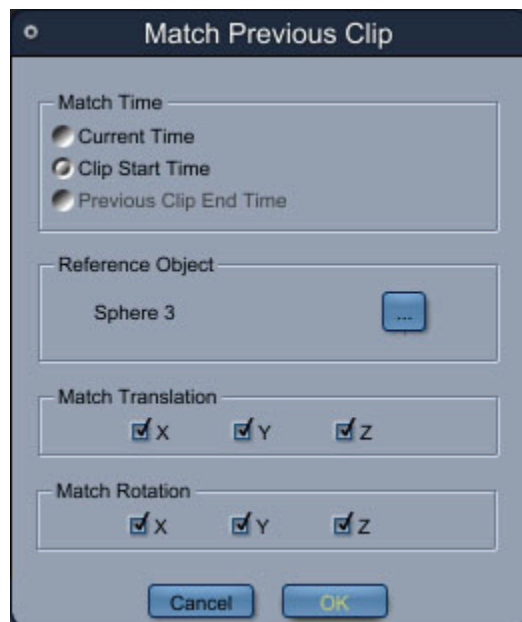
Options are:

- ▪ **Current Time:** Matches the end position of the previous clip in the timeline to the current position in the timeline.

- ▪ **Clip Start Time:** Matches the end position of the previous clip in the timeline to the beginning of the current clip.

- ▪ **Previous Clip End Time:** Matches the beginning of the current clip to the end of the previous clip in the timeline.

- ▪ **Reference Object:** Click the button to select the reference object that you want to match.

- ▪ **Match Translation:** Check or uncheck options to match the reference object along the X, Y, and/or Z axes.

- ▪ **Match Rotation:** Check or uncheck options to match the rotation of the reference object along the X, Y, and/or Z axes.

- **Edit Clip/Finished Editing Clip**: Click this toggle button to edit the animation in the current clip, or to finish editing the clip.

- **Convert to Master Clip**: Click this button to convert the editable clip to a master clip as described in "Creating Master Clips."

Creating Master Clips

Master Clips are displayed in the Sequencer in the Master Clips category. You can expand the section to display all Master Clips in your current project.

When you add multiple instances of a master clip into your scene, you can make changes to one instance and the changes will apply to all other instances of the same clip. For example, if you create a clip of a figure dancing and apply it to several characters, you can change the timing in one instance and the timing will update in the other figures.

To convert an editable clip to a Master Clip:

1. In the Sequencer, click the Editable Clip that you want to convert.

2. The General tab displays the properties for the editable clip. Scroll to the bottom of the properties and click the **Convert to Master Clip** button. The Clip Conversion dialog appears with the following options:

   - **Loop Offset**: Click to select an object in the clip that will determine how loop offsets are calculated. If you use a clip to create a loop on the animation, you still need to offset the motion inside the clip itself. The Loop Offset controls how this offset is calculated, which body part is used as a reference (foot or hip, for example) and the offset vector between two repetitions.

   - **Parameters**: Check or uncheck the items in the hierarchy that you want to include in the master clip. This controls which parameters are controlled by the clip. If the clip does not control some parameters then the current value of the track and/or previous track (depending on the blending options) are used for those parameters.

3. Click OK to exit the dialog. If not previously present, a new Master Clip tab appears in the right portion of the screen, behind the General tab. This tab later allows you to adjust the loop offset of the master clip whenever you need to.

Using Existing Master Clips

When you create master clips in your project, they appear in the Clips tab in the Browser, as shown in the following figure.

1. Drag and drop any of the clips into the sequencer, and drop them onto any NLA track.
2. Drag the clip forward or backward in the timeline to reposition the clip if necessary.
3. Use the Loop Offset settings in the General tab to control which body part is used for a reference, and to control how the offset is calculated.
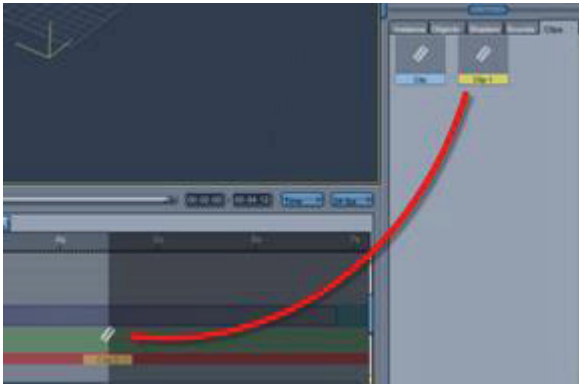
Blending Animation Clips

To blend clips, extend the blending area to the left and right of the clip. If two clips overlap, the two motions are blended, much like a crossfade between audio or video tracks in a non-linear editor.

Physics (Standard, Pro)

The physics motion method specifies that your objects accelerate, move, and interact in ways consistent with the laws of physics.

Using physics allows you to create realistic animations that would be excessively complex with key frame motion alone. You can get objects to slide down a slope, bump into each other and bounce away, and other useful motions. You can learn a lot about physics and how to use it by experimenting, or by viewing existing animations.

 NOTE! Physics may not work with some of the modifiers.
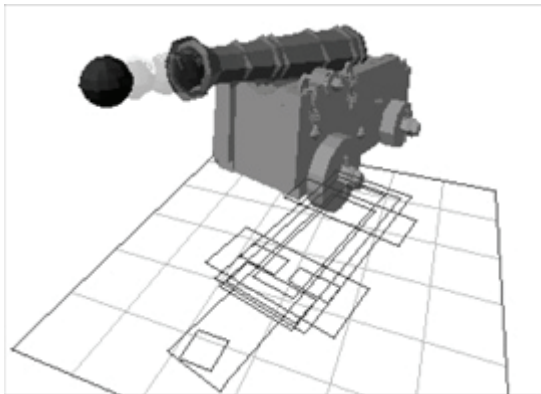
Physics and Key Frames

Physics precludes arbitrary position settings in key frames. For example, if you apply Directional Force to get an object to fall, you can't also make it float in the air, defying gravity. The object must either obey physics (and fall) or float.

When you assign physics to an object, Carrara removes all key frames except the starting key in frame 1 that would arbitrarily set position or orientation on the object. Its other properties, like shading, remain

animatable with key frames. To preserve physics logic, Carrara won't let you move an object that has physics applied later than frame 1.

You can't set position or orientation key frames for objects that use physics. You can, however, set key frames for the property values in the physics themselves. For example, you can accelerate an object, and then apply damping to stop it, and then accelerate it in a different direction. You'll learn more about this later in this section.

Note Initial Velocity describes the object's inertia in frame 1. Therefore, this force cannot be changed with a key frame.

You might want an object to respect physics, but still be animated with key frames. The primary application of this is to create an object that can move with key frames while having objects responding to physics bounce off of it. However, such an object will be unaffected by forces.

Another possibility is to develop the animation using physics, and then translate the result into key frames by setting the motion method back to key frame.

About Key Frames in Physics

You can change the physics settings to create key frames. Before you try this, you need to understand that the forces describe acceleration, not velocity. (Initial Velocity does set velocity, but it is only the initial state. It cannot be used for a key frame.)

For example, you can start an object moving with a Directional Force of X=10. A couple of seconds into the animation, change the Directional Force values to X=0 to create a key frame.

When you play the animation, the object will not stop at the moment of the key frame. It will continue moving in X until it encounters a physical object (and rebounds) or some other force acts on it.

This works a lot like a rocket in space because Carrara doesn't take into account friction or gravity, because there is no air friction or default gravity in Carrara.

Applying Directional Force is equivalent to an engine burn. Shut off the engines and the rocket coasts along on its trajectory. Without friction to slow it down or gravity to pull it down, the rocket continues on this vector at this velocity indefinitely.
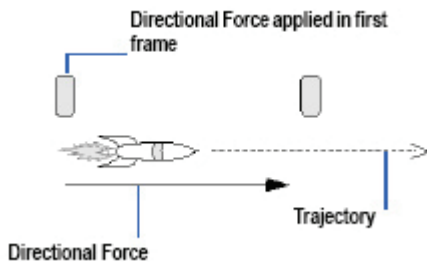
Next, the rocket rotates 90° and does another engine burn (this is the same as adding a new perpendicular Directional Force). The new trajectory is the combined effect of the initial acceleration and the secondary, perpendicular acceleration. The new velocity is the product of the old velocity and the new force (acceleration).
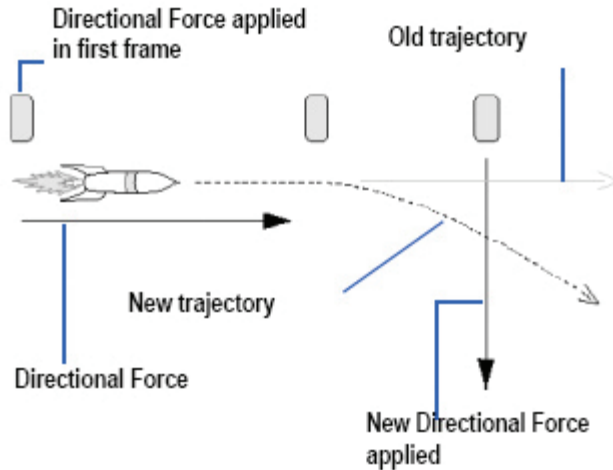
If you want to exert more control over transitions, you can use a Discrete tweener or use Damping Force in key frames just prior to a change of acceleration.

For more information, refer to "Key Frame Animation."

Applying Physics to an Object

Carrara includes a number of different forces that you can use to affect objects in your scene. However, forces cannot affect an object unless you assign physics motion to the object or objects you want to affect.

To apply physics to an object:

1. Rewind your animation to frame 1.

2. Select the object to which you want to apply physics.

3. In the **Properties tray: Motion tab: Motion panel, choose Physics** from the animation list.
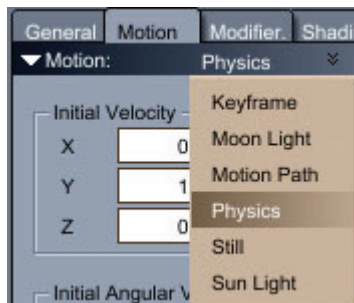
4. Carrara notifies you that existing keyframes will be converted. After you select conversion quality options, click OK to continue.

5. The physics controls for the selected object become available in the **Properties tray: Motion panel.** These controls are described in the following section.

6. Set additional options in the Properties tray: Effects tab: Physical Properties panel as outlined in Physics Effects, immediately following the next section.

Physics Controls

When you select physics as your motion method, a control panel appears in the **Properties tray: Motion tab: Motion panel** when the affected object is selected. This panel allows you to enter information on

how physics behave in your animation. You can experiment with modifying values to achieve the effect you want.
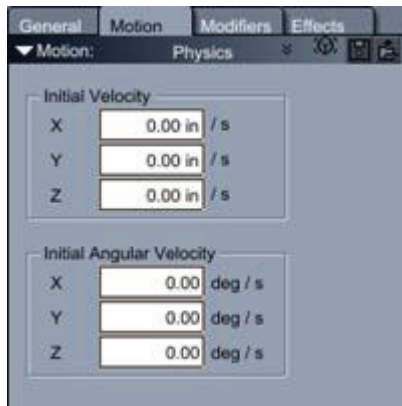
- **Initial Velocity:** Set Initial Velocity options for one or more axes to determine the direction of the object's movement. You can enter positive or negative values. This setting applies directional motion to an object in frame 1. If no other force influences the object, it will maintain its initial velocity indefinitely.
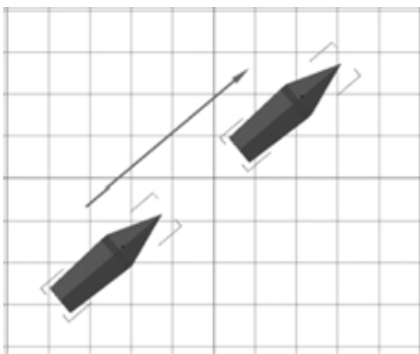
- **Initial Angular Velocity:** Set Initial Velocity options in each dimension to determine the angle of the object's movement. You can enter positive or negative values. This setting applies angular motion to an object in frame 1. If no other force influences the object, it will maintain its initial velocity indefinitely.

Physics Effects

With the object selected, click the **Properties tray: Effects tab**, to view the **Physical Properties** panel and its controls.
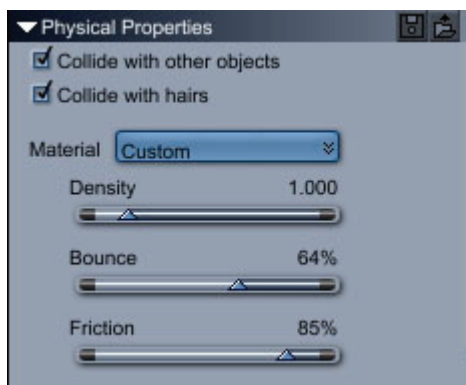
Physical Properties appear in the Effects tab.

- **Collide with Other Objects:** Specifies whether the specified object will collide or not. An object with this option turned off can still have forces applied to it, but it will ignore geometry and move through other objects.

- **Collide with Hairs:** Check this option if you want physics to interact with the dynamic hair in your scene. Note that this will increase system resource requirements.

- **Material:** Allows you to specify common materials, which have preset values. These materials can be used as is, or you can modify their values as needed.

- **Density:** Sets the object's density.

- **Bounce:** Determines how far the object will bounce back after colliding with another object. A value of 0 sets no bounce, while a value of 100 specifies that no energy is lost in the bounce. Values over 100 are unrealistic although they can be used to create special effects.

- **Friction:** Determines how slippery the surface of the object is. When other objects slide on an object with a high Friction setting, they will lose kinetic energy and come to rest faster.

Adding Forces to the Scene

After you apply physics to an object and set the physics controls and effects, you need to add one or more forces to the scene in order to animate the object with physics.

There are five different types of forces, each of which will be described in more detail in the sections that follow:

- **Directional Force:** Propels an object in one or more directions.

- **Point Force:** Draws an object or objects toward its center.

- **Damping Force:** Dampens or counters the effect of another force.

- **Torque Force:** Rotates an object or objects (like the blades of a fan)

- **Flow Force:** Randomizes direction of one or more objects.

To add forces to a scene:

1. Use one of the following methods:

2. Choose **Insert >** and select one of the Force options.
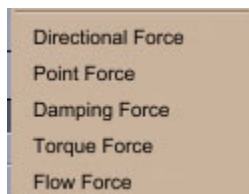
3. Drag one of the physics tools into the scene. To apply a force to a single object or group, drag the force on top of the object in the hierarchy.

4. Each force type has its own **force helper object** that appears in the scene. This is a non-rendering wireframe representation of the selected force. You can select this object to orient the forces and set the effect properties.

    Reorient the physics helper object to change the direction of gravity.

5. Enter the effects settings for the force you've selected. The effects you add are specific to the type of force you are adding, and are described in the sections that follow.

6. Repeat steps 1-3 to add additional forces if desired. You can apply any number of forces in any order.

7. To simulate the physics motion for previewing purposes, click the **Simulate Physics tool**.
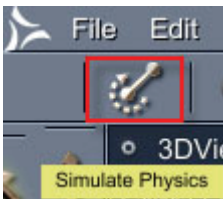
8. Preview the animation to see how the forces influence object motion. You might want to extend the render range to see more action.

9. After previewing, remember to rewind to frame 1. If you make changes without rewinding, you'll be setting a key frame at this later moment of the animation.

Units of Measurement for Forces

The default values for the units of forces can be set or modified by selecting **File > Preferences> General Scene Settings**.

Carrara uses the following consistent set of units of force:

- Length: inches
- Time: s (seconds)
- Mass: g (grams)
- Acceleration: m/s2
- Angle: degrees
- Angular velocity: degrees/s
- Volumetric mass: g/dm3

- Force: g*in/s2
- Velocity: in/s.

Directional Force and Effects

Directional force applies a force to an object. It does not set velocity. Directional force accelerates an object in one direction to produce the effect of gravity:such as falling objects.

Without direction force, the ball would continue along its original trajectory indefinitely.
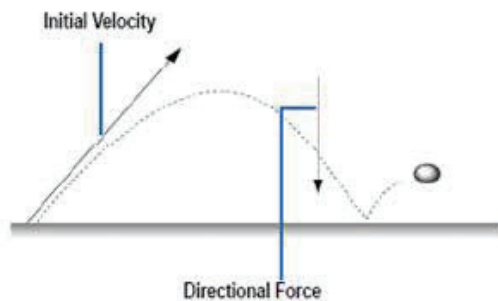
This trajectory is accomplished by adding a directional force.
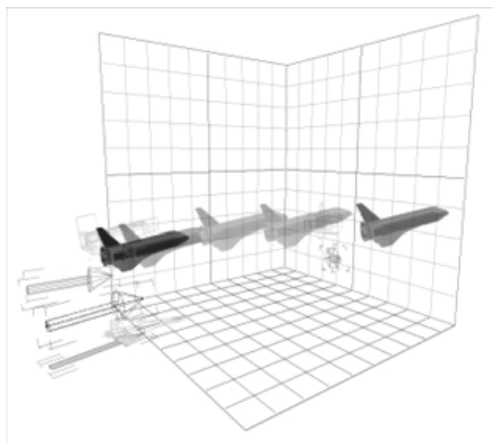
To set Directional Force Effects:

1. After adding physics to your object and setting physics controls and settings, do one of the following:

2. Select **Insert > Directional Force**.

3. Drag the **Directional Force** tool onto an object or objects in the hierarchy.

4. Open the **Properties tray: Effects tab**. The following options appear:

   - **Strength**: Sets the intensity of the force.

   - **Rocket**: This property is used to apply the force as a rocket, by making the force a child of the object.
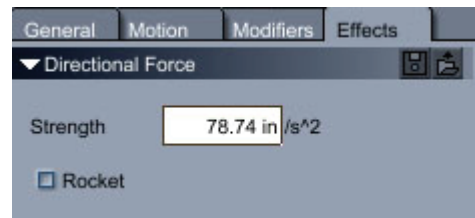
Point Force and Effects

Point Force accelerates an object towards the force center. You can use point force to simulate the gravitational pull of a celestial body.
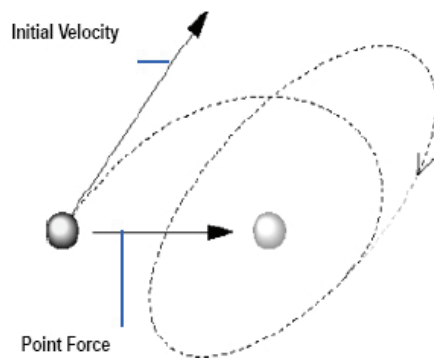
It can also be used to simulate electromagnetism or gravity in space. You can also invert the force to repulse objects.
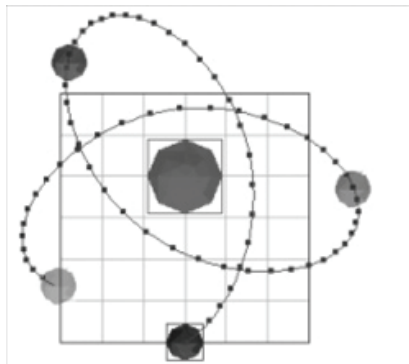
To set Point Force Effects:

1. After adding physics to your object and setting physics controls and settings, do one of the following:

    o Select **Insert > Point Force**.

    o Drag the **Point Force** tool onto an object or objects in the hierarchy.

2. Open the **Properties tray: Effects tab**. The following options appear:

    o   **Strength**: Controls the intensity of the force.

    o   **Field Decay Rate**: Controls the rate at which the force diminishes with the distance.

Damping Force and Effects

Damping Force counters any other forces or movement working on the object. It will slow an object's velocity. Regardless of what inertia an object has or what force is applied, damping counters it.

Damping force works like fluid friction in that opposes motion the same way a liquid surrounding an object would.
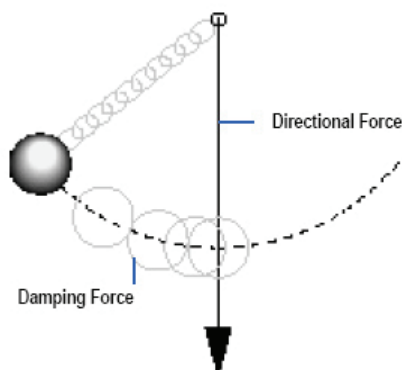
The effect of damping is not immediate. It's like applying the brakes in a car. Even at maximum braking, the car takes a few feet to slow down and stop.

To set Damping Force Effects:

1. After adding physics to your object and setting physics controls and settings, do one of the following:

    •   Select **Insert > Damping Force**.

    •   Drag the **Damping Force** tool onto an object or objects in the hierarchy.

2. Open the **Properties tray: Effects tab**. The following options appear:

- **Strength**: Controls the intensity of the force.

Torque Force and Effects

Torque Force applies a force that tends to rotate objects. Use it to simulate a motor.

To set Torque Force Effects:

1. After adding physics to your object and setting physics controls and settings, do one of the following:

   - Select **Insert > Torque Force**.
   - Drag the **Torque Force** tool onto an object or objects in the hierarchy.

2. Open the **Properties tray: Effects tab**. The following options appear:

   - **Strength**: Controls the intensity of the force.

Flow Force and Effects

Flow Force creates randomized directional motions. You can use Flow Force to give motion to autumn leaves swept up by the wind.

Physics Settings

Additional physics properties can be applied to the entire scene. Select the Scene from the instance tab and click the Physics tab to display the following options:

- **Simulation Accuracy**: High is better than Medium only when objects interact at an exceedingly rapid rate.

- **Geometric Fidelity**: At Low or Medium levels, surface geometry is simplified to speed up calculations. If an object doesn't seem to move correctly, revert to the High setting

- **Gravity**: Check the Use Default Gravity checkbox to add gravity to your scene. Enter the amount of gravity force to apply to the scene in the value boxes provided.

Animating in Other Rooms

Animating an Object's Shape

You can animate an object by changing its shape over time. To do this, create the object, change its shape using a feature such as the modeler or a modifier, then create a key frame for both the original and the new shape. Carrara creates the transitions necessary to "tween" the original into the modified shape, thus creating motion.

The type of object determines which modeler you'll use, and therefore, what modifications you can make. For example, with the Spline modeler, you can edit the cross-sections, sweep path, and scaling envelope. With the volumetric primitives, like Fire and Cloud, you use sliders to change the character of the object.
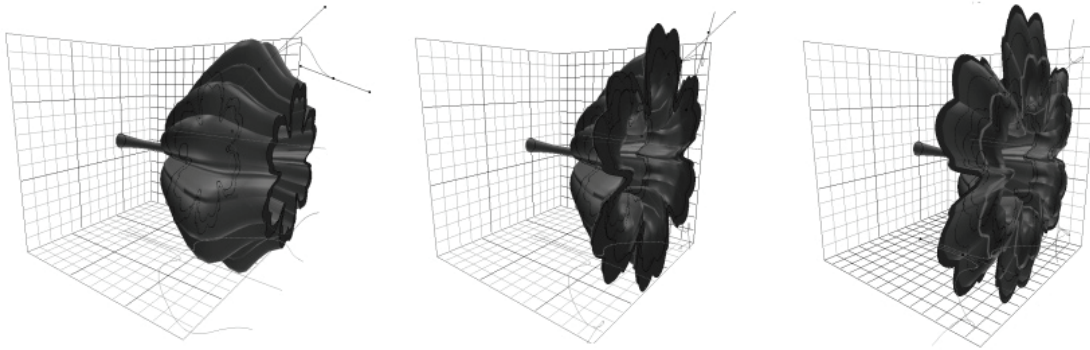
Animating Spline Objects (Standard, Pro)

You can change the shape of a spline object during animation using the Spline modeler. For complete information on the Spline modeler, refer to "Spline Modeling Overview."

There are some limitations to the kinds of changes you can make in your models, but generally you can edit all cross-sections and points on an extrusion path or scaling envelope.

NOTE! You **cannot** add or delete cross-sections or points, or change the basic structure of the object.

To animate the shape of a spline object, first create an object in the Spline modeler using cross-sections and a sweep path.
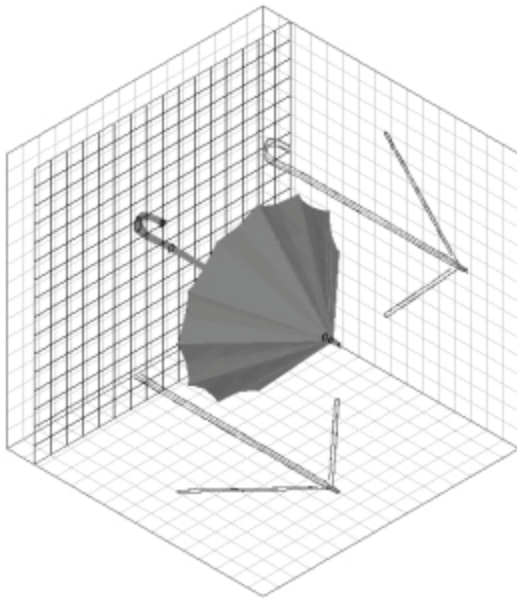
...then adjust the shapes in a different frame.
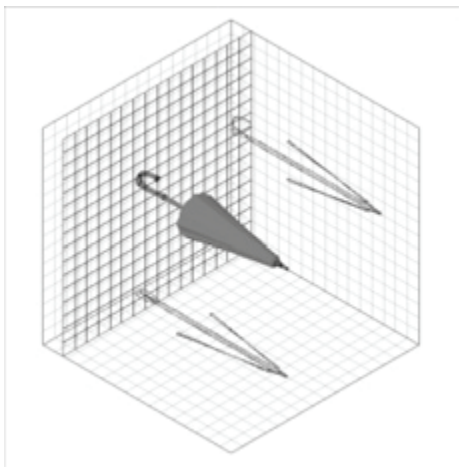
To create spline models whose shapes you want to animate:

1. Create all of the control points for the extrusion path.

2. Create all of the cross-sections, with all of the control points.

   Start with the most complex shape your object will have during the entire animation.

To animate the spline shape:

1. Set the current time bar to the point where you want the key frame to occur.

2. Select an object.

3. In the Spline modeler, adjust the points and curves of any cross-section, extrusion path, or envelope.

   You may not add or delete points or add a cross-section shape.

   Carrara adds a key frame marker to the spline object's Time Line.

When you run the animation, Carrara fills in the action between the two shapes to show the change of shape or movement.

You can change the tweener between the two event markers and set the transition method.

Animating Vertex Objects

In Carrara Standard and Pro, You now have two ways to animate vertex objects. You can change the shape of a vertex object during animation using Morph Targets in the Vertex modeler. For complete information on the Vertex modeler, refer to "Vertex Modeling Overview," "Vertex Modeling Basics," and "Vertex Modeling Techniques."

Additionally you can create a skeleton and attach it to a vertex object and use the skeleton to animate the object. See "Character Animation" to learn about character animation using a skeleton.

In Carrara 3D Express, you can change the shape of a vertex object during animation using Morph Targets in the Vertex modeler.

NOTE! With the Vertex modeler, you can only animate movement. You cannot perform any operations such as loft, extrude, or sweep on an object that will add or remove vertices.

Animating Metaball Objects (Standard, Pro)

You can change the shape of a Metaball object during animation using the Metaball modeler. For complete information on the Metaball modeler, refer to "Metaball Modeling."

To create metaball models to animate:

1. Create all of the metaballs for the object. Since you cannot add or delete metaballs during the animation, you must create all you'll need at the outset.

2. Shape the object into the desired form using any of the modeler's features. Since you can adjust the position of metaballs during the animation, the object you create at the onset doesn't have to be the final form. You can move metaballs to deform the object over time.

To animate the metaball shape:

1. Set the current time bar to the point where you want the key frame to occur.

2. Select the object to be animated.

3. In the Metaball modeler, change the position of the object's metaballs.

    You may not add or delete metaballs.
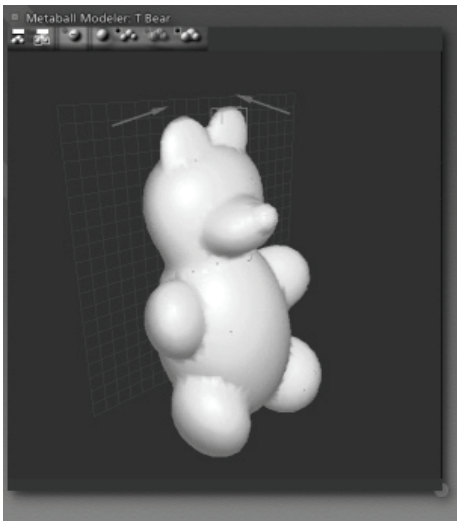
Carrara adds a key frame marker to the metaball object's Time Line.
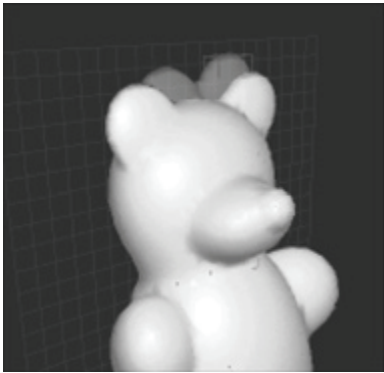
4. You can now change the tweener between the two event markers and set the tweener's transition method.

Animating Environmental Primitives

The **environmental** primitives are Fire, Clouds, Fountain, and Fog. Although each of these primitives has its own modeler, the technique for animating them is the same.

Animations of the environmental primitives will not be visible when previewing. You'll need to do a test render to see the results.

To animate an environmental primitive's shape:

1. Create the primitive and set the attributes you want it to have at the start of the animation.

2. Click the Current Time bar at the point where you want the key frame to occur.

3. Use the primitive's modeler or the features in the Properties tray to change the primitive's properties.

4. Click the time bar at the point where you want the next frame of the volumetric primitive animation.

5. Continue to adjust the properties to create new key frames. Set the completion slider.

    Some of these primitives have completion sliders which you can animate from 0 to 100%.

    For example, you might want the Fire to change the quantity of flames or its upward speed.

    - Fountain primitive with completion at 0%.
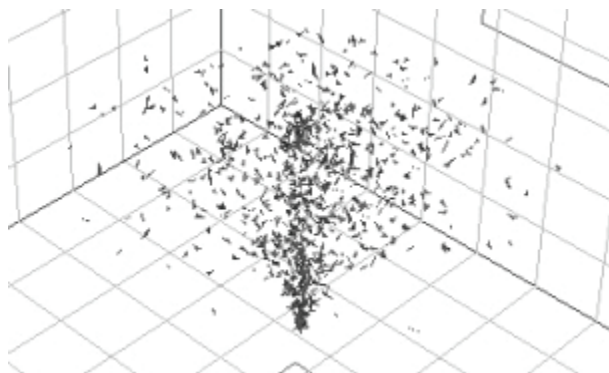
- Fountain primitive with completion at 50%.
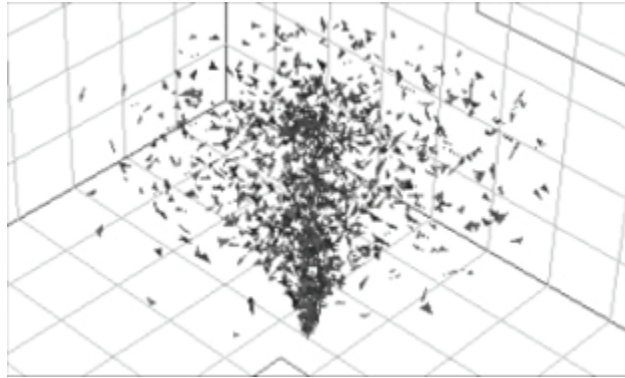
- Fountain primitive with completion at 100%.
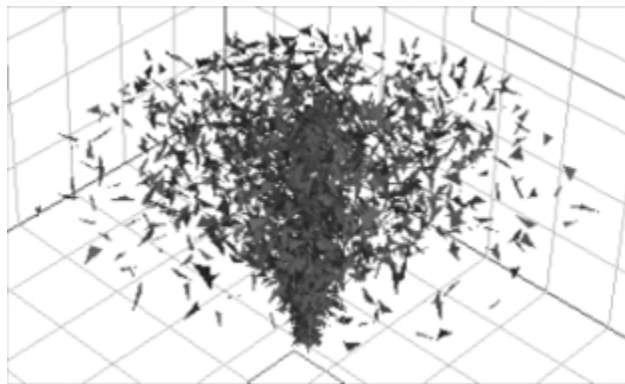
6. You can now change the tweener between these event markers to set the transition method. For example, to get flickering flames, use the Oscillate tweener.

Animating Object Motion

You can use rotation, duplication, and cloaking to enhance some motion effects.

Refer to "Animating with Rotation" to learn about rotation or "Duplicating Relative Motion with Groups" to learn about duplication.

For instructions on working with hot points, refer to "Moving the Hot Point."

NOTE! You can rotate an object around its hot point or around its center. To use the hot point, be sure it is set in the Properties tray: Motion tab: Transform panel: Rotate around control.

Cloaking an Object

When you want an object to enter or exit (appear or disappear) during the course of your animation, you use a technique called cloaking.

Cloaked objects are visible and can be manipulated in the 3D view; they simply are not included in the rendering of your animation.

You can take advantage of cloaking to switch one version of an object for another version of the same object. This is advantageous when the two versions have contradictory properties, which you need at

different times of the animation. As long as the two objects look the same, occupy the same space, and cloak and uncloak in the same frame, the exchange from the one to the other will be seamless.

To cloak or uncloak an object at a specific point in time:

1. Select the object.

2. Click on the key frame marker for that object on the Time Line.

3. Enable or disable the Visible checkbox.

Animating Shaders

Animating shaders opens a wide range of possibilities. You can animate virtually any type of change to a shader that you can imagine, from a simple color change to a shifting geometric pattern. (You can even make an object appear to change from bricks to glass or from marble to metal.) For complete information on shaders, refer to "Creating and Editing Shader Trees."

The only limitation is that you cannot replace one shader component with another during the course of an animation. You cannot, for example, replace a Wood function with a Spots function at a particular point along the Time Line. Thanks to the flexible, modular nature of the Shader Tree Editor, this turns out to be only a minor limitation.

To animate simple changes to shaders, use the Shader Tree Editor to adjust the parameters of individual shader components at specific points along the Time Line. Examples of parameters you can adjust include:

- Colors

- Values for attributes like transparency and shininess

- Procedural function parameters, like the number of squares in a checker pattern, or the undulation of the veins in a wood pattern.

Animating Shaders with the Multi-Channel Mixer

The Multi-Channel Mixer component allows you to animate more drastic changes to an object's shader. In the following example, a Multi-Channel Mixer is used to create a shader which animates gradually from wood to glass.

Rotoscoping in Non-Color Shader Channels

The most common use of rotoscoping is in a shader's Color channel, where it allows a movie to be played on the surface of an object. However, rotoscoping can also be used in other shader channels to create stunning visual effects. See "Rotoscoping."

The example that follows illustrates the use of a movie in an object's Transparency channel.

The three animation frames in the right column show a doorway materializing from the floor up. This effect was achieved by playing a grayscale movie in the doorway's Transparency channel.
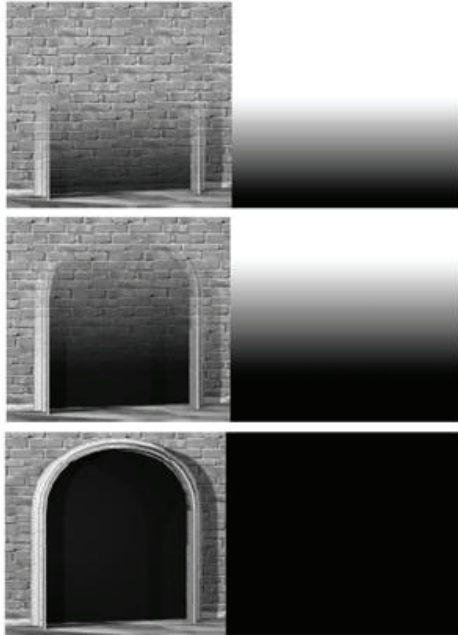
The three corresponding frames from that movie are shown in the left column. The white region in each frame represents the portion of the doorway that is transparent. The black region represents the portion that is opaque.

Animating with Modifiers

Modifiers allow you to dynamically alter the shape of an object depending on the type of modifier.

Modifiers let you give objects sets of instructions that give them an activity during the animation or that dynamically alter the shape of an object. Modifiers also reduce the number of key frames you need to create and let you create actions that are not otherwise possible.

Modifiers are especially useful in animation because they can be used to animate the shape of entire groups, which cannot be edited directly in a modeler.

Types of Modifiers

- Modifiers like Stretch, Bend and Twist, Explode, Dissolve, and Shatter produce interesting animated effects that cannot be achieved by other means. These do not need any special instructions.

- Some modifiers, such as Bounce and Spin, automatically assign complex actions to an object. greatly reducing the number of key frames you need to create. These and the other modifiers are explained in detail in "Modifiers."

- Other modifiers, such as Point At and Track, define or modify the motion of one object based on the location or motion of another object.

- The physics modifiers allow objects to move and interact as though they obeyed the laws of physics. The physics modifiers have special characteristics. which are described in "Physics." (Standard, Pro)

Character Animation (Standard, Pro)

In "Animating," you learned about key frame animation and were introduced to the animation features in Carrara. In "Animation Techniques," you learned about additional animation techniques.

This chapter discusses the character animation for vertex objects.

In Carrara 6, bending an arm is as simple as defining an articulation for the elbow and rotating the articulation; the vertices of the arm will automatically adjust their positions to make the bend appear natural. To animate a character, you simply need to create a skeleton, attach it to the character, and then animate the skeleton.

Working with Bones

A **bone** is a helper object made of Wireframe that is visible when you manipulate a scene but does not show up in the final rendering.

A bone's articulation (or **joint**) is represented by an octahedron around the bone's center of rotation. This is also the hot point for the bone. This joint cannot be moved independently from the bone.

A **skeleton** is a hierarchy of bones. You make a skeleton by inserting bones in the scene and dragging in the Hierarchy section of the Sequencer to create associations (see "Working with Objects") or by using

the Bone tool. When a bone is a child of another bone, two lines pointing to its parent bone are added to its wire frame display.

- The root of the skeleton is the top-most bone of the skeleton's hierarchy:

- Two bones:

- A complete skeleton:
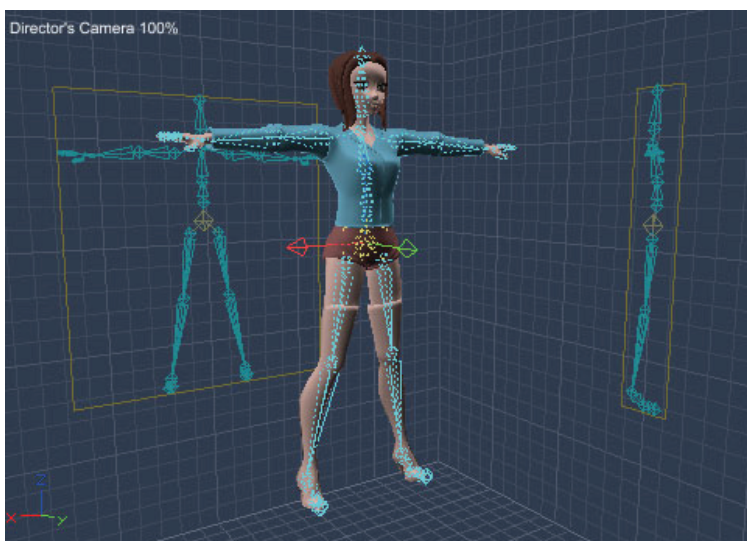
The Bone Tool

You can easily insert a bone into a scene.

You can insert a bone using one of the following methods:

- Use the **Insert > Bone** command.

- Drag the Bone tool's icon to the scene.

- Select the Bone tool, and then click inside the scene.

- When you select the Bone tool you can do the following:

  - Click and drag in the scene to have the newly inserted bone follow the mouse.

  - If another bone is selected, the new bone is created as a child of the selected bone. This allows you to swiftly create skeletons.

  - If you click on a bone to move it around, use the Ctrl/Cmd key to constrain the movement to the Z axis instead of the X/Y axis. This makes it easier to move bones into position.

  - When you click to place the next bone, hold the Shift key while you drag to change its size instead of moving it.

TIP! Creating a skeleton with the Bone tool is easier when using the four view mode. See "Displaying Multiple Camera Views."

Selecting a Bone

A bone can be selected with the Selection, Scale, and Rotate tools, or with the Bone tool. When you are working with the Bone tool, clicking a bone selects the existing bone rather than creating a new bone. This allows you to quickly create a new limb on a skeleton without changing tools.

Bones are represented by different colors. All these colors can be changed in the Preferences dialog, under 3D View Colors:

- By default, the Wireframe of a bone is drawn with the Primary Wireframe color.

- When a bone is selected, it is drawn with the Selected Object color (yellow, by default).

- The children of a selected bone are drawn with the Secondary Wireframe color (cyan by default).

Changing the Size of a Bone

One way to change the size of bones while you are creating them is to press the Shift key while dragging to position your bones. This scales the current bone to the desired size. You can also change the size of the selected bone or joint in the **Properties tray: Motion tab: Transform panel**. Enter a new value in the Overall field. Check the Propagate Scaling option to apply the new value to all child bones beneath the currently selected bone. The Propagate Scaling option also works when you set a new scale and then add additional child bones.
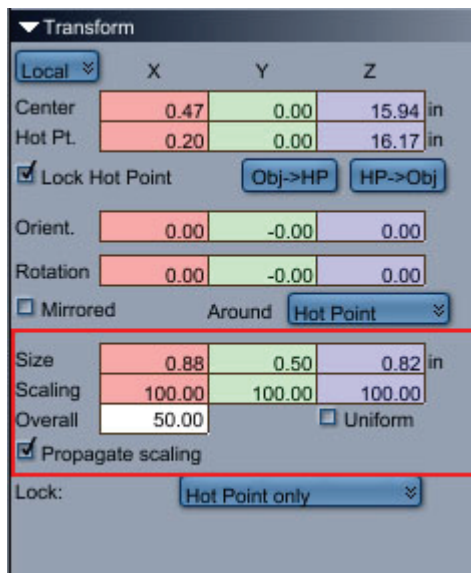
It is possible to rescale a bone with the Scale tool. But once a bone has children, the distance between then changes and the skeleton doesn't match the object it is going to control.

To adjust the display size of a bone without affecting the rest of the skeleton:

1. Select a bone that does not have any children.

2. Open the **Properties tray: Effects tab** and under Bone, enter a new **Display Size** value.
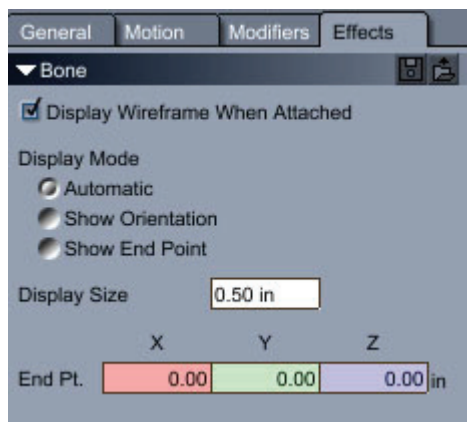
Working with Joints

Once you have created your skeleton, you usually have to move the joints to put them in the right position. In doing that, if you do not rotate the local space of those joints and you may have difficulties setting up constraints, for instance.

Orienting Joints Automatically

A tool can orient the joints of a skeleton for you automatically.

Choose the **Animation > Orient Joint** command, and check or uncheck the following options:
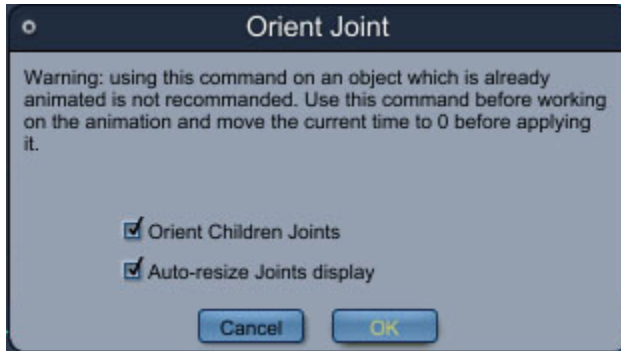
- **Orient Children Joints**: Check this option to orient joints automatically.

- **Auto-resize Joints Display**: When enabled, bones are automatically resized according to the starting and ending points of the mouse position. When disabled, bones are created at a default size and you can resize later.

Orienting a Joint Manually

If you need to manually adjust the orientation of one or more joint, you can use the Lock Children mode. This mode lets you rotate a joint and adjust its local space, without moving the children. See "Arranging Objects Without Modifying its Children."

How to manually orient one joint:

1. Select the joint that you want to align.

2. Click the Lock Child Objects icon in the main toolbar area.

3. Select the Rotation tool. This tool shows a small local space (three different colored lines meeting at a center point) in the center of the rotation sphere.

4. Rotate this local space to align it the way you want.
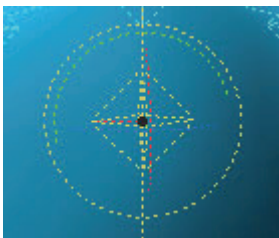
Working with Skeletons

You can attach a skeleton to a vertex object so that the movements of the bones change the geometry of the object. A vertex object attached to a skeleton is referred to as a **skinned object**.

When an object is attached to a skeleton, you'll notice the following:

- You cannot directly modify the position, scaling, or orientation of the object. Instead, you need to change the position, scaling, or orientation of the root of the skeleton.

- The position, scaling, and orientation of each bone defines the **reference position** of the bone. A bone can be sent back to its reference position at anytime, even after the skeleton has been detached from the object.

- The position of any single vertex of the object depends upon the position of several bones of the skeleton. The bones affecting the position of that vertex **influence** it by a percentage. The sum of the influences of the bones of the skeleton over a single vertex is always 100%. See "Editing the Bones Influences."

Attaching a Skeleton

The **Animation > Attach Skeleton** command attaches a skeleton to one or more vertex objects. You cannot use this command on a skeleton that is already attached to an object.

As the geometry and position of the attached objects depends only on the position of their skeleton, the Attach command sets the animation method of those objects to still.

NOTE! Since bones can only deform vertex objects, other objects are automatically converted to vertex objects by the attachment process.

To attach a skeleton:

1. Create a skeleton and move it inside the object (or objects). Select one bone of the skeleton.

2. Shift+click the objects in the 3D view, or in the hierarchy, to select one or more vertex objects or a group containing vertex objects. If you select a different type of object, it will be converted to a vertex object when you attach the bones.

3. Choose **Animation > Attach Skeleton**. The Attach Skeleton dialog appears.

4. If desired, check the **Only attach selected objects** option.

   o When checked, only the selected vertex objects will attach to the skeleton. Descendants of the selected object will not attach.

   o When not checked, the skeleton will affect all of the vertex objects that are descendants of the selected objects or groups.
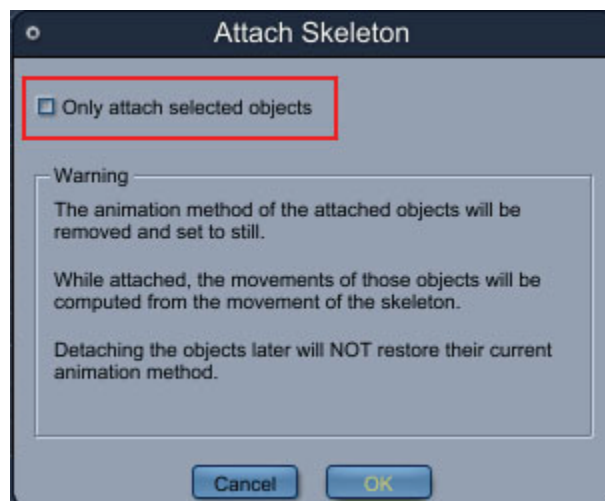
Detaching a Skeleton

If you want to modify the geometry of your vertex object, you need to detach its skeleton.

The **Animation > Detach Skeleton** command releases an object from the influence of its skeleton. It also restores its geometry the state it was in before you applied the Attach Skeleton command.

To detach a skeleton:

1. Select one or more skinned objects or a group containing skinned objects (Shift+ click the objects in the 3D view or in the hierarchy).

2. Choose **Animation > Detach Skeleton**, and click the checkbox if desired. You can choose to detach only the selected objects.

Restoring Bones to Their Original Positions

You can set the position and orientation of one or more bones to the values they had when the bones were attached.

To restore bones to their original position:

1. In the Assemble Room, select the object to which the skeleton is attached.

2. Choose **Animation > Send Bones to Reference Position**. The bones return to their original values and the object moves accordingly.

Inverse Kinematics

Inverse Kinematics is a specialized modifier that is applied to linked objects. Inverse Kinematics provides the versatility of child-to-parent control. It reduces the time it takes to create realistic animations.

Normally, movement is transmitted downward from parent to child in the hierarchy. When you link a child to a parent object and apply inverse kinematics, movement can also propagate up the hierarchy:from the child to the parent. This means that when you move a child object, the parent follows. Motion cannot propagate from a linked child to its parent without Inverse Kinematics. Refer to "Linking Objects" for information on link properties and constraints.

The movement is affected by the type of link property and constraint you apply, for example, a Lock, Slider, or 2D link can constrain the movement of the child object in relation to the parent.

If you use an arm as an example, linking the objects that make up the arm in a hierarchical structure and placing the hot points at the joints, you can rotate the arm at the shoulder or elbow, and everything below that joint (toward the hand) moves accordingly.

To take a simple example, assume that an arm is made of three parts: shoulder, elbow, and lower arm. With Inverse Kinematics, you can move the lower arm, which in turn forces the elbow and shoulder to rotate and move to accommodate.
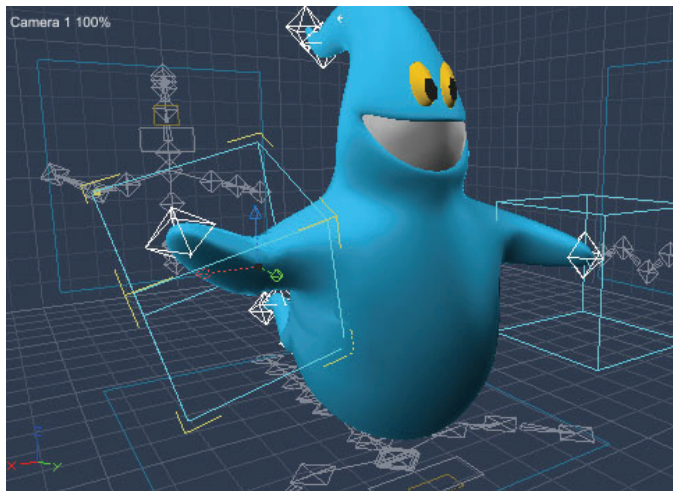
Before applying the Inverse Kinematics modifier to an articulated object, you must link the objects that make up the model and define properties and constraints for the movement of each child in relation to its parent. These constraints should describe the desired range of motion for each part.

Refer to "Linking Objects" for information on linking one object to another. Refer to "Applying Constraint Properties" for information on setting constraints.
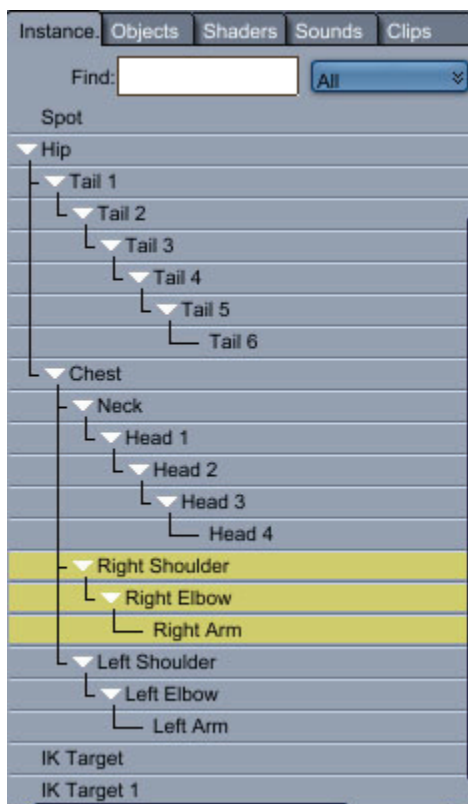
NOTE! You must apply Inverse Kinematics to the last child object in the link chain. You can also apply it to other children higher up the link chain. Then if you start a movement farther up the chain, objects higher up the chain move as well.

Tracking an Object

When animating a character with inverse kinematics, the object with the IK behavior can track another object (called the target). For example, you can make a foot track a **foot step** object. This way, as you move the character, the inverse kinematics will keep the foot at the position desired for the foot step (generally by bending or extending the knee). If the target is not reachable, the IK chain (the leg) extends itself to the maximum possible and points toward the target. See "IK Chains."

IK Terminators

When you use the Inverse kinematics behavior, the propagation of the movement stops by default on the last parent that has a constraint. Using the IK terminator makes it possible to stop that propagation earlier. For example, if you have a skeleton with constraints on every bone and an Inverse kinematics behavior on the hand, moving the hand will also move the spine. But if you apply the IK terminator to the shoulder bone, moving the hand will only move the arm.

IK Chains

When animating a character, a common movement is to make the arm reach for an object.

Since articulations in the human body only allow rotations, positioning the arm using forward kinematics involves first rotating the shoulder, then the elbow, and finally the hand (and generally starting over because the hand is not really where it was supposed to be).

On the other hand, using inverse kinematics (IK) automatically calculates the position of the elbow and the shoulder so that the hand is as close as possible to where it is supposed to be. The hand and the joints it affects when using IK constitute what is called an IK chain.

Characters typically use inverse kinematics to position intermediate joints when you select and move or rotate a body part. An inverse kinematic chain allows you to position all parts of a leg from the hip down when you move a foot.

Creating an Inverse Kinematics chain with its target is very easy with the IK Chain tool, located in the left toolbar area.

For example, you would use the following steps to create an IK chain for the left leg of a character:

1. Use the Selection tool and click on your character's left hip bone.

2. Click the IK Chain tool, and then and click on the left foot bone. Carrara automatically creates an IK chain for the left leg, and inserts in the scene an IK target object that the left foot will follow.

   The IK tool automatically adds:

   - A ball joint constraint on each bone of the chain that didn't have a constraint. Ball joint constraints allow you to rotate joints, but not move them (see "Arranging Objects with Constraints" and "Types of Constraints").

   - An IK terminator behavior (see "IK Terminators") on the bone at the beginning of the chain (the left hip in the example).

   - An IK target object. This is an object similar to the target helper, but with a different shape. See "Tracking an Object."

- • An Inverse Kinematics behavior (see "Inverse Kinematics") on the bone at the end of the chain (the left foot) set up to follow the IK target object that was created.

Representation of IK Chains

In the 3D view, IK chains are represented by a line between the first and the last bone of the chain.
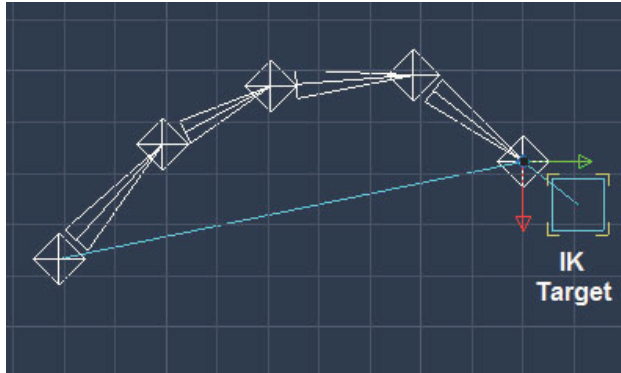
Like the other modifier wireframes, the display line can be hidden, displayed only when the object is selected, or always displayed. To toggle between those states, select the bone with the Inverse Kinematics behavior, select the Modifiers tab in the Properties tray, and then click on the direct manipulation icon of the IK behavior.
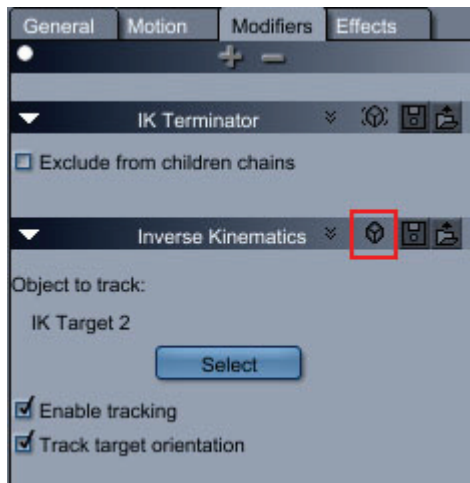
Limitations of IK Chains

Two IK chains can share one bone, but only when they follow each other. For example it is possible to have one IK chain consisting of the hip, knee, ankle, and another chain between the ankle, foot, and toes as shown in the following image.
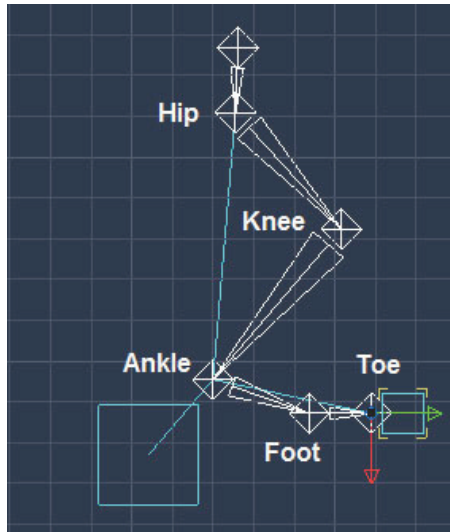
Any other case of sharing a bone between two IK chains will result in the unwanted motion of one chain when moving the other. For example, the IK chains for the legs should not join at the pelvis. One should end on the left hip, while the other ends on the right hip.

IK Target

When a character is walking, the feet are supposed to stay on the ground. But moving the character up and down will move the feet above or below the ground. If the character has an IK chain on each leg, adding a target position for each foot will ensure that moving the character down bends the knees and keeps the feet on the ground. Any object can be used as target, but in most case one that is not rendered (like the target helper) is more suitable.

To specify the IK target:

1. Select the object with the IK behavior (such as the foot).

2. Click on the **Properties tray: Modifiers tab**.

3. Click on the **Select** button in the **IK settings** to choose the target. The target can be any object in the scene, but generally the IK target object works best.

4. To turn tracking on or off, click on **Enable tracking**. This property can be animated so the IK chain doesn't have to track the target during the entire animation.
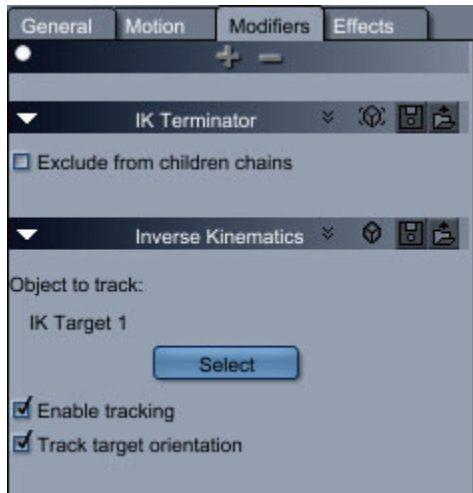
Editing the Bones Influences

When you attach a skeleton to a vertex object, Carrara automatically computes how the different bones of the skeleton affect the position of the different vertices of the object. Additionally, Carrara allows you to adjust the calculated influences to fine-tune the results.

After you attach a skeleton and a vertex object, you can switch to the Vertex modeler. The skinned object is displayed in the Vertex modeler with the deformations produced by the current position of its skeleton. The tools and commands that allow you to modify an un-skinned mesh are disabled, and you can only change the influences of the bones on the vertices.

You have two ways to edit the bones influences:

- Use the Weight Painting tool to quickly assign or remove influence for some of the points.

- Use the slider to control precisely the influence at the level of the vertices.

Using the Weight Painting Tool

To use the Weight Painting tool:

1. With your boned object selected, switch to the Model Room.

2. Set the Edit Mode in the Model Room to Animation Mode.

3. Select the Weight Painting tool.

4. Select a bone, using one of the following methods:

   - Choose a bone from the Bone menu in the Properties tray (in the Weight Painting section).

   - Press the Control key while you select it directly in the view.

5. Choose the following settings as needed for the Weight Painting tool:

   - **Mode**: Choose **Add** to increase the influence in areas, or **Subtract** to decrease the influence.

- **Brush Size**: Sets the radius of the brush. Increase or decrease the slider as needed.

- **Strength**: Decrease the setting for more subtle changes, or increase it for stronger changes.

- **Smoothness**: Controls the amount of falloff in the brush.

- **3D Sphere Brush**: Check this option to make the brush aware of the depth. With this option, you can paint locally on the mesh without modifying influences somewhere behind this area.

6. The influence for the selected bone is represented by a color range: yellow for strong, orange for medium and grey for no influence. Use the brush directly on your model to modify the influences like a paint tool. You will see the influence and the position of the model being updated in real time.
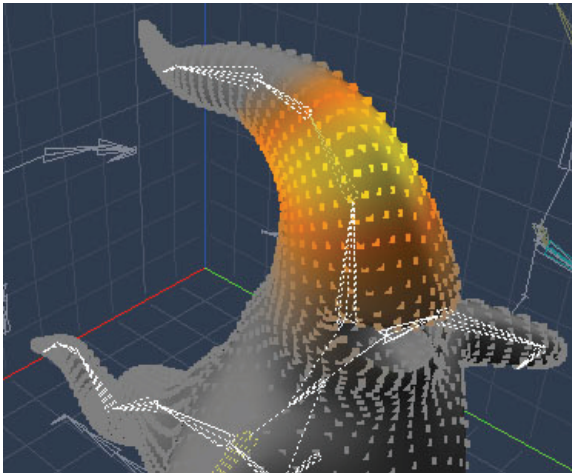
Weight Painting on DAZ Studio and Poser Figures

You can also use the Weight Painting tool to edit bone influences on DAZ Studio/Poser figures. For example, vertices can be selected by morph area and then you can edit weights for the selected body part with the weight painting tool.
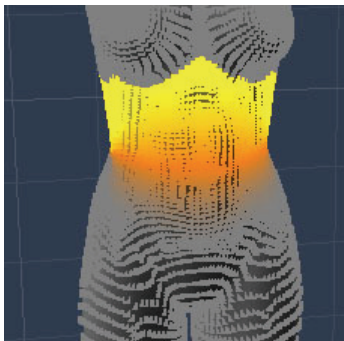
To use the weight painting tool on a DAZ Studio/Poser figure:

1. Load the desired figure into a Carrara scene.

2. Select the figure in the Assemble room, and switch to the Model room.

3. Switch to Animation mode.

4. Choose **Select > Select By > Morph Area**. The Choose a Morph Area dialog lists all morph areas in the selected figure. Each of these morph areas corresponds with one of the bones in the DAZ or Poser figure.

5. Select the morph area/body part that you want to influence and choose OK. The vertices associated with that part are selected and highlighted.

6. Select the Weight Painting tool.

The Weight Painting options appear in the Properties tray.
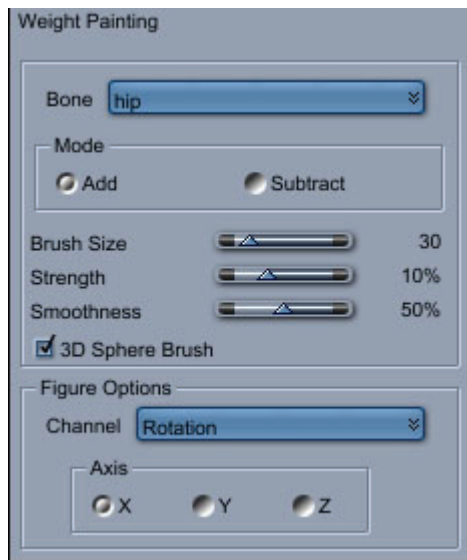
7. Select the bone that you want to paint from the **Bone** menu.

8. Choose the following settings as needed for the Weight Painting tool:

    - **Mode**: Choose **Add** to increase the influence in areas, or **Subtract** to decrease the influence.

    - **Brush Size**: Sets the radius of the brush. Increase or decrease the slider as needed.

    - **Strength**: Decrease the setting for more subtle changes, or increase it for stronger changes.

    - **Smoothness**: Controls the amount of falloff in the brush.

    - **3D Sphere Brush**: Check this option to make the brush aware of the depth. With this option, you can paint locally on the mesh without modifying influences somewhere behind this area

9. In the Figure Options section, select the channel that you want to influence. The availability of these options depends on the bone you select:

    - **Rotation**: Allows you to paint influences that are applied when the selected bone is rotated.

    - **Smooth Scale**: Allows you to paint influences that are applied when the selected bone is scaled.

    - **Negative Bulge**: TBD

- **Positive Bulge**: TBD

- In the Axis section, select the axis of Rotation, Negative Bulge, or Positive Bulge.

10. Paint on the model with the weight painting tool to modify the existing influence as desired.

Saving Modified Weight Maps

You can save a modified weight map so that you can later reuse it in another scene.

To save a modified weight map:

1. After your weight map is modified, choose the Bones tab in the Model room.

2. In the Weight Map section, click the Save button. The **Save Weight Map As** dialog appears.

3. Locate the folder that will store your modified weight map.

4. Assign a name to your file and click **Save**. Weight maps are saved with a .cwm extension.

Loading Modified Weight Maps

You can save a modified weight map so that you can later reuse it in another scene.

To save a modified weight map:

1. Select the figure that you want to modify.

2. Switch to the Model room, and select the Bones tab.

3. From the Weight Map panel, click the **Load** button. The Load Weight Map dialog appears.

4. Use the Look in menu to select the folder that contains the weight map you want to load.

5. Highlight the file and click **Open**. The new weight maps are added to the figure.

Editing the Influences Over One or Many Vertices

When you are in Animation Mode in the modeler, the Bones tab of the Properties tray displays information about the influence of each bone that affects one or more selected vertices. Each bone's influence is represented by a slider with the name of the bone.
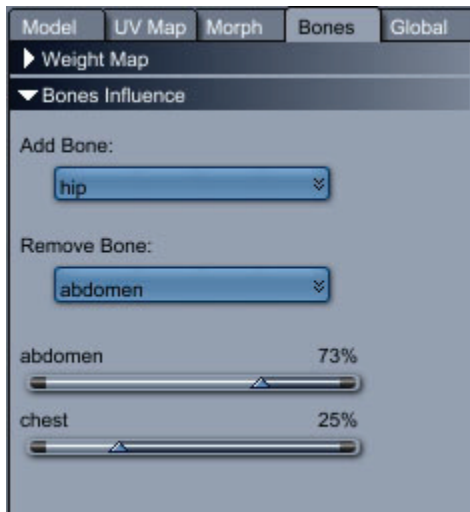
After you select one or more vertices in the Model room, switch to the Bones tab in Animation Mode, and use the controls as follows:

- **Add Bone**: To add a new bone's influence, select a bone from the Add Bone pop-up menu. A new slider will be added to the list.

- **Remove Bone**: To remove a bone's influence, select the bone from the Remove Bone pop-up menu. Note that you will not be able to remove the last influence.

- **Bone sliders**: To change the influence of a bone, select its slider and change its value. The position of the vertices changes as you modify the value. Also, the sliders of the other bones will be automatically adjusted to keep the sum of the influences on each vertex to 100%.

Examples of Influence

The following screen shots show the skinning of an arm represented by a cylinder. The shoulder bone is on the left of the images, the elbow in the middle and the wrist on the right.

The first image shows the arm and its skeleton just after they were attached (horizontal skeleton). In the other images, the elbow is rotated by 90 degrees to bend the arm and each image shows how changing the influences affect the skinning.

The second image shows the skinning with the default influences on each vertex while the third, fourth, and fifth images show extreme influence values on each vertex.
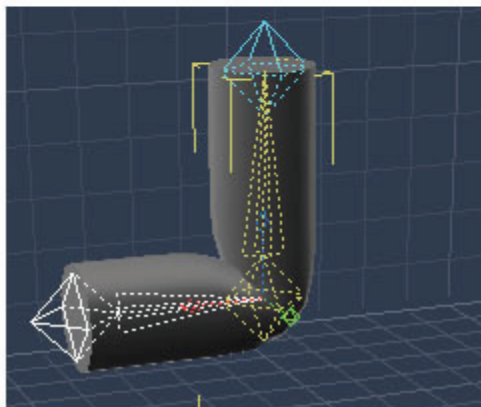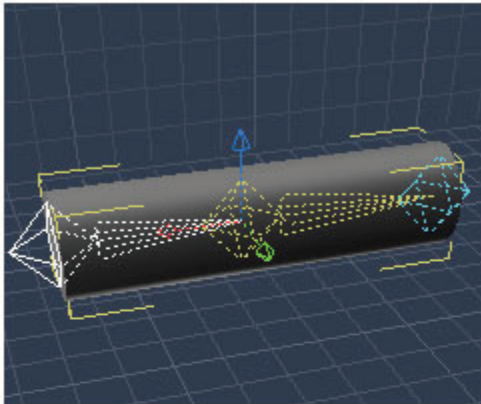
The next image shows the shoulder's influence set at 100% for each vertex. Note that the arm stays horizontal. When the elbow's influence is set at 100% for each vertex, notice that the arm follows the elbow and becomes vertical.
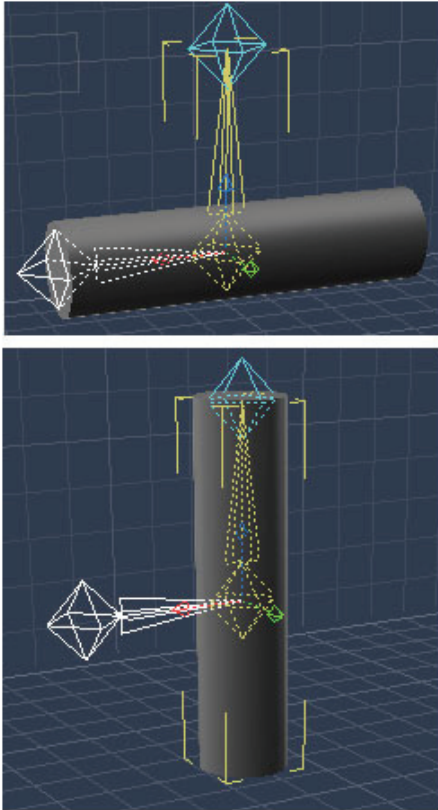
When the shoulder and elbow influences are set at 50% for each vertex, the arm takes an average position.
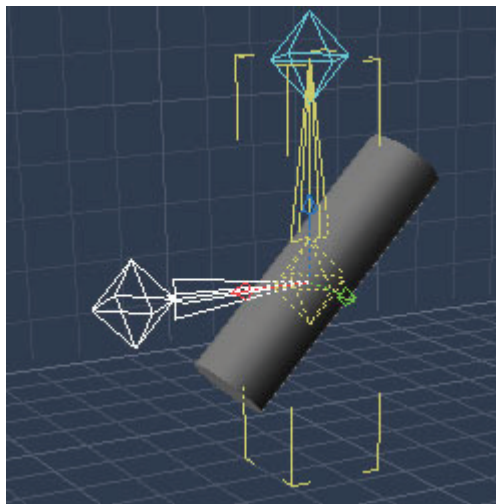
NOTE! After you attach a skeleton to an object, some commands become unavailable such as copy, paste, duplicate, flip, and group. When the current selection contains an item on which a command cannot be performed, that command becomes unavailable. If you must perform the command on the item, you must first detach the object from its skeleton.

Tips and Tricks

- You can convert any object to a vertex object by double-clicking it in the Assemble room and using the **Edit > Convert to Other Modeler** command.

- The geometry of the object you want to skin must have enough vertices. To increase the number of vertices, use the Subdivide command of the Vertex modeler.

- You can put a skeleton in the Browser by dragging its root from the Hierarchy section of the Sequencer to the object's tab in the Browser.

- Creating a skeleton is easier when using the four-views mode and inserting new bones in the top, left, and front views.

- To speed up the interaction with a complex skinned model, open the Interactive Renderer settings, and set the silhouettes to Boxes Only.

- To resize a skinned object, resize the root of its skeleton.

- To change the geometry of a skinned object, select the object, detach it, edit it, select a bone of its former skeleton, and send all the bones to their reference position, then select the object and choose the **Animation> Attach Skeleton** command.

- To change the hierarchy of the bones, use the Hierarchy tray the way you would with any other object or group.

Dynamic Hair and Fur (Standard, Pro)

One of the most exciting new features of Carrara 6 is the addition of dynamic hair and fur: hair that moves realistically when you animate a feature, and that responds to forces and collisions with other objects in your scene.

For Carrara 3D Express users, dynamic hair and fur can be loaded into Carrara 3D Express and rendered, but cannot be modified.

About Dynamic Hair

Dynamic hair is different than geometry-based hair in that it is strand-based. There are two types of hair strands: guide hairs and generated hairs. Guide hairs help you style and shape the hair. Generated hairs are the hairs that actually render in your scene. When you move a guide hair, the generated hairs move in the same direction.

Another key difference between dynamic hair and geometry-based hair is that you won't see a true representation of dynamic hair until after it is rendered. During render time, Carrara examines all of the properties that you assign to the hair: thickness, length, color, hair scale, and much more. All of the properties are calculated out for the final render. As a result, you may need to perform an occasional render while you are styling the hair to ensure that the final result appears the way you intend it to appear.

Adding Hair to an Object

The Hair tool, located in the main toolbar, allows you to add hair to an object. Simply drag the Hair tool and drop it on to an object in your scene.

There are two ways that you can add hair to objects, depending on whether or not the geometry has UV maps assigned.

- Geometry-based objects without UV maps present some challenges in regards to hair placement. If you change the geometry of an object that does not have UV maps, whether by adding or removing polygons or turning subdivision surfaces on or off, the hair will move to unpredictable places on the object.

- When objects are UV mapped, there is less impact when you make changes to hair that you have already placed. For example, if you subdivide the surfaces of a UV-mapped object you may also need to modify the UV maps. However, the hair still appears pretty much where you expect it to appear.

To add hair to an object:

1. Use the Content Browser to load your figure or object into the Carrara scene.

2. From the main toolbar, select the Hair tool. Drag and drop the icon onto the object to which you want to add hair. The Hair First Setup dialog appears.

3. In the **Hair Placement** section, select one of the following options:

- Choose **UV Based** if your model is UV-mapped. Most objects that are compatible with DAZ Studio or Poser have UV Mapping assigned to the geometry, making this the most likely choice when importing DAZ content. If your object comes with texture maps, it is probably UV mapped.

- Choose **Geometry Based** if your object does not have any UV mapping assigned to it. Typical cases for this might be geometry primitives.

4. In the **Density** section, choose one of the following options:

- Choose **Guide Based** if you want to control the density of the hair by increasing or decreasing the settings associated with the hair guides. This is the most common option. This option offers more customizable control than the Texture Map option.

- Choose **Texture Map** if your object has UV maps assigned. You will be able to use the UVs and material assignments to control density maps.

- If you choose the **Texture Map** option, click the **Manage Density Maps** button to create and choose the size of the density map that you want to use. Instructions on how to use this dialog are described in "Managing Density Maps."

5. Check or uncheck the **Start with empty grow region option** as desired:

   - When unchecked, hair will grow from all polygons in the object.

   - When checked, all polygons are left unassigned so that you can select specific polygons that will generate hair.

6. Choose **OK** to complete the hair setup process.

   You are immediately transferred to the Model Room, where you can set hair properties in the Tool, General, and Dynamics tabs. The Tool tab is automatically selected for you in the right side of the interface. You'll also notice a special Hair Toolbar beneath the menu bar.
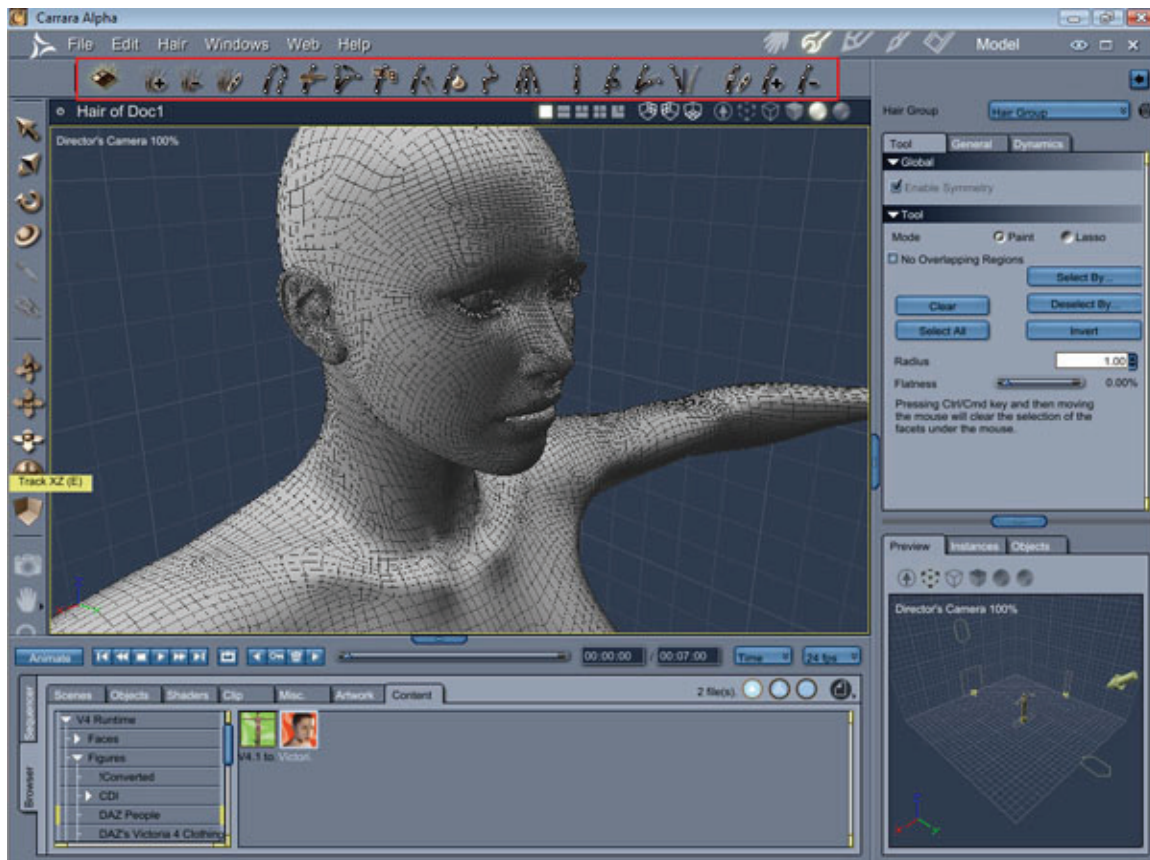
If your object contains hair, when you look closer at the hair you'll notice the following:

- When polygons cannot grow hair they are colored grey when in the hair room. When polygons can grow hair they are highlighted a different color.

- When you zoom into the hair on the object, notice that some strands are colored and others are white with a dot on the end. The white/circled hairs are the guide hairs, which allow you later to style and comb the hairs more realistically.

Managing Density Maps

When you initially set up your hair object as described in "Adding Hair to an Object," you are asked if you want to manage the density of the hair with guides or with density maps. If you click the **Manage Density Maps** button you see the Manage Density Maps dialog shown in the following figure.

Density is based on the type of UV mapping that is set up on your objects. It is important to note that you can use a single global map only if the UV coordinates on your object do not overlap. Polygons share UVs when the texture is mapped in two different places in your mesh, which can cause overlapping UVs. In cases such as this, there are a few ways to address it:

- Create more than one density map for your object, assigning different density maps to the shading domains in your object. This will only work as long as the UV coordinates do not overlap in a single shading domain.

- Use the Geometry-based option in lieu of the UV map option.

- Edit the UV mapping in the UV editor so that you no longer have overlapping UVs. Then apply the hair to the mesh.

The density of the hair determines the number of hairs that will be generated for each guide hair. When you initially create a hair object, the density is set to the maximum level (or 100%).
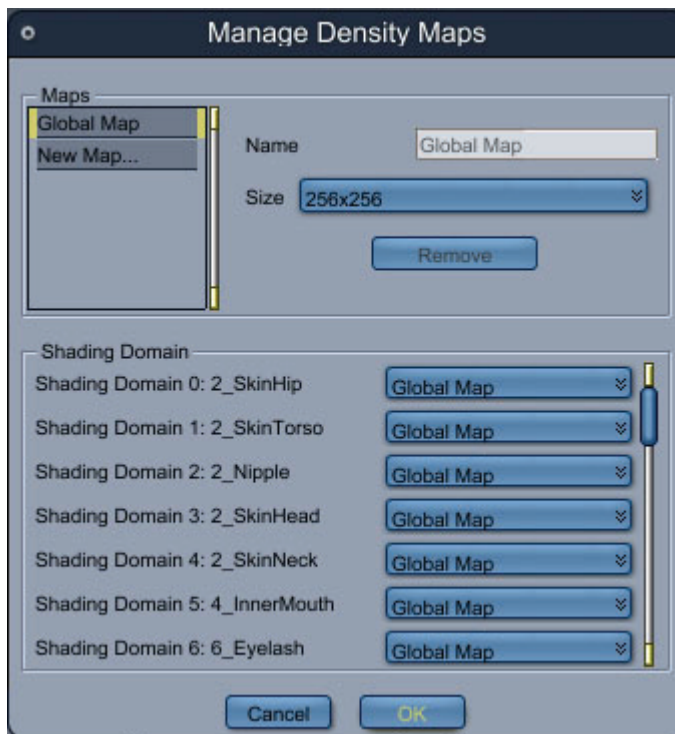
To manage density maps:

1. The Maps section listed in the upper-left corner of the dialog initially lists two maps: a global map and a new map.

    - Use the global map if you want to add hair to the entire object. By default, global maps use an image map size of 256 by 256 pixels. Use the Size drop-down menu to select any size (in a power of 2) from 64 by 64 pixels to 2048 by 2048 pixels. Larger maps provide finer densities.

- You can also create additional maps with different resolutions, and apply them to other shading domains in your object. This allows you to vary the density of the hair from one part of your object to another. To define a new map, click the **New Map** button in the **Maps** panel. A new button named "New Map" appears in the list. Assign a name to the map in the Name field, and select a map size from the drop-down menu.

2. To assign your custom map to other shading domains in your object, scroll through the Shading Domain list until you find the material that you want to change. Click the button to view a list of the available density maps, and select the one you want to apply.
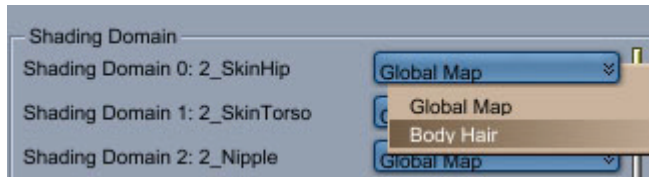
3. When you are done with your choices, click OK to exit the Manage Density Maps dialog. You return to the Hair First Setup dialog described in "Adding Hair to an Object."

Creating Hair Groups and Grow Regions

Objects need at least one hair group to grow hair. Carrara creates one hair group on an object or group when you initially add hair to an object.

You can create more than one hair group in a single object. In fact, multiple hair groups can prove very useful you need to create different types of hair on the same figure or object. For example, you can create separate hair groups for the skull, eyebrows, eyelashes, and facial hair of a male character. After you create the groups and assign polygons to them, you can assign different properties to them. The hair on the skull can be short and thick. The eyebrow hair can be coarser than the hair on the head. The eyelash hairs are shorter and much less dense. The facial hair could become a full-grown beard, or the stubble of someone that is in need of a shave.

You can also create a hair style that uses multiple hair groups. For example, you can create a man's hair style that has a hair group that creates short hair at the neck and the back of the head. A second hair group can create longer hair that is styled to wave or curl the hair.

Multiple hair groups can also prove useful when styling hair in layers. For example, you can create a hair group that includes a small number of polygon rows near the nape of the neck, and style them properly. Then create a new group that adds a few more rows of polygons immediately above the first group, and style them. Continue in this manner until all of the polygons in the hair are assigned and styled.

You create new hair groups in the Model Room. Initially, your object may only consist of one hair group and all of the polygons in that group may be assigned to that single group.
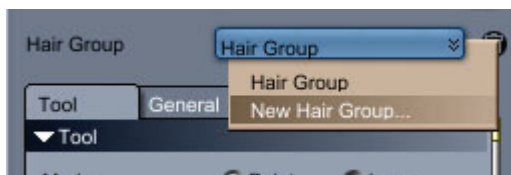
The following steps show you how to remove all polygons from an existing hair group and create additional hair groups:

To create hair groups:

1. Select the Hair object that you want to change. Polygons that are assigned to hair groups will display in color. Unassigned polygons will be gray.

2. If the area that you want to include in the new hair group is already assigned to an existing hair group, you'll need to remove the assigned polygons:

- Select the **Grow Region tool** from the Hair Toolbar.

- Click the **Clear** button in the Tool tab to remove all polygons from the current hair group, or press the **Ctrl/Cmd** key while you paint with the Grow Region tool to remove selected polygons.

3. Click the **Hair Group** button above the Tool tab. Existing hair groups are displayed in the drop-down list.

4. To create a new hair group, select **New Hair Group**. The Create New Group dialog appears.

5. Enter a name for the new hair group and press **OK**.

6. Use the **Grow Region tool** to select the polygons that you want to assign to the new group.

7. Repeat Steps 3 through 6 for each new group you want to create. As you continue to create new groups, the polygons that are already assigned will display in one color, and the polygons that you are assigning to the new group display in another color. Unassigned polygons remain gray.

NOTE! Additional hair group options appear in the General tab in the Model Room. These options allow you to rename the hair and toggle its visibility.

Adding Hair to Selected Shading Domains

Most DAZ figures (as well as other Poser-compatible figures) use multiple textures and multiple material zones. Most likely, you'll want to confine hair growth to the head of a human figure, rather than applying it to the entire body.

If your object includes more than one shading domain, you can select specific shading domains on which to grow hair. You use the Grow Region tool (the first tool in the Hair toolbar) to accomplish this.

NOTE! The shading domains that you select vary, depending on the material zones that are present in your figure. DAZ's Victoria 3, for example, includes a material zone named Skin Scalp, which is ideally suited for this purpose. DAZ's Victoria 4 includes two material zones: SkinHead (which includes the ears and the back of the head), and SkinFace (which includes the front half of the head). Selecting one or both of these material zones will require clearing some of polygons from the shading domains, using the Grow Region tool.

To select a shading domain on which to grow hair:

1. Choose the **Grow Region tool** from the Hair toolbar. Its properties appear in the **Properties tray: Tool tab**.

2. If your character is displayed entirely in color, it means that the entire figure is set to grow hair. Gray areas indicate where no hair growth occurs. To remove the hair growth areas from the entire figure. click the **Clear** button in the **Properties tray: Tool tab**.

3. Click the **Select By** button in the Tool tab. The Select Shading Domain dialog appears.

4. Locate the shading domain you want to select (such as 2_SkinHead, as used on DAZ's Victoria 4.x figure). The selected shading domain is highlighted by a yellow bar.
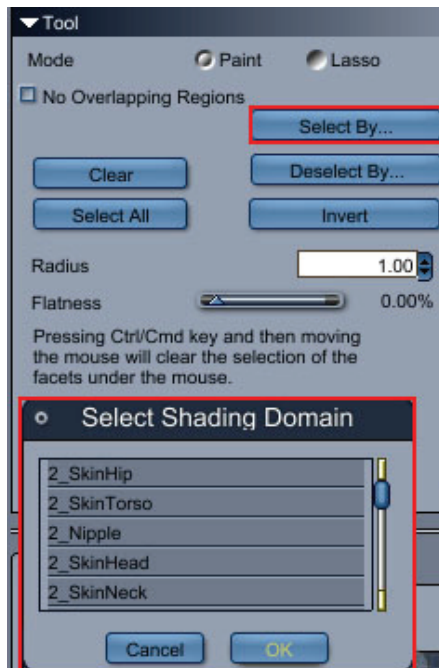
5. Click OK to assign the shading domain.

6. Repeat Steps 3 through 5 for each additional shading domain that will grow hair.

Painting and Selecting Grow Regions

If you do not want all polygons in the hair group to grow hair, you can use the Grow Region tool in the hair toolbar to add or remove specific polygons.

The Grow Region tool allows you select or deselect the polygons that will grow hair on your object. The Global pane of this tab displays the **Enable Symmetry** option, which is discussed in more detail in "Symmetrical Hair Styling."
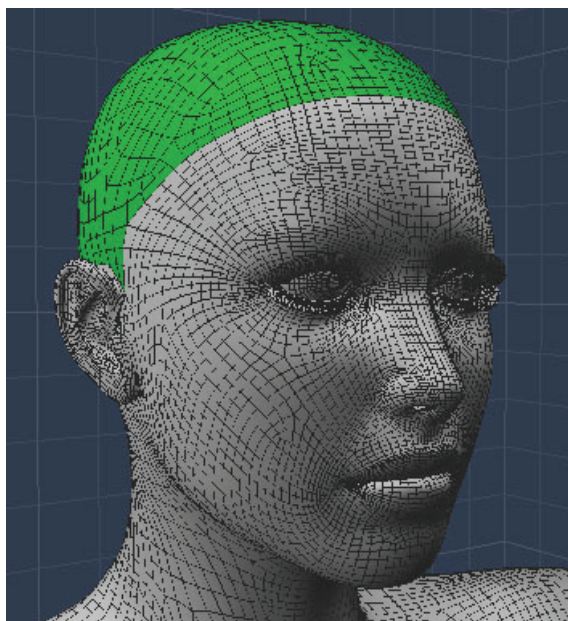
The options for the Grow Region tool appear in the **Properties tray: Tool tab** when the Grow Region tool is selected.
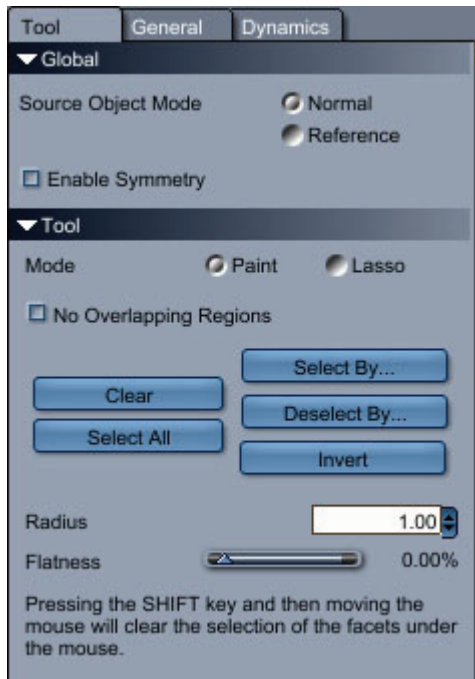
The Tool tab settings for the Grow Region tool provide the following functions:

- **Mode:** Use controls in this area to choose the mode in which polygons will be selected.

  - o **Paint:** When selected, you can use the mouse to paint the polygons that grow hair.

  - o **Lasso**: When selected, you can draw a selection area around the polygons to select them. Press the Cmd/Ctrl key while selecting to remove polygons.

  TIP! Press the Shift key while painting to remove polygons.

- **No Overlapping Regions**: Check this option to ensure that when two grow regions meet that the polygons in those regions do not overlap. If you decide to add polygons to a new growth group, they will be removed from other groups that the polygons are already assigned to. This ensures that polygons are only assigned to one grow region.

- **Clear:** Click this button to remove hair from all polygons in the object. If you are adding hair to a human figure, you'll probably want to click this button immediately after you add a hair object, and then proceed to select polygons on which to grow hair.

- **Select All:** Selects all of the polygons in the current object to add hair to them.

- **Invert:** Reverses the current polygon selection so that all polygons that were formerly selected are deselected, and all polygons that were formerly deselected are selected.

- **Select By:** Click this button to open the Select Shading Domain dialog. When in Select mode, hair will be added to the groups (or shading domains) that you select.

- **Deselect By**: Click this button to open the Select Shading Domain dialog. When in Deselect mode, hair will be removed from the groups (or shading domains) that you select.

- **Invert**: Inverts the current polygon selection to add the polygons that were formerly not assigned, and to remove the polygons that were formerly assigned.

- **Radius**: Adjusts the width of the Grow Region tool to select less or more polygons as you move the tool over the object.

- **Flatness**: Controls the shape of the brush. At 0% the brush has a spherical shape, while at 100% the brush is flattened into a circle.

TIP! You can also Cmd/Ctrl click and drag the tool to resize it.

To use the Grow Region tool:

1. Click the **Grow Region** tool in the Hair toolbar.

2. In the Global section, choose the desired source object mode:

   - **Normal**: In Normal mode, you edit the hair on the object in its skinned position.

   - **Reference**: In Reference mode, you edit the hair on the object in its reference position. This is useful when you want to use symmetry on a figure that is in a non-symmetrical pose.

3. To make symmetrical changes to the grow region, check the **Enable Symmetry** option. For further information on enabling this option, see "Symmetrical Hair Styling".

4. Choose one of the painting modes:

   - Choose **Paint** mode to assign polygons by painting with a brush.

   - Choose **Lasso** if you want to select polygons with a selection marquee.

5. Check the **No Overlapping Regions** option, if desired, to add polygons that you paint to the current growth region and remove them from previously assigned regions.

6. Set the size and shape of the brush. You can press the Ctrl/Cmd key while using the tool to resize the brush.

7. Polygons that grow hair are colored green. To add polygons to the grow region, do any of the following:

   - If your object has multiple shading domains (such as different material zones on a figure), click the **Select By** button and select the shading domain that you want to include. Repeat for each additional shading domain that you want to add.

   - To paint your selection, increase or decrease the **Radius** of the brush to the desired size. Adjust the **Flatness** setting, if desired, to achieve the desired shape. Then paint on the object to select the polygons.

   - To select all polygons in the object, click the **Select All** button.

8. Unassigned polygons are colored gray. To remove polygons from the grow region, do any of the following:

   - If your object has multiple shading domains, click the **Deselect By** button and select the shading domain that you want to remove. Repeat for each additional selection.

   - To deselect by painting, press the **Shift** key while you paint over polygons you want to remove.

Assigning General Hair Properties

After you create your hair groups, the next step in the process is to assign some general properties to the hair, so that you can obtain the best results with the styling tools.

The Model Room contains three tabs that pertain to hair: the Tool tab (which provides options for each of the tools in the Hair Toolbar as they are selected), the General tab, and the Dynamics tab. We'll start with the General tab first, since it is the next logical place to visit.

The General tab provides options and controls that affect all hair groups. You can control various aspects of the hair, including length, thickness, hair width (or scaling), rendering quality, and more. You can also increase or decrease the number of segments in the hair to make it less "stiff." These, and more properties, are described below.

Setting Hair Length, Thickness, and Rendering Quality

Options in the General section allow you to adjust the length, thickness, scaling, and rendering quality of the hair. When working with generated hairs, you can use controls in the General section to hide them completely, or set a percentage of generated hairs that are visible.

The options are displayed in the following figure and described below.
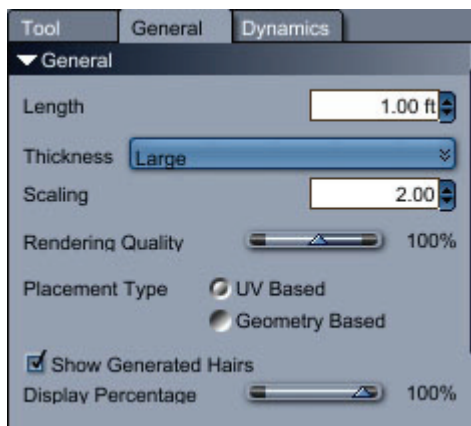
- **Length:** Adjust the Length slider to lengthen or shorten the hair. The default hair length is one foot.

- **Thickness:** Controls the thickness of each strand of hair. Choose **Thin, Medium, Large,** or **Very Large** from the selection menu. You see the results of this setting at render time. Medium and Large settings are most appropriate for commonly-used figures. Thin is most appropriate for small objects viewed at closeup range. Very Large thickness is suitable for grass and other grass-like plants.

- **Scaling:** Provides finer control of the thickness of each strand of hair. Larger values create coarser, thicker hairs. You see the results of this setting at render time.

- **Rendering Quality:** Controls the appearance of hair after it is rendered. Lower values create hair that is rough and has aliased edges. Higher values create smoother anti-aliased hair, but may take more resources to render.

- **Placement Type:** Determines whether hair placement is determined by UVs (**UV Based)** or by geometry (**Geometry Based).** You can use this option to switch between modes at any time after you create your hair object.

- **Show Generated Hairs:** Check this option to display an approximation of the generated hairs before you render. Uncheck this option to view only guide hairs.

- **Display Percentage:** With Show Generated Hairs (above) checked, you can dial in the percentage of hairs that are visible, from 0% to 100%.

At a minimum, you'll find it helpful to make the following adjustments to the hair before you begin to style it.

To set hair length and thickness:

1. To set hair length, use one of the following methods:

   - Choose **Hair > Set Length**, and enter the length in the Set Hair Length dialog. Click OK to apply the new setting.

   - In the **Properties tray: General tab: General pane**, adjust the **Length** value to the desired length.

2. Choose the **Thickness** that is most appropriate for the type of hair you are creating. Medium or Large is recommended for DAZ or similar figures. Use the Spot Render tool to preview the changes you make.

3. Adjust the **Scaling** setting, if necessary, to fine tune the width of the hair strands.

4. Set remaining options discussed earlier in this section as required.

Naming, Hiding, and Showing Hair Groups

Controls in the **Hair Group** section allow you to rename the current hair group or toggle its visibility. The properties in the hair group section are as follows:
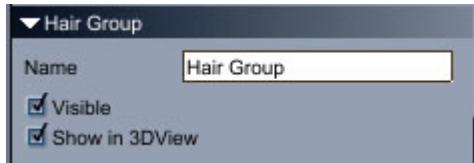
- **Name:** Enter a name for the current hair group in this field.

- **Visible:** Determines whether or not the hair group is visible in the scene viewport.

- **Show in 3D View:** When checked, displays the generated hairs for the current hair group in the 3D view. When unchecked, hides the generated hairs but keeps the guide hairs visible.

Controlling Hair Segments and Hair Count

The options in the **Hair (Per Hair Group)** section are also important to configure before you start styling the hair. These settings control the number of segments in each hair, and the number of hairs in each hair group. You can also choose to create short hairs or long hairs, and enable or disable automatic grouping.

Each hair group has a number of generated hairs associated with it. You can increase or decrease the number of generated hairs associated with each hair group to make the hair appear thicker or thinner. For example, if you are creating hair on a human figure, you may need more generated hairs on the head than on the eyelashes.

To make the hair easier to style:

1. Adjust the **Segments Per Hair** value as necessary for the length or amount of curl you want to achieve in your hair style. Shorter hair won't need as many segments as longer hair unless you want to make it very curly. For longer hair, increase the number of segments for easier styling and more fluid dynamic performance. A lower number of segments per hair makes the hair straight and stiff, while a higher number makes the hair flow, relax, or curl more naturally. Note, however, that a higher number of segments per hair can affect overall system performance.

2. Set the **Hair Count** to control the number of hairs that are generated for each hair group on the head. Increase the hair count to make the hair fuller.

3. Use the **Mode** setting to choose the type of hair that is being used on your object.

   - **Short Hairs/Fur:** Choose this option for short human hair, animal fur, and other similar objects.

   - **Long Hairs:** Choose this option for longer human or animal hair similar to a horse mane or tail).

Guide Hairs Options

Guide hairs are colored white, and have an adjustable point at their end. These guide hairs determine the direction, length, flow, and style of the hairs that are associated with it. You can also brush the guide hairs to control the direction that the generated hairs fall. Simple hair styles will need fewer guide hairs than more complex styles. Even simple styles will require more than one guide hair to style the hair more naturally.

When you add more guide hairs, you have more precise control over the shape of the hair, since the system won't have to do as much interpretation. When you make a style, you might find it easier if you turn off the generated hairs. Then use the set the guide hairs to somewhere between 200 and 1,000, and use at least 16 hair segments. This will help you create a shape for your hair style.

After your style is complete, reduce the number of guides to somewhere between 100 and 400, depending on the complexity of the style.

Settings in the **Guide Hairs (Per Hair Group)** section allow you to control the number of segments in each hair, and the number of guide hairs that appear in each hair group.
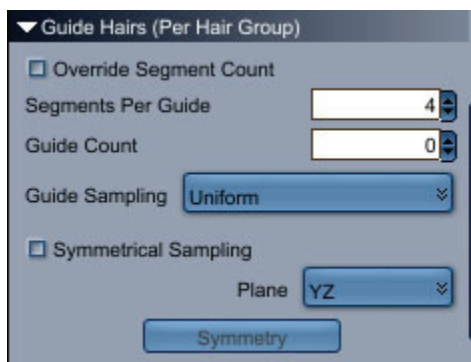
- **Override Segment Count:** By default, Carrara hair is created with four segments in each guide. Check this option to set a different value.

- **Segments Per Guide:** Increase or decrease the number of segments that each guide hair uses. A lower number of segments produces hair that is straighter. A higher value allows you to create hair that is more wavy. In addition, short straight hair needs fewer segments than longer hair.

NOTE! Many of the Hair tool settings will not appear to produce different results when you have a very low number of segments. When you increase the guide hair segments to 6 or more, many of the effects of the Hair tool settings will be more obvious.

- **Guide Count:** Increase or decrease the setting to create more guide hairs in your hair groups. This setting applies the number of guide hairs based on the Guide Sampling setting (below). If you need to vary the number of guide hairs, you can use the Add Guide Hairs tool in the Hair Toolbar.

- **Guide Sampling**: Determines the placement of the guide hairs in relation to the polygons beneath them.

- **Custom:** Choose this option when you add guides with the Add Guide Hair tool.

- **Polygon Center:** Places the http://www.nero.com/enu/downloads-nero8-trial.phphairs at the center of the polygon that generates the hair.

- **Uniform:** Places the guide hairs randomly on the surface, using uniform repetition.

- **Symmetrical Sampling**: Check this option if you want the hair to be symmetrical from one side to another.

- **Plane**: Choose the plane of symmetry (YZ, XZ, or XY) from the Plane popup menu.

- **Symmetry**: TBD

Parting and Grouping Hair

Many hair styles contain parts along one side or at their center. The Interpolation controls aid in creating parts in the hair and allow you to assign different properties to selected hair groups. They are most useful if you want a group of guide hairs to be interpolated together. They should only be used to fix particular issues when the automatic mode fails to interpolate the guides correctly.

The Interpolation settings control how the hair is generated from the guides. By default, the hairs are interpolated between all of the guides.

- **Auto Grouping Section:** Controls how hairs move when you style them.

  - **Enable:** When enabled, allows you to control which guides will be used for interpolation of the guide shape. If you turn on auto-grouping, you can control how the hair interpolates between the groups.

  - **Distance:** Restricts auto-grouping to the guides that are close enough to the distance setting. The Distance value sets the maximum distance from a hair to a guide, where the guide can affect the shape of the hair.

- o **Angle:** Restricts auto-grouping to guides that are within the angle setting. The Angle value sets the maximum angle from a hair to a guide. Guide hairs that have very different angles are not interpolated together.

- o **Distance Along**: Sets the distance along the length of the hair that the groups are interpolated.

- **Groups Section**: The Groups section allows you to create new interpolation groups that allow you to fine-tune the interpolation and correct problems with the auto grouping. We recommend that you use this feature only at the end of the process to correct problems that cannot be solved any other way.

  - o **New Group**: Click this button to open the Create Interpolation Group dialog, which allows you to assign a name to the new Interpolation group.

The Hair Toolbar and Tool Tab

Controls in the Hair Toolbar allow you to visually modify the hair density, length, and curliness. Other tools allow you to perform global changes on the hair, to add and remove guides, and to reset the shape of the hair.

- Grow Region Tool
- Density Add Tool
- Density Remove Tool
- Density Paint Tool
- Hair Length Tool
- Brush Tool
- Straighten Tool
- Hair Cut Tool
- Push Tool
- Smooth Tool
- Curl Tool
- Clump Tool
- Reset Shape
- Even Hair Segments
- Drape
- Symmetrize
- Paint Selection Tool
- Add Guide Hair Tool
- Remove Guide Hair Tool
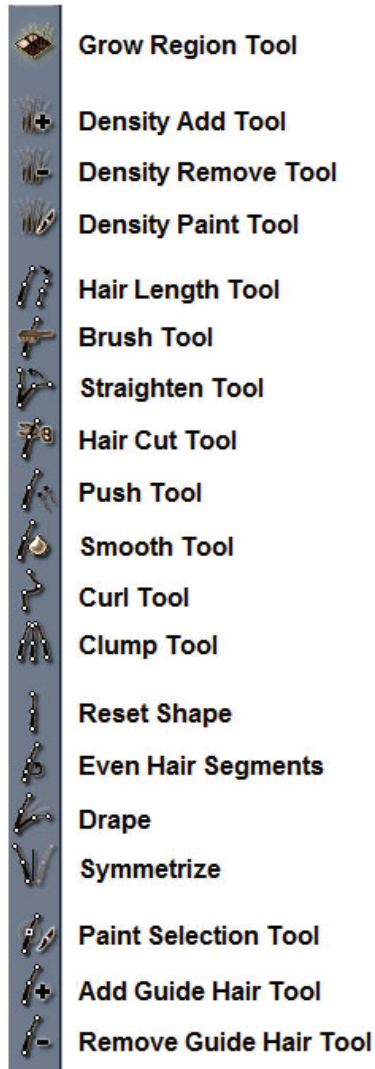
The tools in the Hair Toolbar allow you to visually style the hair. Use the Tool tab in the Model Room to set options for the selected tool.

The Hair Toolbar appears beneath the Model Room's menu bar (by default) when you have a hair object selected. You can undock the toolbar to leave it free-floating, or place it in the toolbar area at the left side of the document window. The icons in the toolbar, from left to right, are described in the sections that follow.

From left to right, the icons are as follows:

- **Grow Region** tool, discussed earlier in "Adding Hair to Selected Shading Domains"

- **Density Add** tool discussed in "Density Add Tool and Density Remove Tool"

- **Density Remove** tool discussed in "Density Add Tool and Density Remove Tool"

- **Density Paint** tool discussed in "Density Paint Tool"

- **Hair Length** tool discussed in "Hair Length Tool"

- **Brush** tool discussed in "Brush Tool"

- **Straighten** tool discussed in "Straighten Tool"

- **Hair Cut** tool discussed in "Hair Cut Tool"

- **Push** tool, discussed in "Push Tool"

- **Smooth** tool, discussed in "Smooth Tool"

- **Curl** tool, discussed in "Curl Tool"

- **Clump** tool, discussed in "Clump Tool"

- **Reset Shape** tool discussed in "Reset Shape"

- **Even Hair Segments** tool discussed in "Even Hair Segments"

- **Drape** tool discussed in "Drape"

- **Symmetrize** tool, discussed in "Symmetrize"

- **Paint Selection** tool discussed in "Paint Selection Tool"

- **Add Guide Hair** tool discussed in "Add Guide Hair Tool"

- **Remove Guide Hair** tool discussed in "Remove Guide Hair Tool"

Symmetrical Hair Styling

The tools in the Hair modeler allow you to perform symmetrical operations while you create your hair style. When applicable, you'll find an **Enable Symmetry** option in the **Properties tray: Tool tab: Global pane**. Enable this option to model and style your hair symmetrically.

Note: It is important to note that symmetrical hair styling will not work on an object that is not symmetrical.
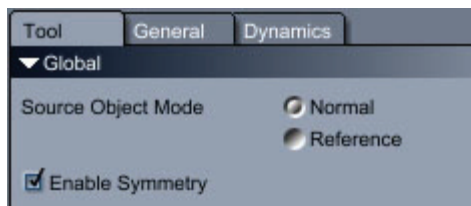
The **Source Object Mode** options also assist in applying symmetrical changes. These modes are only used for skinned objects, and function as follows:

- **Normal**: In normal mode, you edit hair on the object in its skinned position.

- **Reference**: In Reference mode, you edit the hair on the object in its original reference position (without applying the skinning). This mode is very useful when you want to use symmetry on a figure that is not posed symmetrically.

Common Tool Settings

When you choose a hair styling tool from the Hair Toolbar, its options and settings appear in the Model Room's Tool tab. There are a number of settings that are common to several tools that allow you to "paint" settings onto your object. These settings are as follows:

- **Radius**: This setting controls the size of the brush that you are using, and is represented by the size of the outer circle when the brush is used. Reduce the setting to use a smaller brush, and increase the setting to use a larger brush. You can also adjust the radius of the brush by pressing the Shift or Ctrl/Cmd key while pressing the left mouse button to resize dynamically.

- **Falloff**: The falloff setting determines the area of the brush that has the most strength. The inner circle in the brush determines the area where falloff begins. For example, if the falloff setting is 50%, the brush will have full effect in the area designated by the inner circle. The effect will gradually fade to no effect by the time it reaches the outer circle. You can also adjust the falloff setting of the brush by simultaneously pressing the Shift and Ctrl/Cmd keys while pressing the left mouse button to resize it dynamically.

- **Strength**: This setting controls the amount of effect that the brush has on the operation. With 100% strength, the brush has full effect. Reduce the strength setting to produce smaller changes with each stroke of the mouse until you get the effect you want.

Density Add Tool and Density Remove Tool

Use the **Density Add tool** to make hair more dense in areas that you paint over. This increases the number of generated hairs that are created for each hair group.

Use the **Density Remove tool** to make hair less dense in areas that you paint over. This decreases the number of generated hairs that are created for each hair group.

Use the Density Add and Density Remove tools to paint in grayscale while increasing or decreasing the density of the hair. Black areas are equal to 0% density and white areas are equal to 100% density.

When either of these tools are selected, the Tool tab displays options that allow you to set the radius, falloff, and strength of the brush.
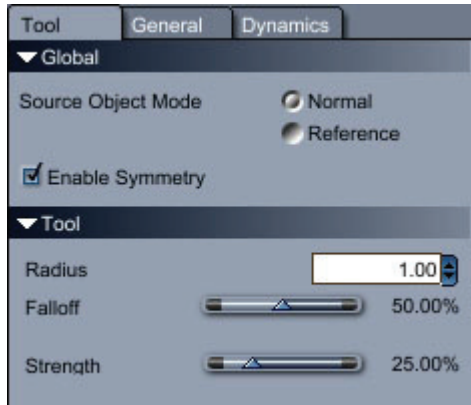
To increase or decrease density:

1. Choose one of the following tools:

    - To increase density in painted areas, choose the **Density Add tool**.

    - To decrease density in painted areas, choose the **Density Remove tool**.

2. In the Global section, choose the desired source object mode:

    - **Normal**: In Normal mode, you edit the hair on the object in its skinned position.

    - **Reference**: In Reference mode, you edit the hair on the object in its reference position. This is useful when you want to use symmetry on a figure that is in a non-symmetrical pose.

3. If you want to apply symmetrical changes to the density, check the **Enable Symmetry** option in the Global panel.

4. Set the following properties:

    - Set the **Radius** of the brush to achieve the correct brush size.

    - Set the **Falloff** to determine how quickly the brush will fall of from maximum effect (at the center) and no effect (at the outside boundary of the brush).

    - Set the **Strength** of the brush. 100% applies the density setting at its maximum value over the polygons that you paint. Lower settings require repeated strokes to achieve the same result.

5. Paint over the object to add or remove density in the areas that you desire: When the hair is at its maximum density the polygons will be white. When density is at 0% the polygons will be black.

Density Paint Tool

Use this tool to increase or decrease density in the areas that you paint. Unlike the Add Density or Remove Density brushes, the Density Paint Tool allows you to set a specific value for density, and it increases or decreases appropriately where it paints.

The Density setting controls the number of hairs that exist in relation to a hair group. A hair density setting of 100% generates the full number of hairs that are associated with a guide hair. A 50% hair density setting generates half the number of generated hairs.
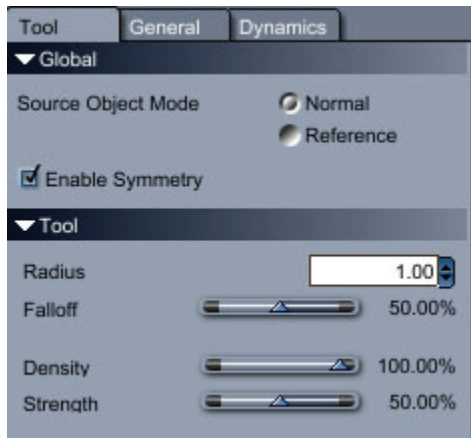
When using one of the Density tools, the display of the scene changes to display your model in grayscale:

- White areas on the model indicate the maximum amount of hair density. When you initially create hair, density is set to the maximum value.

- Black areas indicate where hair will not grow or where the density is set to zero.

- Shades of gray in a hair area indicate values that are somewhere between no density (black) and maximum density (white).

In addition to the common Radius, Falloff, and Strength settings, the Density Paint tool provides the following options in the Tool tab:

- **Density**: This slider determines the maximum amount of density that will be added to the polygons that you paint with the brush. In other words, you can set the Density slider to 50%, and then paint in the areas that you want to set to that value.

To set hair density to a specific value:

1. To set density to a specific value over all polygons in the hair group, choose **Hair > Set Density**.

2. Enter the desired value in the Set Hair Density dialog.

To change hair density in selected areas:

1. If you want to affect only certain guide hairs, use the Paint Selection tool to select the guide hairs you want to affect. With no selection made, all guide hairs will be affected.

2. To increase or decrease density on all or selected guides, click the **Density Paint tool**.

3. In the Global section, choose the desired source object mode:

    - **Normal**: In Normal mode, you edit the hair on the object in its skinned position.

    - **Reference**: In Reference mode, you edit the hair on the object in its reference position. This is useful when you want to use symmetry on a figure that is in a non-symmetrical pose.

4. If you want to apply symmetrical changes to the density, check the **Enable Symmetry** option in the Global panel.

5. Set the following properties:

    - Set the **Radius** of the brush to achieve the correct brush size.

    - Set the **Falloff** to determine how quickly the brush will fall of from maximum effect (at the center) and no effect (at the outside boundary of the brush).

    - Set the **Density** setting to the amount of density that you want to apply to the polygons over which you paint.

- Set the **Strength** of the brush. 100% applies the density setting at its maximum value over the polygons that you paint. Lower settings require repeated strokes to achieve the same result.

6. Paint over the object to add or remove density in the areas that you desire.

Hair Length Tool

Use the Hair Length tool to lengthen the hair on the guide hairs that you paint.

In addition to the common Radius, Falloff, and Strength settings, the Hair Length tool provides the following options in the Tool tab:

- **Flatness**: Controls the shape of the brush. At 0% the brush has a spherical shape, while at 100% the brush is flattened into a circle.

- **Restrict to Selection**: Confines actions of the Hair Length tool to the guide hairs that have been selected with the Paint Selection tool, discussed previously.

- **Length**: The Length setting determines whether hair will be lengthened or shortened when you paint. If the Length setting is higher than the current length, the hair will grow. If the Length setting is smaller than the current length, and if Absolute is checked, the hair will get shorter.

- **Absolute**: With the Absolute option checked, the Length setting that you enter is applied to all of the hairs that you paint. When unchecked, the length of all underlying hair is increased or decreased based on the Length setting.
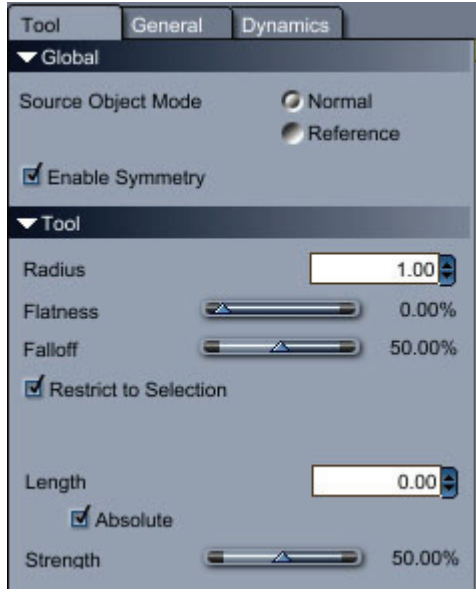
To change the length of the hair:

1. If desired, use the **Hair Selection tool** to select the guide hairs that you want to change. With no selection made, all guide hairs will be affected.

2. Choose the **Hair Length tool**.

3. In the Global section, choose the desired source object mode:

- **Normal**: In Normal mode, you edit the hair on the object in its skinned position.

- **Reference**: In Reference mode, you edit the hair on the object in its reference position. This is useful when you want to use symmetry on a figure that is in a non-symmetrical pose.

4. If you want to apply symmetrical changes to the density, check the **Enable Symmetry** option in the Global panel.

5. Set the following properties:

- Set the **Radius** of the brush to achieve the correct brush size.

- Use the **Flatness** slider to change the shape of the brush if desired.

- Set the **Falloff** to determine how quickly the brush will fall of from maximum effect (at the center) and no effect (at the outside boundary of the brush).

6. If you want to affect only selected guide hairs, check the **Restrict to Selection** option. To affect all areas, uncheck this option.

7. Enter a hair length in the **Length** field.

8. Check the **Absolute** option if you do not want the hair to exceed the set length.

9. Uncheck the **Absolute** option to allow hair to grow by the amount designated in the Length field each time you paint over the hairs.

10. Adjust the **Strength** setting for more subtle or more pronounced changes while you paint.

11. Paint over the areas where you want the hair to grow.

Brush Tool

The Brush tool adds curl, waves, and styling to the guide hairs. In addition to the common Radius, Falloff, and Strength settings, the Brush tool provides two modes of operation: Screen and Surface.

Screen Mode Options

In Screen mode, the following options apply in addition to the standard Radius, Falloff, and Strength settings:
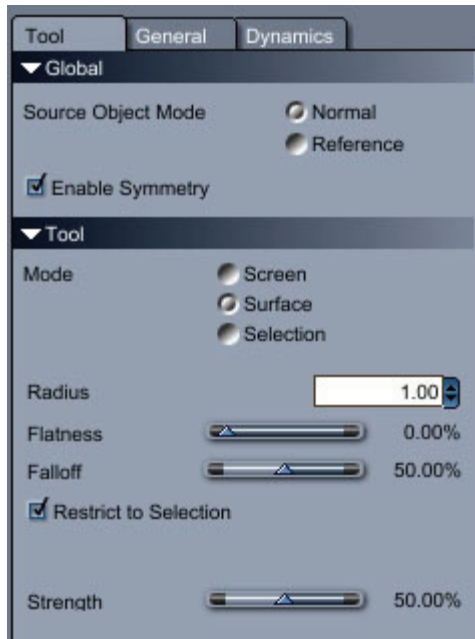
Screen mode options for the Brush tool

- **Point**: The Point slider adjusts the position of the guide's point. You can use the + or – buttons to move the point closer to or farther away from the base of the head. This point determines the point at which the hair moves when you style the hair. For example, if you want to affect the style at the middle of the hair guide, position the point in the middle, and then brush the hair to pull the middle of the hair in that direction.

- **Restrict to Selection**: Confines actions of the Remove Guide tool to the guidehairs that have been selected with the Paint Selection tool, discussed previously.

- **Apply Only to Visible Guide Hairs**: Check this option to affect only the hairs that are facing toward you. Backward-facing geometry will not be affected.

- **Stiffness**: With a high stiffness setting, pulling on one point will attract the points of other guide hairs that are near it. For example, if the point is placed in the middle of the hair, you will see the brush influence more hairs around the brush when the Stiffness setting is increased.

- **Use Hair IK**: Check this option to maintain the hair segment lengths while you brush the hair. This works similar to inverse kinematics in bones. You can pull the last point of the hair and all points before it will move accordingly to match the new position if it can be reached. When IK is off, you can change the length of the hair and add additional segments to the hair.

- **Collision Distance**: Allows you to specify a value at which the hair will be considered to be colliding with the surfaces it comes into contact with. Hair will not be allowed to get any closer than this value.

To brush hair in Screen mode:

1. If desired, use the Hair Selection tool to select the guide hairs that you want to change. With no selection made, all guide hairs will be affected.

2. Choose the **Brush tool**.

3. In the Global section, choose the desired source object mode:

    - **Normal**: In Normal mode, you edit the hair on the object in its skinned position.

- **Reference**: In Reference mode, you edit the hair on the object in its reference position. This is useful when you want to use symmetry on a figure that is in a non-symmetrical pose.

4. Check the **Enable Symmetry** option if you want to brush the hair symmetrically.

5. In the Mode section, choose the **Screen** option.

6. Adjust the **Point** slider to determine the point from which the brush will move the hair.

7. Check the **Restrict to Selection** option if you want to restrict brushing to the selected guide hairs.

8. Check the **Apply only to visible guide hairs** option to prevent accidental brushing of the guide hairs on the opposite side of the head.

9. Adjust the **Strength** and **Stiffness** sliders to obtain the desired amount of response from the Brush tool.

10. Check the **Use Hair IK** option if you want to keep the hair at the set length while you brush it.

11. Set the **Collision Distance** value to closest value that hair will move toward adjacent geometry.

12. Drag the Brush tool across the guide hairs to style the hair as desired.

Surface Mode Options

In Surface mode, the following options apply:
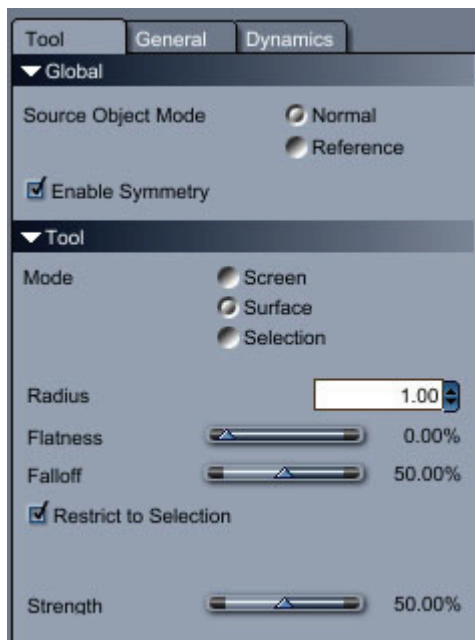
- **Flatness**: Controls the shape of the brush. At 0% the brush has a spherical shape, while at 100% the brush is flattened into a circle.

- **Restrict to Selection**: Confines actions of the Brush tool to the guide hairs that have been selected with the Paint Selection tool, discussed previously.

- **Curve**: Controls the amount of curve that is added to the guide hair as it is being brushed.

To brush hair in Surface mode:

1. If desired, use the Hair Selection tool to select the guide hairs that you want to change. With no selection made, all guide hairs will be affected.

2. Choose the **Brush tool**.

3. In the Global section, choose the desired source object mode:

    - **Normal**: In Normal mode, you edit the hair on the object in its skinned position.

    - **Reference**: In Reference mode, you edit the hair on the object in its reference position. This is useful when you want to use symmetry on a figure that is in a non-symmetrical pose.

4. Check the **Enable Symmetry** option if you want to brush the hair symmetrically.

5. In the Mode section, choose the **Surface** option.

6. Set the **Radius**, **Flatness**, and **Falloff** settings for the brush.

7. Check the **Restrict to Selection** option if you want to restrict brushing to the selected guide hairs.

8. Adjust the **Strength** and **Curve** settings to control the amount of styling and curl that will be applied to the hair.

9. Drag the Brush tool across the guide hairs to style the hair as desired.

Straighten Tool

The Straighten tool removes curl or straightens the guide hairs. The options for the Straighten tool are very similar to those listed for the Brush tool, but provide three different modes of operation: Screen, Surface, and Selection.

Screen Mode Options

In Screen mode, the following options apply in addition to the standard Radius, Falloff, and Strength settings:

- **Point**: The Point slider adjusts the position of the guide's point. You can use the + or – buttons to move the point closer to or farther away from the base of the head:

    o When the point is placed near the head, changes take effect over the entire length of the hair.

    o When the point is placed at the tip of the guide hair, the styling controls affect only the tip of the hair.

    o When the point is placed in the middle of the guide hair, styling is applied from the position of the point to the tip of the hair.

- **Restrict to Selection**: Confines actions oft he Straighten tool to the guidehairs that have been selected with the Paint Selection tool, discussed previously.

- **Apply Only to Visible Guide Hairs**: Check this option to affect only the hairs that are facing toward you. Backward-facing geometry will not be affected.

To straighten hair in Screen mode:

1. If desired, use the Hair Selection tool to select the guide hairs that you want to change. With no selection made, all guide hairs will be affected.

2. Choose the **Straighten tool**.

3. In the Global section, choose the desired source object mode:

    - **Normal**: In Normal mode, you edit the hair on the object in its skinned position.

    - **Reference**: In Reference mode, you edit the hair on the object in its reference position. This is useful when you want to use symmetry on a figure that is in a non-symmetrical pose.

4. Check the **Enable Symmetry** option if you want to brush the hair symmetrically.

5. In the Mode section, choose the **Screen** option.

6. Adjust the **Radius** and **Falloff** for the brush to achieve the desired size and area for the Straighten tool.

7. Adjust the **Point** slider to determine the point from which the brush will move the hair.

8. Check the **Restrict to Selection** option if you want to restrict brushing to the selected guide hairs.

9. Check the **Apply only to visible guide hairs** option to prevent accidental brushing of the guide hairs on the opposite side of the head.

10. Adjust the **Strength** and **Stiffness** sliders to obtain the desired amount of response from the Brush tool.

11. Drag the Brush tool across the guide hairs to style the hair as desired.

Surface Mode Options

In Surface mode, the following options apply:
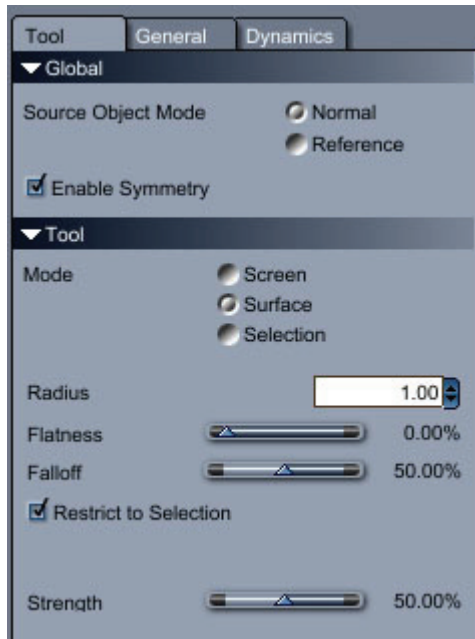
- **Flatness**: Controls the shape of the brush. At 0% the brush has a spherical shape, while at 100% the brush is flattened into a circle.

- **Restrict to Selection**: Confines actions of the Straighten tool to the guide hairs that have been selected with the Paint Selection tool, discussed previously.

To straighten hair in Surface mode:

1. If desired, use the Hair Selection tool to select the guide hairs that you want to change. With no selection made, all guide hairs will be affected.

2. Choose the **Straighten tool**.

3. In the Global section, choose the desired source object mode:

    - **Normal**: In Normal mode, you edit the hair on the object in its skinned position.

    - **Reference**: In Reference mode, you edit the hair on the object in its reference position. This is useful when you want to use symmetry on a figure that is in a non-symmetrical pose.

4. Check the **Enable Symmetry** option if you want to brush the hair symmetrically.

5. In the **Mode** section, choose the **Surface** option.

6. Set the **Radius**, **Flatness**, and **Falloff** settings for the brush.

7. Check the **Restrict to Selection** option if you want to restrict brushing to the selected guide hairs.

8. Adjust the **Strength** and **Curve** settings to control the amount of styling and curl that will be applied to the hair.

9. Drag the Straighten tool across the guide hairs to straighten the hair as desired.

Selection Mode Options

In Selection mode, you can draw a selection marquee around the guide hairs that you want to straighten. To deselect the guide hairs, either click outside the selection, or press the Ctrl/Cmd key while creating another selection marquee area around the guides you want to deselect.
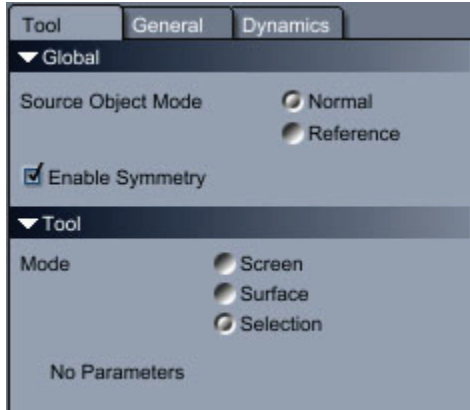
Hair Cut Tool

The Hair Cut tool allows you to shorten the length of all or selected guide hairs.

In addition to the common Radius setting, the Hair Cut tool provides the following options in the Tool tab:
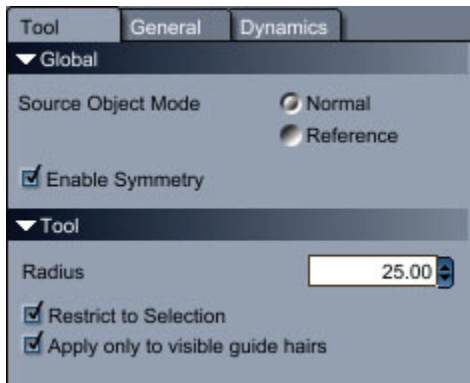
- **Restrict to Selection**: Confines actions of the Remove Guide tool to the guide hairs that have been selected with the Paint Selection tool, discussed previously.

- **Apply Only to Visible Guide Hairs**: Check this option to affect only the hairs that are facing toward you. Backward-facing geometry will not be affected.

To cut hair:

1. If desired, use the **Hair Selection tool** to select the guide hairs that you want to change. With no selection made, all guide hairs will be affected.

2. Choose the **Hair Cut tool**.

3. In the Global section, choose the desired source object mode:

   - **Normal**: In Normal mode, you edit the hair on the object in its skinned position.

757

- **Reference**: In Reference mode, you edit the hair on the object in its reference position. This is useful when you want to use symmetry on a figure that is in a non-symmetrical pose.

- Check the **Enable Symmetry** option if you want to brush the hair symmetrically.

4. Set the Radius of the brush to the desired size.

5. Check the **Restrict to Selection** option if you want to restrict brushing to the selected guide hairs.

6. Check the **Apply only to visible guide hairs** option to prevent accidental brushing of the guide hairs on the opposite side of the head.

7. Drag the Hair Cut tool across the points at the bottom of the guide hairs to shorten the hair to the desired length.

Push Tool

Use the Push tool to pull all or selected hairs away from the surface of an object. This tool is particularly useful to use after guide hairs have been moved too close to or inside the head after draping or styling.

The Push tool provides the following options:
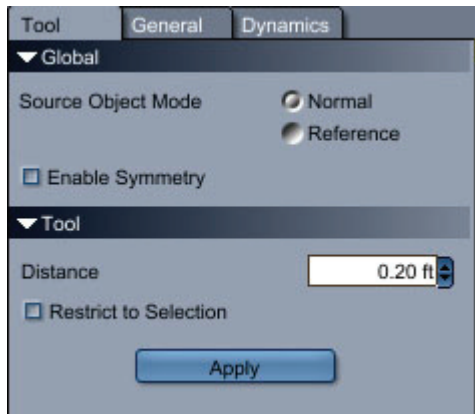
- **Distance**: Set the desired distance that you want to push the hairs.

- **Restrict to Selection**: Confines actions of the Remove Guide tool to the guide hairs that have been selected with the Paint Selection tool, discussed previously.

- **Apply**: Click the Apply button to push the selected hairs by the desired distance.

To push hair:

1. If desired, use the Hair Selection tool to select the guide hairs that you want to change. With no selection made, all guide hairs will be affected.

2. Choose the **Push tool**.

3. In the Global section, choose the desired source object mode:

- **Normal**: In Normal mode, you edit the hair on the object in its skinned position.

- **Reference**: In Reference mode, you edit the hair on the object in its reference position. This is useful when you want to use symmetry on a figure that is in a non-symmetrical pose.

4. Check the **Enable Symmetry** option if you want to brush the hair symmetrically.

5. Adjust the **Distance** setting to the amount of push that you want to obtain.

6. Check the **Restrict to Selection** option to use the tool on the selected guide hairs. Leave this option unchecked to apply the tool to all guide hairs.

7. Click the **Apply** button to push the hair by the desired amount.

Smooth Tool

The Smooth tool smoothes the variation along the hairs. For example, if hair is very kinky you can use the Smooth tool to reduce the kinkiness and move it more toward a completely straight line. When smoothed to the maximum amount, the hair will be just a straight line from root to tip. Moving the mouse farther away from the click point applies smoothing with greater strength.

The Smooth tool has no additional options.
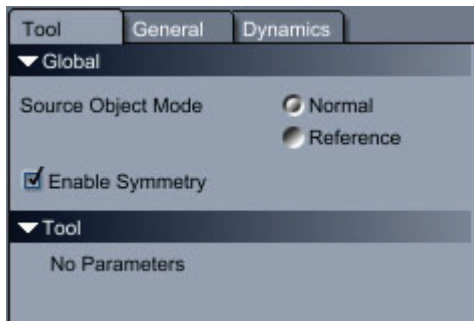
To smooth hair:

1. Choose the Smooth tool.

2. In the Global section, choose the desired source object mode:

    - **Normal**: In Normal mode, you edit the hair on the object in its skinned position.

    - **Reference**: In Reference mode, you edit the hair on the object in its reference position. This is useful when you want to use symmetry on a figure that is in a non-symmetrical pose.

3. Check the **Enable Symmetry** option if you want to brush the hair symmetrically.

4. Use the Smooth tool to make a rectangular selection around the guide hairs that you want to smooth.

    - If Symmetry is enabled, you can select guide hairs on one side of the head.

    - Press the Ctrl/Cmd key while dragging the mouse to deselect a rectangular area.

5. After the selection is made, move the mouse over the selection to achieve the amount of smoothness that you desire. If Symmetry is enabled, you should see both sides getting smoother as you move the mouse.

Curl Tool

The Curl tool applies curl to selected guide hairs. Moving the mouse farther from the click point applies curl with greater strength.

The Curl tool provides the following options:
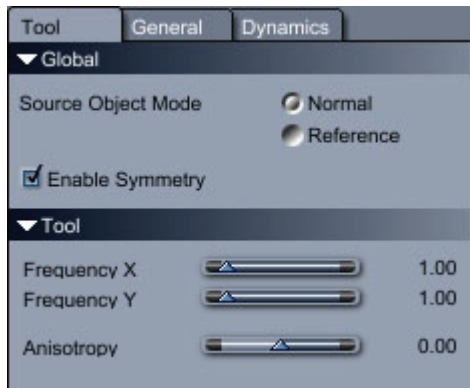
- **Frequency X**: Controls the number of curls along the X axis.
- **Frequency Y**: Controls the number of curls along the Y axis.
- **Anisotropy**: Controls the size of the curl.

To curl hair:

1. In the **General tab: Hairs (Per Hair Group)** panel, increase the **Segments Per Hair** setting to a value that will be suitable for creating smooth curls. If the value is set too low the hair will look zig-zagged. For example, you can initially set the value to 100 segments, and reduce it after you curl the hair.

2. Choose the **Curl tool**.

3. In the Global section, choose the desired source object mode:

    - **Normal**: In Normal mode, you edit the hair on the object in its skinned position.

    - **Reference**: In Reference mode, you edit the hair on the object in its reference position. This is useful when you want to use symmetry on a figure that is in a non-symmetrical pose.

4. Check the **Enable Symmetry** option if you want to brush the hair symmetrically.

5. Adjust the **Frequency X**, **Frequency Y**, and **Anisotropy** settings to achieve the amount of curl that you want to add to the hair.

6. Use the Curl tool to make a rectangular selection around the guide hairs that you want to curl.

    - If Symmetry is enabled, you can select guide hairs on one side of the head.

    - Press the Ctrl/Cmd key while dragging the mouse to deselect a rectangular area.

7. After the selection is made, move the mouse over the selection to achieve the amount of curl that you desire. If Symmetry is enabled, you should se both sides getting curlier as you move the mouse.

NOTE! The length of the guide hairs will increase so that the curled hair remains at approximately the same length as the uncurled hair.

8. To use less system resources, return to the **General tab: Hairs (Per Hair Group) panel** and decrease the **Segments Per Hair** setting a little bit at a time, while keeping the curl at an acceptable level.

Clump Tool

The Clump tool deforms selected hairs toward a reference hair. When you move the mouse farther from the reference point, the clump is applied at a greater strength.

The Clump tool provides the following options:
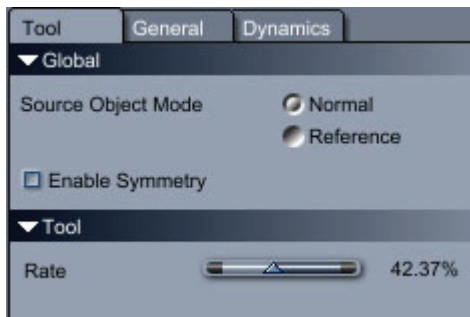
The Clump tool has only one property to set:

- **Rate**: Increase the Rate setting to increase the number of hairs that are pulled

To clump hair:

1. If desired, use the Hair Selection tool to select the guide hairs that you want to change. With no selection made, all guide hairs will be affected.

2. Choose the Clump tool.

3. In the Global section, choose the desired source object mode:

    - **Normal**: In Normal mode, you edit the hair on the object in its skinned position.

    - **Reference**: In Reference mode, you edit the hair on the object in its reference position. This is useful when you want to use symmetry on a figure that is in a non-symmetrical pose.

4. Check the **Enable Symmetry** option if you want to clump the hair symmetrically.

5. Adjust the **Rate** setting to control the size of the hair clumps (the number of hairs that will clump together when you use the tool). A high setting moves all selected hairs toward a common point, whereas a lower setting moves a smaller group of hairs toward a common point.

6. Use the Clump tool to make a rectangular selection around the guide hairs that you want to clump.

    - If Symmetry is enabled, you can select guide hairs on one side of the head.

    - Press the Ctrl/Cmd key while dragging the mouse to deselect a rectangular area.

7. After the selection is made, move the mouse over the selection to achieve the amount of clump that you desire.

    - If Symmetry is enabled, you should se both sides clumping symmetrically. With a high Rate setting, hairs will clump together as if being pulled together to make pigtails.

    - If you use a high Rate setting with Symmetry turned off, all hairs will clump together as if to make a bun or ponytail.

Reset Shape

Use the Reset Shape tool to remove styling from all of the hair. This resets the guide hairs to straight lines that are perpendicular to the polygon surface, and point outward.

To reset the shape of the hair:

1. If desired, use the Hair Selection tool to select the guide hairs that you want to change. With no selection made, all guide hairs will be affected.

2. Choose the Reset Shape tool. The selected hairs are reset to grow straight out and perpendicular to the growing surface.

Even Hair Segments

After you brush the hair, especially with the IK off, you can end up with hair segments that are uneven. This tool resamples the hair shape and makes all segments even.

To even hair segments:

1. If desired, use the Hair Selection tool to select the guide hairs that you want to change. With no selection made, all guide hairs will be affected.

2. Choose the Even Hair Segments tool.

Drape

The Drape tool causes all guide hairs to relax and appear more natural, based on the other settings you have applied to the guide hairs and generated hairs.

To drape hair:

1. Click the Drape tool. The hairs will relax to fall naturally over the growing surface.

2. To convert the guide hairs to the relaxed shape, open the **Dynamics tab: Simulation panel**. Click the **Convert** button in the Draping section.

Symmetrize

Most of the Hair tools allow you to edit the hair symmetrically, relative to a plane. In order for symmetry to work, you need to enable symmetrical modeling and choose the plane that will determine the symmetry.

To enable symmetrical modeling and to choose the plane:

1. Open the **Properties tray: General tab: Guide Hairs (Per Hair Group) pane**.

2. Check the **Symmetrical Sampling** option to enable the symmetry plane.

3. Choose the desired plane (**YZ**, **XZ**, or **XY**) from the **Plane** popup menu.

To symmetrize hair:

- Click the Symmetrize tool.

Paint Selection Tool

Several tools allow you to restrict styling and shaping to selected hair guides. You can use the Paint Selection tool to select the guide hairs you want to edit. You can also control the portion of the hair that you want to affect by adjusting the position of the point.

In addition to the Radius setting, the following settings are provided for the Paint Selection tool:
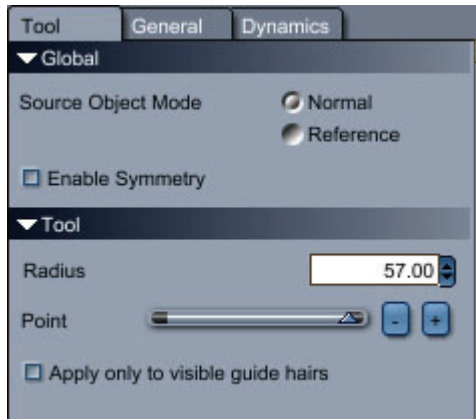
- **Point**: The Point slider adjusts the position of the guide's point. You can use the + or – buttons to move the point closer to or farther away from the base of the head:

  o When the point is placed near the head, changes take effect over the entire length of the hair.

  o When the point is placed at the tip of the guide hair, the styling controls affect only the tip of the hair.

  o When the point is placed in the middle of the guide hair, styling is applied from the position of the point to the tip of the hair.

- **Apply Only to Visible Guide Hairs**: Check this option to affect only the hairs that are facing toward you. Backward-facing geometry will not be affected.

To select or deselect hair guides:

1. Choose the Paint Selection tool.

2. In the Global section, choose the desired source object mode:

   - **Normal**: In Normal mode, you edit the hair on the object in its skinned position.

   - **Reference**: In Reference mode, you edit the hair on the object in its reference position. This is useful when you want to use symmetry on a figure that is in a non-symmetrical pose.

3. Check the **Enable Symmetry** option if you want to select hairs symmetrically.

4. Set the **Radius** of the brush. You can also press the Ctrl/Cmd key while dragging the Paint Selection tool in the document window to resize the brush.

5. Use the Point slider or +/- buttons to adjust the position of the point on the guide hairs. The position of the point determines the portion of the hairs that will be affected by the tools you apply to the hair selection.

6. Check the **Apply only to Visible Guide Hairs** option to prevent accidental selection of back-facing guide hairs.

7. Paint in the document window to select or deselect hairs as follows:

   - To select hairs, paint over the hair guides that you want to select. Selected guide hairs turn orange.

   - To deselect hairs, press the Shift key while painting over the guide hairs that you want to deselect.

Add Guide Hair Tool

When you create hair on an object that does not have UV maps applied, you may find that there are enough guide hairs to style the hair. However, when adding hair to the head of a figure, you may need to increase the number of guide hairs to provide better control over styling the hair.

The General tab (discussed later) includes a setting that allows you to globally increase or decrease the number of guide hairs. However, in cases where you want to add more guide hairs in specific areas, use the Add Guide Hair tool to paint more guide hairs where you need them.

In addition to the common Radius, Falloff, and Strength settings, the Add Guide Hair tool provides the following options in the Tool tab:
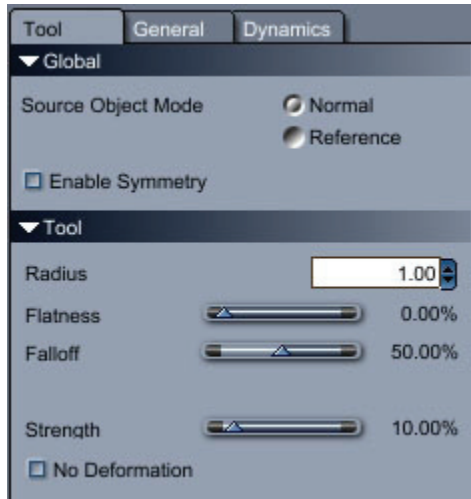
- **Flatness**: Controls the shape of the brush. At 0% the brush has a spherical shape, while at 100% the brush is flattened into a circle.

- **No Deformation**: When unchecked, the guide hairs that you add will have a shape that is similar to those that are nearby. When checked, guide hairs are added perpendicular to the surface, regardless of the shape of nearby hairs.

To add guide hairs:

1. Choose the **Add Guide Hair tool**.

2. In the Global section, choose the desired source object mode:

   - **Normal**: In Normal mode, you edit the hair on the object in its skinned position.

   - **Reference**: In Reference mode, you edit the hair on the object in its reference position. This is useful when you want to use symmetry on a figure that is in a non-symmetrical pose.

3. Check the **Enable Symmetry** option if you want to add guides symmetrically.

4. Adjust the **Radius**, **Flatness**, and **Falloff** of the brush.

5. Increase or decrease the **Strength** of the brush to control the number of guide hairs that are added while you paint.

6. Check or uncheck the **No Deformation** option to determine the shape of the guide hairs that you add.

7. Paint on the head in the areas that will receive the additional guide hairs.

Remove Guide Hair Tool

The Remove Guide Hair tool deletes guide hairs from the polygons that you paint.

In addition to the common Radius, Falloff, and Strength settings, the Remove Guide Hair tool provides the following options in the Tool tab:
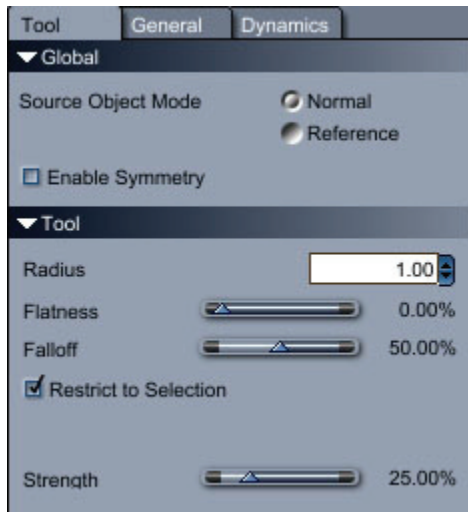
- **Flatness**: Controls the shape of the brush. At 0% the brush has a spherical shape, while at 100% the brush is flattened into a circle.

- **Restrict to Selection**: Confines actions of the Remove Guide tool to the guide hairs that have been selected with the Paint Selection tool, discussed previously.

To remove guide hairs:

1. If desired, use the Hair Selection tool to select the guide hairs that you want to affect. With no selection made, all guide hairs will be affected.

2. Choose the **Remove Guide Hair tool**.

3. In the Global section, choose the desired source object mode:

   - **Normal**: In Normal mode, you edit the hair on the object in its skinned position.

   - **Reference**: In Reference mode, you edit the hair on the object in its reference position. This is useful when you want to use symmetry on a figure that is in a non-symmetrical pose.

4. Check the **Enable Symmetry** option if you want to remove guides symmetrically.

5. Adjust the **Radius**, **Flatness**, and **Falloff** of the brush.

6. Check the **Restrict to Selection** option if you want to only remove hair guides from a selected area.

7. Increase or decrease the **Strength** of the brush to control the number of guide hairs that are removed while you paint.

8. Paint in the areas from which you want to remove the guide hairs.

Dynamics Tab

The Dynamics tab controls how the hair behaves when it is in motion during a simulation. If your hair is too "bouncy" you can increase the stiffness so that it stays in place. The Dynamics tab also allows you to choose the forces in your scene that will affect the movement of the hair.

There are three sections in the Dynamics tab: Dynamics Properties (Per Hair Group), Simulation, and Forces. Each of these tabs is described in the sections that follow.
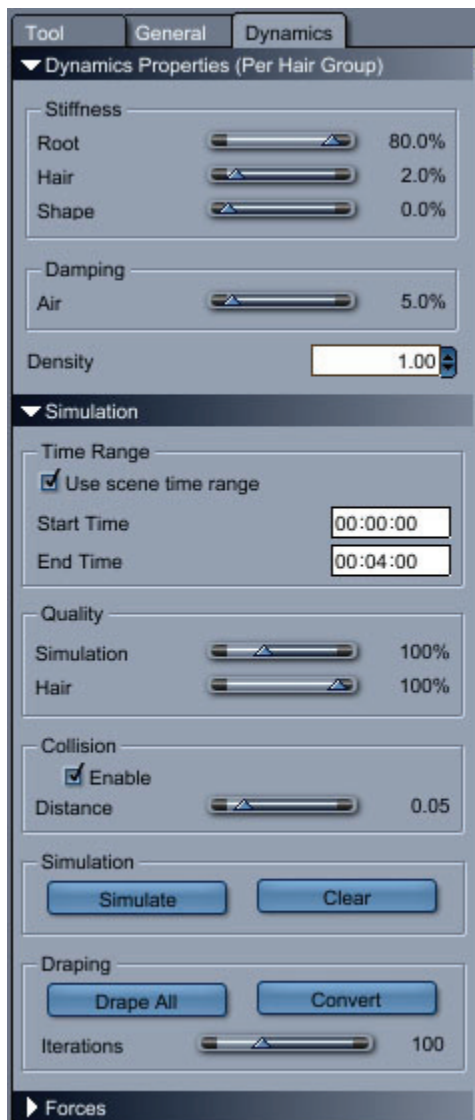
Dynamics Properties
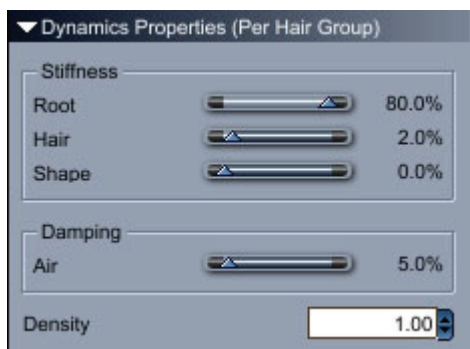
- **Stiffness Section**: Hair that has a higher stiffness setting will be less likely to change shape during simulation.
- **Root**: Controls the amount of bending from the hair's reference position.

- **Hair**: Controls how much the hair is allowed to bend.

- **Shape**: Controls how much the hair will try to keep its reference shape and position.

- **Damping Section**:

- **Air**: This setting controls how much the hair will interact or lose energy with the air.

- **Density**: This setting controls the number of hairs that are generated with each hair group. The number of hairs defined in the Hair Count section are multiplied by this value to determine the density.

Simulation
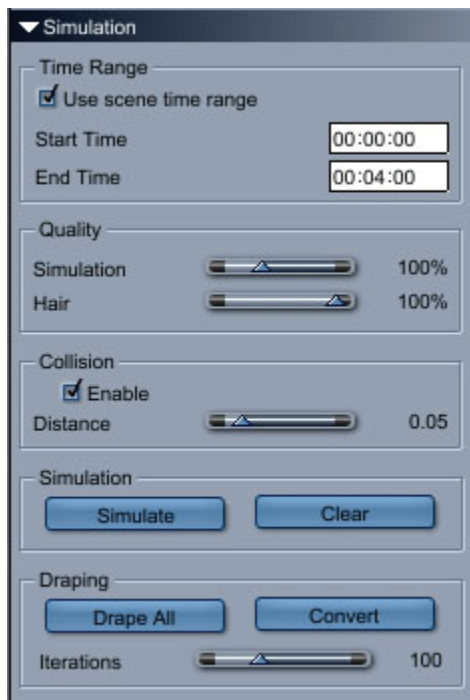
Control the starting and ending times, quality of hair, and collision detection in the Simulation section. You can also simulate or drape the hair on demand, and convert draped or styled hair to a new reference shape.

- **Time Range Section**: This section determines the time range in the animation in which simulation will take place. You can enter your own values if you are working on a particular part of your animation.

  o **Use Scene Time Range**: Check this option to perform dynamic hair simulations throughout the entire range of the scene. Uncheck this option to enter a specific time range.

  o **Start Time**: Enter the start time of the simulation in MM:SS:FF format.

  o **End Time**: Enter the end time of the simulation in MM:SS:FF format.

- **Quality Section**: This section determines the quality of the simulation. Higher settings will produce better results but make simulations slower.

  o **Simulation**: Adjust this setting to raise or lower simulation quality.

  o **Hair**: Controls the percentage of hair segments to control during the simulation.

- **Collision Section**: This section determines whether the hair will be affected when it collides with other objects in the scene.

    o **Enable**: Check this option to enable collision detection. When disabled, hair will poke through objects that it collides with.

    o **Distance**: This setting controls how close the hairs should get to other objects before a collision is detected.

- **Simulation Section**: Controls in the Simulate section allow you to manually initiate simulation calculations.

    o **Simulate**: Press this button to begin simulation.

    o **Clear**: Press this button to remove the current simulation data. When you play an animation after clearing a simulation the hair will not move.

- **Draping Section**: Controls in the Draping section allow you to initiate draping of hair, and to convert draped hair to a new shape.

    o **Drape All**: Press this button to drape the hair a little bit at a time. The hair will react to forces a little bit at a time. This setting is different from simulation in that it does not become animation data. You can press the Escape key at any point during the animation to use the current draped shape.

    o **Convert**: Click this button to keep the hairs in the draped shape.

    o **Iterations**: This setting determines the maximum number of times that forces are applied to the hair before it stops draping.

Forces

The Forces section allows you to choose the forces in your scene that will be allowed to interact with the hair. Forces that are present in your scene will be listed in this area.
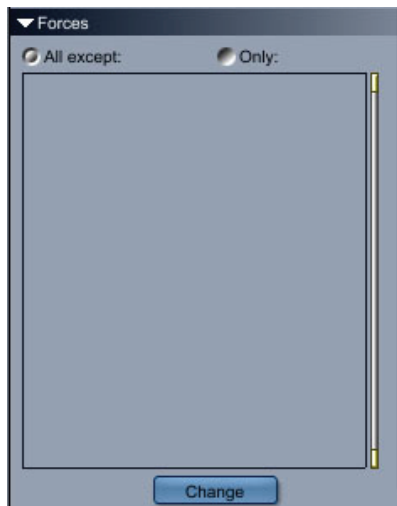
- **All Except**: Check this option if you want to remove one or more forces that currently interact with the hair.

- **Only**: Check this option if you want to add forces to interact with the hair.

- **Change**: Click the Change button to open the Choose Forces dialog. Forces that appear in your scene will be listed here. After you select the forces that you want to add or remove, click OK to return to the scene.

Assemble Room Hair Properties

You have a limited number of properties when you work with hair in the Assemble Room. The upper portion of the General tab in the Assemble Room consists of two sections: General and Hair.
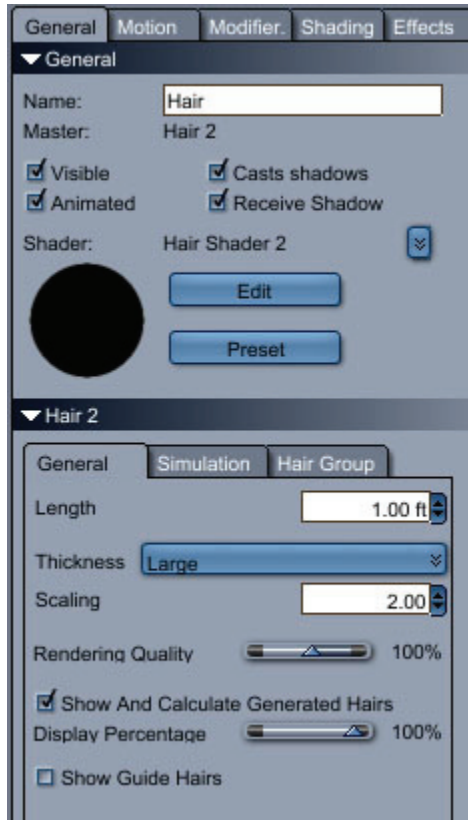
General Properties

Use the General section to assign the following properties to the hair object:
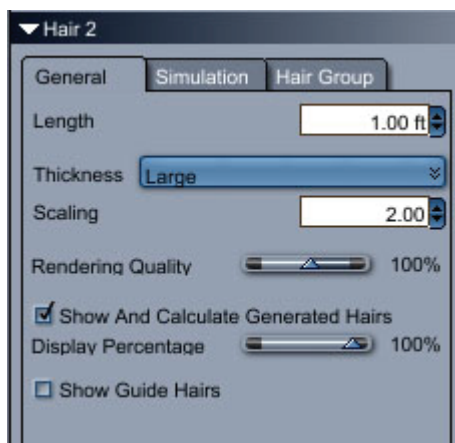
- **Name**: By default, a new hair object is named Hair. You can rename the hair in this field.
- **Master**: TBD

- **Visible**: Check or uncheck this option to make the hair visible in your scene.

- **Animated**: Check or uncheck this option to allow or disallow animation on the hair object.

- **Casts Shadows**: Check or uncheck this option to allow the hair to cast shadows in your scene.

- **Receive Shadows**: Check or uncheck this option to allow the hair to receive shadows from other objects in your scene.

- **Shader Section**: This section allows you to assign or edit the shader that the hair uses.

    - o **Selection Menu**: Click the arrow to the right of the current shader name to select another shader that is already used in your scene.

    - o **Edit**: Click the Edit button to open the Hair Shader window, which allows you to customize the current hair shader.

- **Preset**: Click the Preset button to choose a shader preset for the current hair object.

Hair Properties

You'll find three sub-tabs in the Hair section, each of which is described in the following sections.

General Tab

The options you can set in the General tab are as follows:
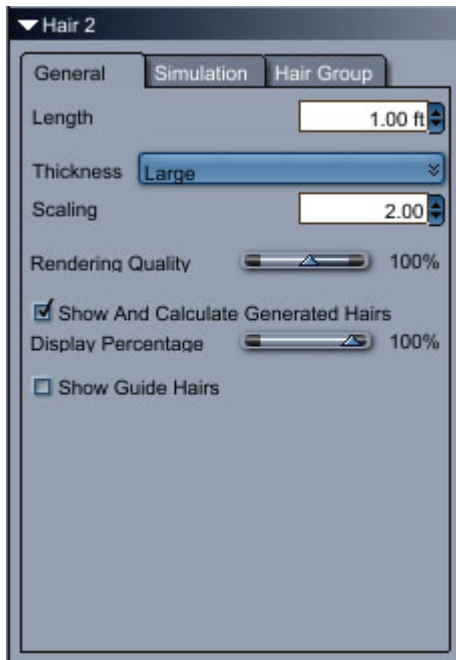
- **Length**: Enter a new value or use the arrows to increase or decrease the length of the hair.

- **Thickness**: Use the drop-down menu to select Thin, Medium, Large, or Very Large thickness setting.

- **Scaling**: Use the arrows or enter a new value to increase or decrease the scaling of the hair, which also makes the hair thicker.

- **Rendering Quality**: Use the arrows or enter a new value to raise or lower rendering quality of the hairs.

- **Show Generated Hairs**: Check this option to see the generated hairs while you work on the hair. Uncheck this option to hide the hairs.

- **Display Percentage**: Adjust this slider to raise or lower the percentage of generated hairs that you see.

- **Show Guide Hairs**: Check this option to display guide hairs (the hairs that are used to style the hair). Guide hairs appear as yellow lines in the Assemble room.

Simulation Tab

The options you can set in the Simulation tab are as follows:
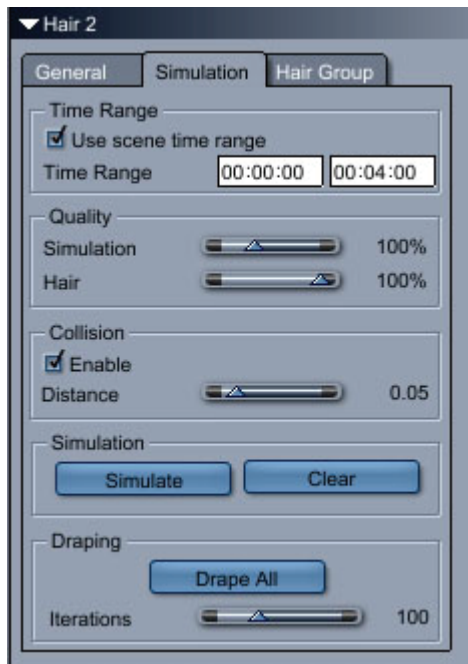
Simulation properties can also be accessed in the Assemble Room through the Simulation section.

- **Time Range Section**: This section determines the time range in the animation in which simulation will take place. You can enter your own values if you are working on a particular part of your animation.

  o **Use Scene Time Range**: Check this option to perform dynamic hair simulations throughout the entire range of the scene. Uncheck this option to enter a specific time range.

  o **Time Range**: Enter the start time and end time of the simulation in MM:SS:FF format.

- **Quality Section**: This section determines the quality of the simulation. Higher settings will produce better results but make simulations slower.

  o **Simulation**: Adjust this setting to raise or lower simulation quality.

  o **Hair**: Controls the percentage of hair segments to control during the simulation.

- **Collision Section**: This section determines whether the hair will be affected when it collides with other objects in the scene.

  o **Enable**: Check this option to enable collision detection. When disabled, hair will poke through objects that it collides with.

- o **Distance**: This setting controls how close the hairs should get to other objects before a collision is detected.

- **Simulation Section**: Controls in the Simulate section allow you to manually initiate simulation calculations.

  - o **Simulate**: Press this button to begin simulation.

  - o **Clear**: Press this button to remove the current simulation data. When you play an animation after clearing a simulation the hair will not move.

- **Draping Section**: Controls in the Draping section allow you to initiate draping of hair, and to convert draped hair to a new shape.

  - o **Drape All**: Press this button to drape the hair a little bit at a time. The hair will react to forces a little bit at a time. This setting is different from simulation in that it does not become animation data. You can press the Escape key at any point during the animation to use the current draped shape.

  - o **Iterations**: This setting determines the maximum number of times that forces are applied to the hair before it stops draping.

Hair Group Tab

The options you can set in the Hair Group tab are as follows:
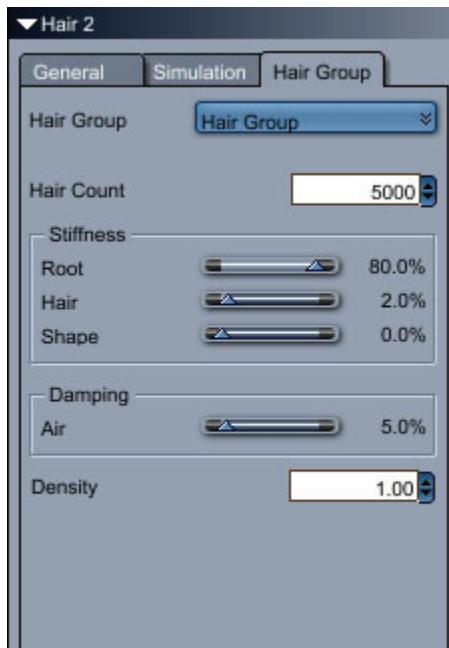
- **Hair Group**: Click this button to select a hair group to work with.

- **Hair Count**: Set the number of hairs that you want to generate for the hair group.

- **Stiffness Section**: Hair that has a higher stiffness setting will be less likely to change shape during simulation.

  - o **Root**: Controls the amount of bending from the hair's reference position.

  - o **Hair**: Controls how much the hair is allowed to bend.

  - o **Shape**: Controls how much the hair will try to keep its reference shape and position.

- **Damping Section**:
  - o **Air**: This setting controls how much the hair will interact or lose energy with the air.
- **Density**: This setting controls the number of hairs that are generated with each hair group. The number of hairs defined in the Hair Count section are multiplied by this value to determine the density.

Using Hair Shaders and Presets

To create or edit a hair shader, go into the Assemble Room. With the hair selected, click the Shading tab, and then click the Edit button to open the hair shader.

Hair shaders have different properties than those of other materials. There are properties that control the colors, shininess, and highlights of the hair. You can also control thickness, length, density, frizz, kink, wave, and clump in the shader properties.

The left portion of the shader window shows the various types of shaders that you can combine to control the appearance of your hair.
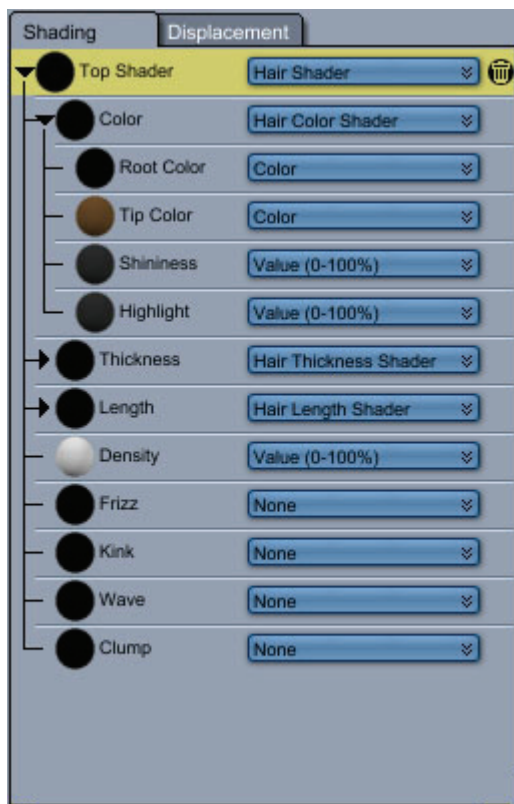
As you make changes to the hair shader, the preview window displays your changes. Check the **Auto Update** button to update your changes manually, or uncheck the option and click the Render button to see a preview as needed.

Controlling Hair Color with Shaders

When you expand the Color portion of the hair shader tree, you'll see several settings that control the appearance of the hair. These properties combine together to define the overall shade of the hair. The properties include root color, tip color, shininess, and highlight.
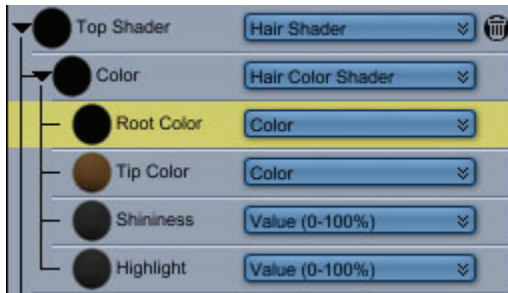
- **Root Color**: Defines the color that is at the root of the hair (closest to the polygons that generate the hair). Root color is generally darker than the tip color. Click the **Color** button to select an option that defines the root colors of the hair. If your object is UV-mapped and texture-mapped, you can also use a Texture Map node to vary the colors on the head. For more realism, select a root color that is slightly darker than the tip color.

- **Tip Color**: Defines the color at the loose end, or tip, of the hair. Click the **Color** button to select an option that defines the tip color. If your object is UV-mapped and texture-mapped, you can also use a Texture Map node to vary the colors on the head. For more realism, select a tip color that is slightly lighter than the root color.

- **Shininess**: Controls the amount of shine in the hair. Set the amount of shininess with Value sliders (0-100% or 1-10000%).

- **Highlight**: Defines the color of highlights that are generated by lights shining on the hair. Set the amount of shininess with Value sliders (0-100% or 1-10000%).

Styling Hair with Shaders

Hair shaders also include various properties that allow you to control the thickness, length, density, frizz, kink, wave, and clumpiness of the hair. These properties appear in the lower portion of the hair shader tree.
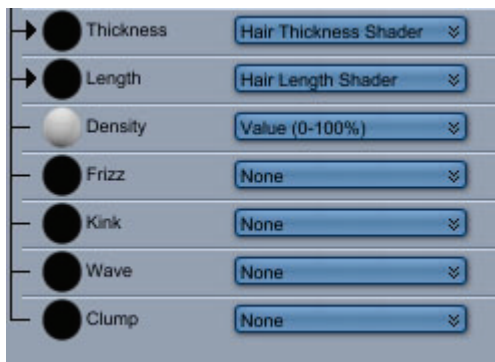
Hair Thickness

The Hair Thickness shader controls the thickness of each individual generated hair. For example, the hairs on a bristle hair brush are thicker than human hair. Set to **None** if you do not want to control thickness with a shader. Choose **Hair Thickness Shader** to set individual properties for root and tip thickness.
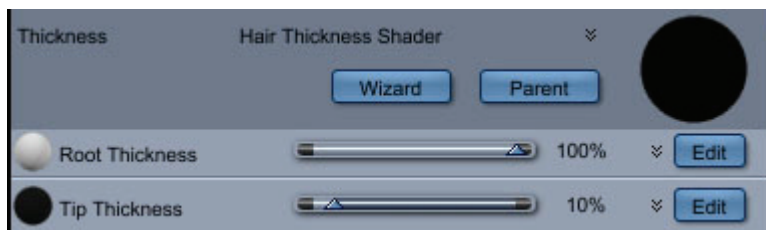
TIP! When using the Hair object to simulate grass, you can use this property to make the tips of the grass narrower than the roots.

To add or remove a hair thickness shader:

1. Expand the Hair Shader tree to reveal the Thickness shader tree.

2. Click the shader selection menu and do one of the following:

   - Choose **None** to remove a hair thickness shader.

   - Choose **Hair Thickness Shader** to add the shader to the property.

3. Set the following properties:

   - **Root Thickness**: Adjust the slider to increase or decrease the thickness of the hair root.

   - **Tip Thickness**: Adjust the slider to increase or decrease the thickness of the hair tip.

Hair Length

The Hair Length shader allows you to vary the length of the hair with a shader. Set to None if you do not want to control length with a shader. Choose **Hair Length Shader** to create a shader that determines the overall length of the hair, and the amount of length variation.
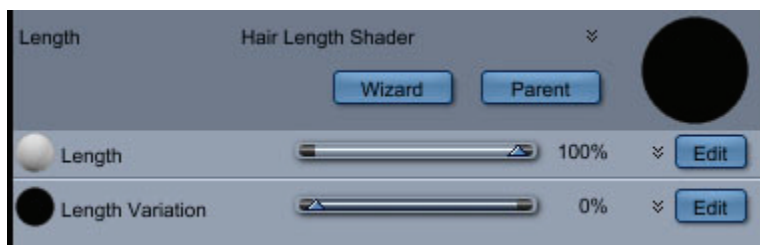
To add or remove a hair length shader:

1. Expand the Hair Shader tree to reveal the Length shader tree.

2. Click the shader selection menu and do one of the following:

   - Choose **None** to remove a hair length shader.

   - Choose **Hair Length Shader** to add the shader to the property.

3. Set the following properties:

- **Length**: Adjust the slider to increase or decrease the length of the hairs.

- **Length Variation**: Adjust the slider to vary the length of the hair by smaller or larger amounts.

Hair Density

The Hair Density shader allows you to control the thickness of the hair. This setting is multiplied by the number of hairs per hair group to determine the number of hairs that appear in a given area. When using a grayscale texture map, black is equal to a density setting of zero and white is equal to full density. Click the button to select an option for your density shader, or choose None if you do not want to control density with a shader.

To add or remove a hair density shader:

1. Expand the Hair Shader tree to reveal the Density shader tree.

2. Adjust the **Value** slider to decrease or increase hair density.
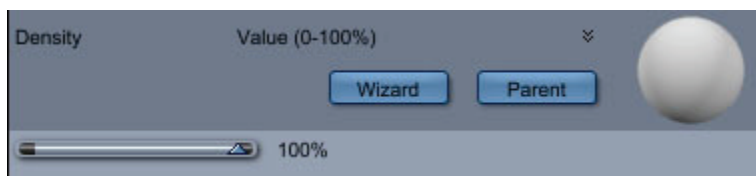
Hair Frizz

The Hair Frizz shader controls the frizziness of the hair. Choose **None** if you do not want frizzy hair. Choose **Hair Frizz Shader** to control the amount of frizz at the root and tip of the hair and to control the frequency at which the frizziness occurs.

To add or remove a hair frizz shader:

1. Expand the Hair Shader tree to reveal the Frizz shader tree.

2. Click the shader selection menu and do one of the following:

- Choose **None** to remove a hair thickness shader.

- Choose **Hair Frizz Shader** to add the shader to the property.

3. Set the following properties:

- **Root**: Adjust the slider to increase or decrease the frizziness of the hair root.

- **Tip**: Adjust the slider to increase or decrease the frizziness of the hair tip.
- **Frequency**: Controls how the shader varies on the surface. A high frequency means that the frizz will change faster on the surface.
- **Frequency Scaling**: Controls the scaling of the frizz.

Hair Kink

The Hair Kink shader controls the amount of kink in the hair. Choose **None** if you do not want to control the amount of kink in the hair. Choose **Hair Kink Shader** to control the amount of kink at the root and tip of the hair and to control the frequency at which the kink occurs.

To add or remove a hair kink shader:

1. Expand the Hair Shader tree to reveal the Kink shader tree.
2. Click the shader selection menu and do one of the following:
   - Choose **None** to remove a hair thickness shader.
   - Choose **Hair Kink Shader** to add the shader to the property.
3. Set the following properties:
   - **Root**: Adjust the slider to increase or decrease the kink amount of the hair root.
   - **Tip**: Adjust the slider to increase or decrease the kink amount of the hair tip.
   - **Frequency**: Controls how the shader varies on the surface. A high frequency means that the kink will change faster on the surface.
   - **Frequency Scaling**: Controls the scaling of the kink.

Hair Wave

The Hair Wave shader controls the amount of wave in the hair. Choose **None** if you do not want to control the waviness of the hair. Choose **Hair Wave Shader** to control the amplitude of the wave at the root and tip of the hair and to control the frequency at which the kink occurs. Frequency, amplitude, and phase are controlled along the X and Y axes.
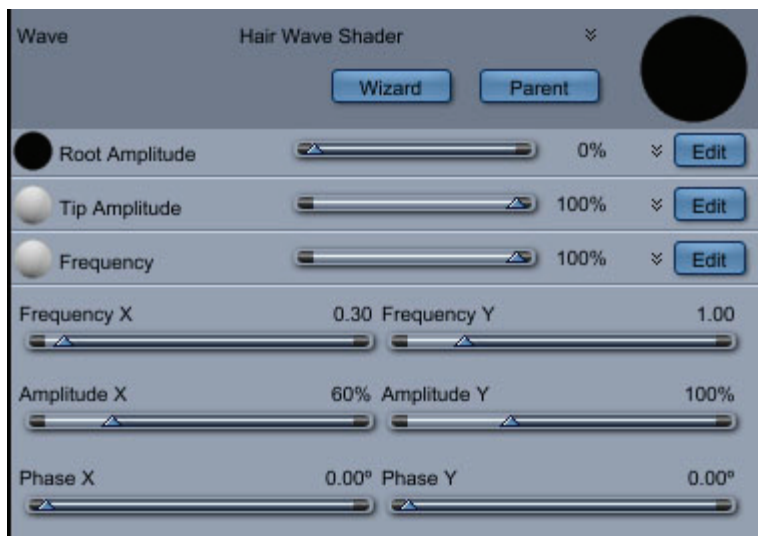
To add or remove a hair wave shader:

1. Expand the Hair Shader tree to reveal the Thickness shader tree.

2. Click the shader selection menu and do one of the following:

   - Choose **None** to remove a hair thickness shader.

   - Choose **Hair Wave Shader** to add the shader to the property.

3. Set the following properties:

   - **Root Amplitude**: Controls the size of the waves at the root of the hair.

   - **Tip Amplitude**: Controls the size of the waves at the tip of the hair.

   - **Frequency**: Controls the number of waves in the hair.

   - **Frequency X**: Controls the number of waves along the X axis.

   - **Frequency Y**: Controls the number of waves along the Y axis.

   - **Amplitude X**: Controls the size of the waves along the X axis.

   - **Amplitude Y**: Controls the size of the waves along the Y axis.

   - **Phase X**: Controls where the wave starts along the X axis, offsetting the wave along the X axis.

   - **Phase Y**: Controls where the wave starts along the Y axis, offsetting the wave along the Y axis.

Hair Clump

The Hair Clump shader controls how much the hair "clumps "or sticks together during simulation. Choose **None** if you do not want to control the clumpiness with a shader. Choose **Hair Clump Shader** to define the hair count, clump percentage, rate of clumping, and radius of clumpiness.
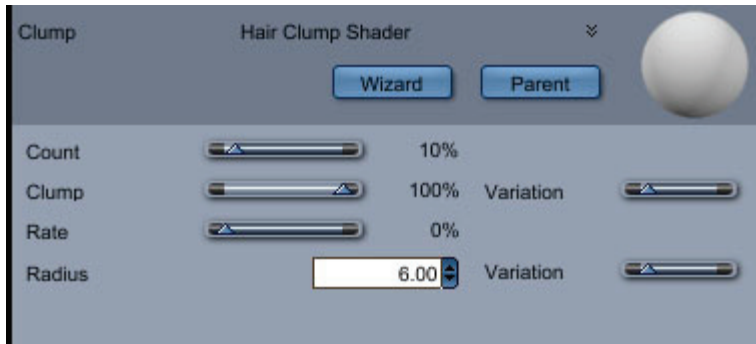
To add or remove a hair clump shader:

1. Expand the Hair Shader tree to reveal the Thickness shader tree.

2. Click the shader selection menu and do one of the following:

   - Choose **None** to remove a hair thickness shader.

   - Choose **Hair Clump Shader** to add the shader to the property.

3. Set the following properties:

   - **Count**: Controls the number of hair clumps that area created.

   - **Clump**: Controls how much of the hair is clumped together.

   - **Variation (Clump)**: Adds variation to the clumps created by the shader.

   - **Rate**: Controls the shape of the clumps (how quickly they clump together).

   - **Radius**: Controls the minimum radius of each clump, which also affects the count.

   - **Variation (Radius)**: Adds variation to the radius of the hair clumps.

Creating Hair on a Human Figure

The following example assumes that you have loaded a human character and have dragged the Hair icon onto it as outlined in "Adding Hair to an Object."

To create hair on a figure:

1. To remove hair from all parts of a figure, first click the **Grow Region tool** in the Hair Toolbar.

   TIP! You may find that it is easiest to start in Lasso selection mode to select a large area of polygons, and then use the Paint method to refine the selection.

   NOTE! When using the Lasso selection method, your selection may also include polygons that are facing away from the camera. Always view your object from multiple camera angles or rotate the camera as needed to verify your selections.

2. After you select the desired polygons, you can set hair properties in the General tab.

3. Set length, thickness, and hair scale in the General section.

4. Add additional guide hairs in the Guide Hairs section, or use the Add Guide Hair tool to paint additional guide hairs where you need them.

5. After you have the desired length and a sufficient number of guide hairs, you can drape the hair to get a more relaxed shape to start with. Use the Drape tool in the Hair Toolbar, or the Drape All button in the Simulation section of the Dynamics tab.

6. Use the hair styling tools in the Hair Toolbar to add curl, cut, straighten, and change density as needed to complete your style.

7. When you get a style that you want to keep as the basic hair style, go back to the Simulation section of the Dynamics tab and click the Convert button. This "sets" the shape of the hair for the simulation.

Rendering

About Rendering

Rendering is the process of capturing a view of a three-dimensional scene and saving it as a 2D image. An image has to be rendered before you can print it or open it in an image editing or page-layout program.

Carrara's rendering process can create photo-realistic images because it considers all of the objects in a scene simultaneously and calculates not just forms, color, and texture, but the interaction of lights and surfaces within the scene.

Since your Carrara scene exists in three dimensional space you can take any number of renderings of it:from different angles and even with different lighting conditions. If you create an object in a 2D drawing program and want to look at it from another side, you have to redraw the object. In Carrara, you simply move the camera to another view and re-render the scene.

A rendering is a separate file and is stored on disk in a different format. Carrara lets you save renderings in many popular image formats: PICT, Windows Bitmap (BMP), TIFF, Adobe Photoshop, and others. You can open your renderings in an image-editing program, or place them into virtually any application.

The G-Buffer is a special feature of Carrara that allows you to include special information in data channels that are saved with the rendered image. These channels can be manipulated in an image editing program to create special effects.

You can shorten rendering time by using multiple computers connected by a network.

You can render either a current scene or several scenes sequentially using the batch list.

Steps Before You Render

Before you produce a rendered image, complete the following steps:

To prepare an image for rendering:

1. Arrange the objects in the scene. Refer to "Arranging Objects" and "Working with Objects" for additional information.

2. If desired, apply shaders to give the objects interesting surfaces. Refer to "Shading" for more on shading.

3. Adjust the position and settings of the camera to frame the view of the scene you want rendered. Refer to "Positioning and Aiming Cameras."

4. Adjust Ambient Light, Background, and Atmospheric settings. Refer to Ambient Light for more information about setting ambient lights.

5.   Apply any desired filters.

6.   Verify that the scene is as you wish. You are ready to render the scene.

Using the Render Room

All of your rendering takes place in the Render room. You can either access the Render room explicitly through the Render room button at the top of the Carrara window, or by pressing Command/Ctrl+5 from another room.

Like other Carrara rooms, the Render room contains trays for Properties and the Browser. In addition, the contents of the Sequencer are replaced by a Render tray you use to select a file to render or to accumulate several files to render in a session. Refer to "The Properties Tray."

The Render tray is unique to the Render room. You can open image files directly from the Render room or open and modify batch lists.

The Render Tray

The contents of the Sequencer are replaced in the Render room by the Render tray. Use this tray to assemble files to be rendered and to manage the way they are opened, rendered, and saved.

The Render tray has two modes: Current Scenes and Batch Queue. When you click the Current Scenes tab in the Render tray, you'll see a list of the currently open files.

When you click the Batch Queue tab, you can set up a list of files for batch rendering.

Batch rendering also allows you to manage system resources by rendering one or more scenes when you are away from your computer.
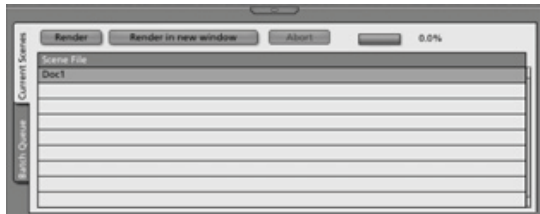
To render a scene, do one of the following:

•   Press Cmd/Ctrl+R to start rendering using the default render settings.

•   Click the Render room button or press Cmd/Ctrl+5 to enter the Render room and make additional rendering decisions such as adding additional effects or setting resolution.

Batch List

The batch list contains the list of the scene files you want to render. It can contain a single file or multiple files.
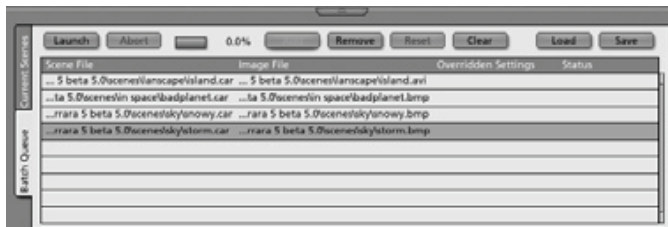
Current Scene Selections

When you select the Current Scene button, you have the two options: Render and Abort. Press Render to start the rending process and Abort to stop it at any point.

Batch Queue Selections

When you select the Batch Queue button, you switch the Render tray to Batch Queue mode and a selection of batch management controls display to the right (top) of the batch list. The selections are:

- **Add** adds another file or batch list to the current batch.
- **Remove** removes the selected file from the current batch.
- **Reset** sets all of the files ready to be rendered again.
- **Clear** clears all entries from the batch list.
- **Load** loads a previously saved batch.
- **Save** saves the current batch.
- **Launch** starts the batch rendering process.
- **Pause** allows you to quit Carrara and resume rendering at a later time.
- **Resume** allows you to continue after selecting Pause.
- **Abort** stops the batch rendering process.
- **Pause** allows you to quit Carrara, turn off your computer, and resume rendering at some later time. Movies are resumed from their last incomplete frame. Images are resumed from their last incomplete rectangle.

If you use Pause while rendering a file using lossy compression such as JPEG or GIF, double compression artifacts will appear.

To assemble images to render:

1. From the Render tray, select Add and Remove as needed to add files to be rendered to the Render list.
2. Select Reset to set all of the files ready to render again.
3. Select Clear to remove a selected image.
4. Use **Load** to load a previously saved batch file or Save to save the current batch as a batch list.
5. Select **Launch** to launch the batch rendering process or Abort to stop it.
6. Select **Pause** to pause the rendering, Resume to continue.
7. The Status indicator shows the progress of your renderings.

The Image Window

When you render a scene, the image is displayed in a window when rendering is done. If you want to keep this rendering, save the file using the **File > Save** command. The images are automatically saved as files when using the Batch Queue mode.

When you open a rendered image, it's displayed in the window shown below.

You can double-click a file name in the Batch Queue list to open that image.

The Properties Tray

The Properties tray contains most of the special rendering features available in Carrara.

The top of the Properties tray contains tabs for Renderer Settings, Output Settings, and Progress/Statistics.

Depending on your selection, different options appear in the main body of the Properties tray.

For example, for Renderer Settings, you can select a type of renderer. Once selected, you can then continue to choose options appropriate for that rendering engine.

Rendering the Current Scene

To render just the current (saved) scene from any room, you can go to the Render room and either render it with the current rendering settings, or customize the render settings in the Properties tray.

To render the current scene:

1. Create or open a scene.
2. Go to the Render room by selecting the Render icon or pressing Cmd/Ctrl+5.
3. Click the Current Scenes tab in the Render tray
4. Click the scene file name in the Render tray.
5. Apply any desired rendering settings in the Properties tray.
6. Select the file size, name, and format for the output. Refer to "Selecting Output Settings."
7. Click the Render button, or press Cmd/Ctrl+R. You can also press Cmd/Ctrl+R while in any room to render the current scene using the current settings.

8. Check on the status of your rendering in the Stats window. Refer to "Rendering Statistics."
9. Observe the rendered image in the Image window. Refer to "Batch List."

Rendering from a Batch List

You can render a batch of files or select a file from a pre-defined batch list.

To render from a batch list:

1. Go to the Render room by selecting the Render icon or pressing Cmd/Ctrl+5.

2. Click the Batch Queue tab

3. Do one of the following:

- Click the Load button to load an existing batch that has been previously saved to your hard drive.

- If desired, you can add or remove files from the batch with the Add and Remove buttons.

4. If you wish to change the render settings on any file, click it, then apply any desired settings in the Properties tray. Refer to "The Properties Tray".

You can specify different settings for each file, or add the same file more than once and set up different settings for each instance. This allows you to render several variations of the same file in a single batch.
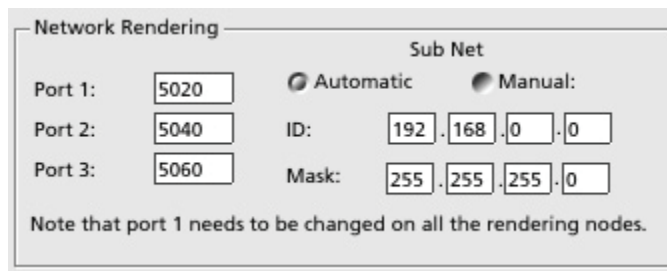
The settings you specify in the batch list are separate from the current settings which are used when you press Cmd/Ctrl+R.

5. Select the file size, name, and format for the output of the selected file. Refer to "Selecting Output Settings".

6. When you have finished setting up all of the files in the batch that you wish to modify, click Launch. Refer to "Current Scene Selections".

7. Check on the status of files you are rendering in the Stats window. Refer to "Rendering Statistics".

8. Observe the each image as it is rendered in the Image window. Refer to "Batch List".

9. Double-click a file name in the batch list to open the image file.

Rendering Preferences

The options available in the Render Room Preferences dialog are as follows:

- **Batch Rendering**:
  - **File Name Prefix**: Enter a prefix to be added to all files that you render.
  - **File Name Suffix**: Enter a suffix to be added to all files that you render.
  - **Network Rendering**: In the preference panel for the render room, you can edit the ports that are used for the network rendering communication.

  - **Manual or Automatic**: If the automatic configuration does not work, you can manually set up the subnet ID and mask.
  - **Ports**: The ports should have a value between 5000 and 65535. Port 1 must match the port number that you set in the preferences of the rendering node you plan to use.
  - **ID** and **Mask**: The ID and the subnet mask are given by the IP address of your machine on the network. The ID and subnet mask can be obtained on Windows by typing "ipconfig" in Command Prompt window.

- **Don't Overwrite Rendering Output Without Permission**: Check this option to prevent overwriting files of the exact same name.

- **Always Render in a New Window**: Check this option to start all renders in a new window.

- **Check Before Overriding Rendering**: TBD

Network Rendering (Pro)

Even though the rendering engines of Carrara are extremely fast, Carrara allows you to use other networked computers to speed up rendering.

Setting up a Network

To perform network rendering, you need access to computers on the same network. This can be done using a simple Ethernet hub or router. Make sure that all the computers are located on the same IP subnet (this is usually the case as long as you have only one hub or router on your network), because Carrara will look for rendering nodes only on this subnet.

Once your network is in place, you need to install a rendering node on each machine that is going to take part in the network rendering (except the machine on which you are using Carrara). Then you need to start the rendering nodes so that they are waiting for the rendering to start.

Rendering on a Network

After the rendering nodes on your networked machines are started, you are ready to launch network rendering.

To launch a network rendering:

1. Add the file you want to render to the batch list (see "Batch List").

2. Select the rendering settings and output options.

3. In the Properties tray, check **Enable Network Rendering** .

4. Click on the **Launch** button.

   The rendering is automatically launched on all the rendering nodes that are available (up to the number authorized by the license you purchase). For each computer actually rendering on the network, you should see a tile with an 'N' (for Network).

Troubleshooting Network Rendering

There are a number of reasons why network rendering might not work:

- Make sure that you are rendering in the batch queue and that enable network rendering is checked.

- First make sure that your computers can see each other on the network (check the "Computer near me" or Network Neighborhood).

- Make sure that the rendering node is using the same port (Port 1) as the main application (see "Rendering Preferences").

- Use the log feature to see if the rendering nodes are properly registered by the main computer. If they are, they should show up in the network log file.

- Make sure that the ports used for the network rendering are not blocked by a firewall or already in use by some other application. You can try to change to ports to see if it changes something.

Using Post Render Filters

Rendering with an Aura

Aura lets you render true glowing objects with an outside aura. The light emitted from the glowing objects can interact with fog or turbulent smoke.

Aura automatically detects which objects have a glow shader, letting you create impressive effects like laser beams, neon signs, or LEDs.

Aura uses the shader Glow channel to know which object are glowing and which are not. Only objects that have some degree of luminance in the Glow channel receive an aura.

To set up the glow channel for an object:

1. Select the object you want have an aura.

2. Click the **Texture** button to go to the Shader room.

3. In the Shader Tree Editor window, make sure the top shader is a Multi-Channel shader.

4.   Click in the Glow channel.

5.   Adjust the Glow settings as desired.

NOTE! For details on using the Glow channel, refer to "Glow". (Standard, Pro)

To apply an aura to a scene:

1.   Click the working box to deselect any objects in the scene.

2.   In the Properties tray, click the Filters tab.

3.   Click the plus (+) sign, and choose 3D Aura from the pop-up menu. The 3D Aura editor window appears.

4.   Drag the Radius slider to set the radius of the aura. Radius describes the number of inches before the aura fades out.

5.   In the Turbulence section, check the Enable box if you want swirls in the aura medium (smoke wreaths).

6.   Drag the Turbulence Scale slider to set the mean size, in inches, of the wreath of smoke.

7.   Drag the Lumpiness slider to set the contrast level in the wreath of smoke.

8.   Enable the Animate option if you want the medium to move (change over time). This option slows down the rendering. Use it only when doing animations.

9.   To change the color of the aura, click on the color chip under the preview window. Carrara opens the system color picker so you can choose a color. The color of the final aura effect results from the interaction of the object's glow color and the aura color. Higher luminance values result in a brighter glow and, therefore, a stronger aura.

10. Drag the Intensity slider to change the strength of the aura effect.

Rendering with Glare
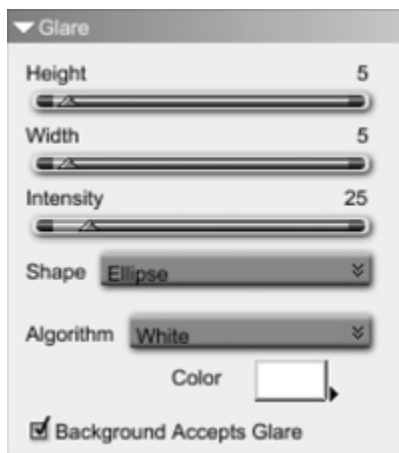
To set glare options:

*   **Height** and **Width** controls the size of the effect.

*   **Intensity** controls how much glare is added.

*   **Shape** defines the shape of the glare pattern.

*   **Algorithm** specifies which algorithm is used to calculate the glare.

- **Color** controls the color of the halo when the Algorithm is White.
- **Background Accept Glare** controls if the glare bleeds over to the background.

Motion Blur (Standard, Pro)

When you take a family picture and someone moves at just the wrong moment, that person's image is blurry. In films, fast moving objects like the wheels of a car will often be blurred. This effect is called Motion Blur.

The Motion Blur post render filter combines successive frames of an animation to achieve that real-life blurred effect on moving objects.
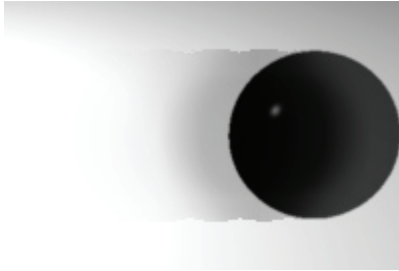
When rendering fast moving objects, 24 or even 60 frames per second are often not enough frames to capture the full complexity of the motion. For example, a spinning propeller can complete a full rotation between 2 consecutive frames.

In a real video camera, the camera shutter remains open for a short period of time and a moving object creates a blurry imprint on the film along its trajectory. This blur along the motion of an object is called Motion Blur.

One could think that Motion Blur is an undesirable artifact of camera hardware. However it is an important part of what makes a movie look real. Without it, fast motions look unrealistic. Carrara allows you to simulate motion blur by rendering extra frames and combining them together to create a similar effect to what a real camera generates.

To add the Motion Blur filter to your scene:

1. In the Assemble room, select the Instances tab of the Properties tray.
2. Click on the first line of the hierarchy to display the scene properties.
3. Select the Filters tab then use the Plus (+) button to add the Motion Blur filter.

To set Motion Blur options:

- **Intensity**: controls how many frames will be used to compute the effect
- **Blur**: enables the blur of the frames used for the effect.
- **Blur Radius**: controls the strength of the blur
- **Render First Frames**: Renders the first frames of the animation when checked. When unchecked, the first frames are replaced by a black image until enough data has been gathered.

Using Motion Blur

The Motion Blur panel is located in the Properties tray in the rendering rooms under the Rendering tab:

To add motion blur to your rendering:

1. Check the **Enable Motion Blur** check box.

2. Select the number of extra frames you want to render.

3. Adjust the motion blur settings.

4. Render your scene.

Vector and 3D Motion Blur (Pro only)

Carrara offers two types of motion blur: Vector and 3D Motion Blur. You can use either one alone or both types can be combined together.

3D Motion Blur uses a number of extra frames to simulate the motion blur. It provides extremely realistic results because it takes into account every single effect that is affected by time (such as a moving shadow, a reflection, or a deforming object). However it often takes a great number of frames to get a good result. When the number of frames is too low, you get a very unpleasant effect.

Vector Motion Blur allows you to render fewer frames and still get a correct motion blur effect. It works by interpolating the motion of objects between two frames to create a smooth looking output. You can combine it with 3D Motion Blur to get better results simply by adding more frames.

Vector motion blur has a number of limitations:

*   It only deals with moving objects and not with shadows, reflections,...

*   It does not handle deforming objects properly.

*   It can only be used with the photorealistic renderer.
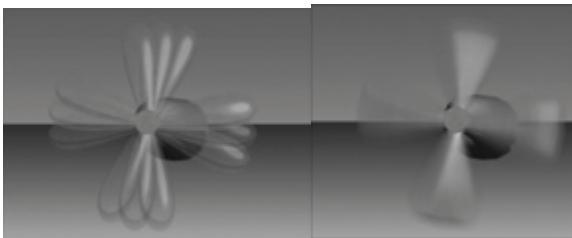
To set Vector Motion Blur options:

1. Check the **Vector Blur** checkbox.

2. Select **Backward and Forward** blur option to blur the motion in both directions around the position of the object at the exact time of the frame. (This is what a real movie camera does).

Select **Backward Only** or **Forward Only** to create a blurry trail only after or before the object (This is especially useful to give the impression of movement in a still image).

3. Adjust the **Extra Frames** slider to control how many extra frames will be rendered to simulate the blur.

4. Adjust the **Blur Intensity** to control how much motion blur will be applied.

Motion Blur at time 0

You can also use motion blur on still images. However you have to be careful that the motion of the object exists before and after the frame you are rendering. For that reason you should render a frame at a time different than time 0 to get the full motion blur effect.

For the same reason, you should not start rendering an animation at time 0. The proper way to render an animation with motion blur is to start your animation at a later time. You can do that by moving the start time of the rendering in the Sequencer tray.

Cartoon Rendering

Toon! is a post-render filter for Carrara that changes your rendered output into a cartoon drawing. Here are some of the features of Toon!:

- Fully supports shaders

- Plays well with other filters

- Can group multiple objects together for the purposes of tracing the outline

- Mixes traditional rendering and Toon!**ing** in the same render

To set Toon! options:

- **Line** sets the color of the lines. **Shadow** sets the depth of the shadows.

- **Base** adjusts the base color. Highlight adjusts the highlight color.

- **Toon Effect** controls the mix of how much Toon! obscures the original render. While **Line Effect** controls the mix of how much Toon!'s lines obscure the original render.

- **Shadow Size** controls the size of the shadow. While **Highlight Size** changes the size of the highlight.

- **Object Edge** traces the edges of your models, Sharp Edges traces the edges of objects, and Edge in degrees controls how sharp the edge must be before drawing a line.

- **Distance** puts a line where there is a change in distance from the camera.

- **Distance Threshold** controls the difference in distance from the camera that pixels must be before drawing a distance line.

- **Shadow Edge** traces the edges of shadows. While **Highlight Edge** outlines specular highlights.

- **Algorithm** reduces the jaggies in Toon!'s lines. Selections are None, Toon 1, Toon 2, and Custom.

    o **Toon** 1 is the original Toon! algorithm. It's slower, but doesn't fatten lines.

    o **Toon 2** is newer, faster, and smoothes better, but fattens the lines. Custom with the default filter smoothes better than Toon 2, but doesn't stay as dark.

- When **Custom** is selected, you can adjust how it smoothes using the nine values below the Algorithm popup. The values control how much line, shadow, or highlight effect from surrounding pixels is mixed with the middle pixel's effect.

- **Passes** controls how many passes of smoothing are applied when **Algorithm** is set to Toon 2 or Custom.

Getting started with Toon!

To add a post-render filter to your scene:

1. Select the scene on the Properties palette.

2. Select the Filters view.

3. Use the Plus button to add the Toon filter.

To Group things together:

Toon can group items together for the purposes of tracing an outline. It's not always convenient to build your models from one giant mesh. But if you want Toon to treat it as one mesh, you can use Toon Group.
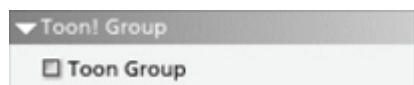
1. Select the items you want together.

2. Use the Edit>Group menu to group the selected items.

3. Select the new group in the Object Hierarchy or in the 3D View.

4. On the Properties palette, select the Effects view.

5. Check the Toon Group checkbox.

To mix Raytracing and Toon:

Toon! can mix Raytracing and the Toon effect. The Toon Override Effect allows you to control the Toon and Line Effects on an object by object basis.
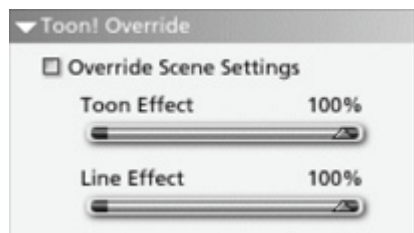
To make Toon! go faster:

Every effort has been made to make Toon! as fast as possible, but there are some options you can set when you really need the speed.

- Turn off the **Smooth Lines** option. On animations with compression, **Smooth Lines** may not be necessary.

- If you do use line smoothing, Toon 2 is much faster than Toon 1.

- Lower your **Line Widths** - each time you double the line widths, Toon! must examine four times as many pixels.

- On your render settings, turn off Adaptive Oversampling.

Rendering Effects

Rendering effects that become a permanent part of the scene file are applied in the Assemble room.

Effects that are applied solely for the rendering and that do not change the original scene file are applied in the Render room and become part of the output file settings.

For example, if you would like to apply shadows or bump, you select them through the Properties tray in the Render room. The settings become a permanent part of the original file as render settings. If you wish to apply fog or backgrounds, you must apply them in the Assemble room, then switch to the Render room to render the image.
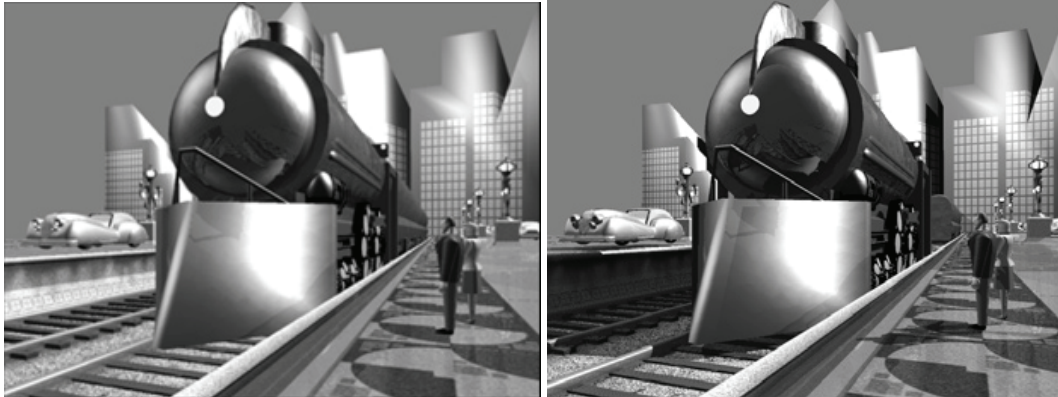
The Rendering Procedure

After you've built a scene, set lights, and chosen a viewpoint, you are ready for rendering.

You will probably render your scene several times at different resolutions as you test your scene and make adjustments.

The first few renderings will probably be low resolution proofs, so you can check your work.

At each proof stage, you can change your Rendering Settings, depending on the image aspects you want to check. For example, if you want to look at just the objects, you might use the Draft Renderer. Or if you are concerned only with shading and shadows, you might turn off the other ray-tracing options, like reflections and transparency, which add to rendering time.

After making all the adjustments you need in order to create the desired image, you can generate the final high-resolution rendering.

Enhancing the Rendering

Carrara offers a number of options and settings to control and enhance rendering. To get the rendering results you want, you must make several choices and adjustments.

For example, you may want to:

- Change the camera or render from several camera views. For details on cameras, refer to "Setting Cameras."

- Set compression or render to several file types.

- Include the scene in a batch to render with other scenes.

- Change the renderer, including quality and features.

Your choices will be based on your expectations:whether you are creating a draft rendering to check the objects, rendering final artwork, or rendering an animation.

All of these choices are available in the Properties tray of the Render room.

Speeding up Rendering

There are several techniques you can use to reduce your rendering time. These techniques are especially useful when rendering animations that may have hundreds, or even thousands, of individual images to be rendered.

- If a Spline object is distant from the camera, open it in the Spline modeler and lower its surface fidelity to the minimum. This results in the minimum amount of RAM being used for its geometry. (Standard, Pro)

- When an object leaves the frame, and isn't reflected in any surface that you can see in the rendering, activate its Cloak property. This won't save memory, but it does save rendering time. Objects that pass behind solid objects can be cloaked to save time, too. Be careful that you do not lose a cast shadow where one is needed.

You can speed up rendering by applying these techniques:

- Use the Draft Renderer to preview your object positions and movement.

- Work with smaller image (frame) sizes. The standard for CD-ROM-based QuickTime movies is half-screen size, or 320 by 240 pixels, although satisfying results can be achieved at 240 by 180. (You can then scale up to full-screen size for VHS output, although pixilation may be apparent when you scale up.)

- Limit the number of frames per second to 15, which is adequate for QuickTime movies. (Video uses 30 fps and film 24 fps.)

- Simplify objects and reduce the number of objects in the animation.

- Limit the number of lights.

- For the Photorealistic Renderer, disable the Reflection, Transparency and Refraction options.

- Use low detail accuracy (2-4 pixels) to speed up rendering.

- Keep objects near the background simple. (A complex bitmapped background does not affect rendering speed.)

- Limit the size of texture maps.

- Limit the number of reflective and transparent objects.

- Divide animations into smaller files and eliminate objects that won't be seen on-screen.

- Turn off anti-aliasing when previewing rendered animation.

- Close any opened Modeling windows to free up memory.

Computer System Optimization

Carrara's performance is affected by available RAM and disk space. You can never have a fast enough computer, enough RAM or disk space, a large enough monitor, or a fast enough computer. With that in mind, here are some things you can do to get the best performance out of your system.

If you have a scene that uses more RAM than you have, Carrara uses its own virtual memory system to keep things running, spooling data back and forth to the hard disk. The greater the disparity between

RAM needs and what is available, the more speed it costs you. If the scene ends up dipping into the operating system's virtual memory as well, performance can be very poor. In this case, adding RAM can improve speed, although this makes the computer run more efficiently, not any faster.

The fastest rendering takes place on a fast processor with plenty of RAM. Systems used in professional animation often need 128 MB RAM or more.

Because all rendering involves some spooling to disk, you need a large hard disk with plenty of free space.

To keep your hard disk running efficiently, use disk utilities to scan for disk errors, and defragment your hard disk prior to starting a project or even a long rendering session.

Don't run other programs in the background. Some screen savers use as much CPU time as Carrara.

Renderings generated from the Batch Queue are automatically saved to disk. You can open these files in Carrara by double-clicking on the file icon, or by double-clicking the name in the Batch Queue list. Or, you can open and view them in another graphics application.

Selecting Output Settings

You must select the image size, camera, file format, file name, and any G Buffers before you render.

Image Size

The resolution of a device, such as a monitor, printer, scanner, or image setter, is described as the number of pixels it can create for each inch of image area: pixels (or dots) per inch (PPI or dpi).

(For example, the screen of an Apple color monitor has 72 dots per inch (dpi). Each dot is a pixel, so an image that is 72 x 72 pixels would be one inch square on the screen of that monitor. VGA monitors have slightly higher resolutions. Many color printers are capable of 300 dpi, and some image setters can produce more than 2,500 dpi.

This discrepancy in device resolution creates a slight problem:pixels are different sizes on different devices, so an image displays or prints at a different size on devices of different resolution. For example, the 72 x 72 image that appears 1 in. square on the screen would be less than 1/4 in. when printed on a 300 dpi printer. (72 pixels drawn at 300 dots per inch: 72/300 = 0.24 in.)

The solution is to set the rendering parameters according to the size of the image you want from a particular output device.

The original size, resolution and pixel aspect ratio of your image appears in the fields in Image Size selection of the Properties tray.

You can accept the default size of the image, or enter new information to change the image size and other size-related properties.

To set the image size and resolution:

1. Choose the **Properties tray: Output Settings button** .

2. Select the Image Size panel.

3. In the Width and Height entry boxes enter the dimensions and units you want for the picture. Set the size according to your final output.

4. In the Resolution entry box, enter the resolution (dots per inch) of the expected output device.

   A six-inch square image at 600 dpi may occupy as much as 100 MB on disk. (The size varies depending on file format and image content.)

5. Enable the Keep Proportions option if you want to keep the same aspect ratio (ratio of width-to-height) when you change either the width or height.

6. Select a Pixel Aspect ratio if you are working on a system that does not use square pixels.

   The ratio is the width of the pixel divided by its height. A ratio of 1.5 specifies large but not tall pixels. A ratio of 0.75 specifies slim, tall pixels.

7. Verify the properties by examining the automatically generated information displayed under Properties in the Image Size panel.

Rendering Camera

You can render from any of the cameras in the scene. By default you will render from the current camera. Refer to "Setting Cameras" for more information about creating and using cameras.

To change the camera, you use the Camera option in the Properties tray.

To select a camera view:

1.   Choose the **Properties tray: Output Settings button** .

2.   Select the Rendering Camera panel.

3.   Select the Camera button.

The list displays all cameras in your scene.

NOTE! If you want to render one scene several times from different cameras, use the Batch Queue.

File Format

Before you can render, you must select a file format for the output, as well as other information needed to save in the selected format. You can render either the current frame or a movie.

The File Format controls let you choose the type of file saved to disk and set options for it. These controls also let you specify the frame range and rate for rendering animations.
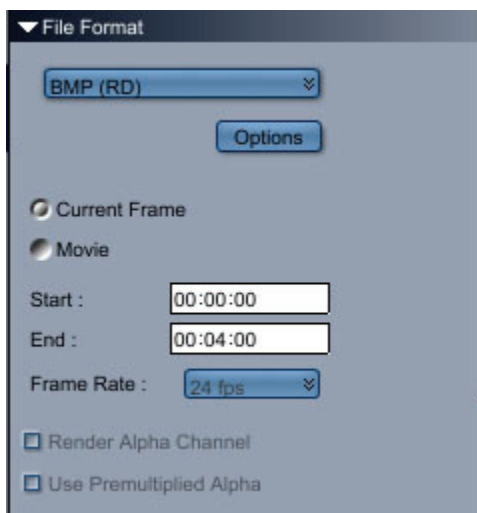
To select a file format for a still scene:

1.   Select **Properties tray: Output Settings button**.

2.   Select the File Format panel.

3. Select Current Frame.

4. Select a file type from the list.

5. Click Options. Carrara displays a dialog that lets you set features for this file format.

The Alpha Channel

The alpha channel (also referred to as a mask in other 2D and 3D software) is an extra channel that can be saved in your image file. It is a "shadow" image of your scene. It describes where objects are versus where they are not.

The alpha channel is used as a selection of your image when you paste onto a background in an image-editing program. Pasting a foreground image onto a background is called compositing.

Certain effects such as lens flares or light cones will also appear in the alpha channel allowing you to create nice transparency effects when compositing with another image.

Refer to your image-editing application's documentation for instructions on selections, masks and compositing.

Carrara puts the alpha channel data in channel #4, the Alpha channel, which is where masks are usually kept in applications like Painter and Adobe Photoshop.

To render the alpha channel:

1. Select a File Format that supports alpha channels (The checkbox for the alpha channel will be enabled when you select such a file format. You can use Tiff for still images or QuickTime for movies, for example).

2. Select **Render Alpha Channel** to enable alpha channel rendering.

3. Select **Use Pre-multiplied Alpha** if you need the alpha channel to be pre-multiplied into your image.

Most image-editing applications (including Adobe Photoshop) use standard alpha and not pre-multiplied alpha. Refer to the documentation of your image-editing or movie-editing application to know if you need to use pre-multiplied alpha.

Name File

Use either the default file name or select a file name. If you use the default, you automatically write over the existing file. If you select In Named File, you must choose the file name and storage location.
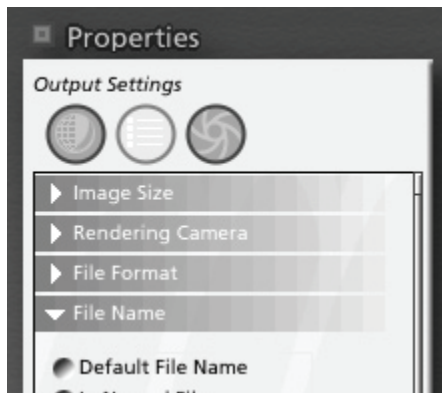
To name your rendered file:

1. Select the **Properties tray: Output Settings button** .

2. Select the File Name panel.

3. Select either Default File Name to write over the existing file or In Named File to create a new file. Default File Name only automatically writes a file when you are rendering using a batch queue.

4. If you selected In Named File, select a path and enter a file name.

Multi-Pass Rendering (Pro)

Carrara 7 Pro adds multi-pass rendering capabilities which enables you to render multiple passes and save these off to separate image files. These render passes or layers (multi-pass render layers) can be edited in another application like Photoshop to increase contrast in the rendered image, change a reflection color, make an edit to a shadow map, or even swap out a background color.

This tool makes it faster to make changes to a rendering which if you needed to re-render to see the changes it could take hours and hours. The edits to layers can be changed in minutes.

To use multi-pass rendering:

1. Choose the **Properties tray: Output Settings tab** .

2. Set your image size, file format, etc. Select Photoshop format to have the layers rendered to a single Photoshop file with rendered layer as a Photoshop layer. Selecting another image format will render the layers out as individual images which can then be combined in any paint program.

3. Select the **Multi-Pass** panel (all the way at the bottom).
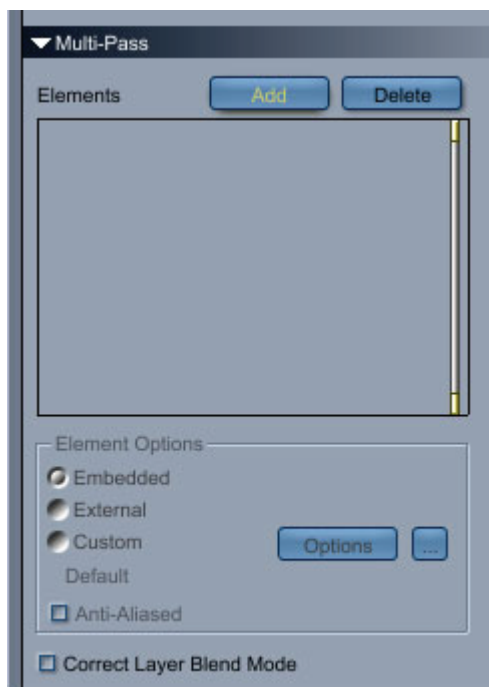
4.   Select **Elements** -

a.   Click on the **Add** button. The **Add Multi-Pass Layers** dialog will come up.
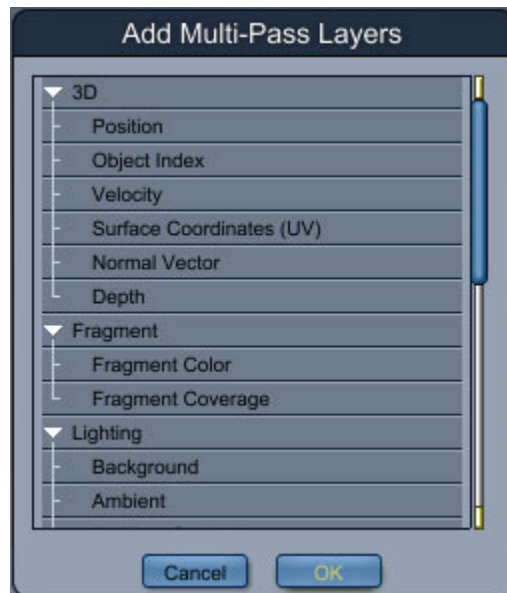
b.   Select which layers to add to the Multi-Pass Render. Use Ctrl+click to select multiple layers.

Select individual layers or the group layer, i.e., select Material to add the ambient, diffuse, and specular layers all at the same time.

The layers will be added to the Elements listing.
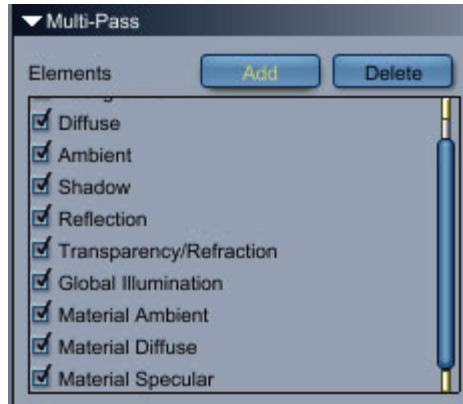
    c. To delete layers, select the layer and click on the **Delete** button.

    d. To not render a layer but keep it in the listing, click in the check box to deselect it. This is useful when you only need to re-render a specific layer but need to keep the remaining layers available for later rendering.

5. Select **Element Options** -

- **Embedded**: When selected, will save the renders into a single image file using multiple layers. Make sure the selected file format will support layers, such as Photoshop.

- **External**: When selected, will save the renders as a separate image file for each multi-pass element using the same file format used for the main image.

- **Custom**: When selected, will save the renders as a separate image file for each multi-pass element using the selected file format. With Custom, each layer can be saved as a different file type.

6. To not use anti-aliasing on a layer, click in the check box next to **Anti-Aliased** to deselect it. Note that this option is not always deselectable depending on the layer type. This option is available to assist with mask creation since the edges are not anti-aliased and if the items on as specific layer should not be anti-aliased.

7. To use the **Correct Layer Blend Mode** option, click in the check box next to it to select it. This option may produce better blending results in some paint applications. By default, this option is not selected but if you have layer blending issues in your paint program, select this option for the layer having the blending issue.

8. Click on the **Render** button.

Example of the rendered layers as separate images:
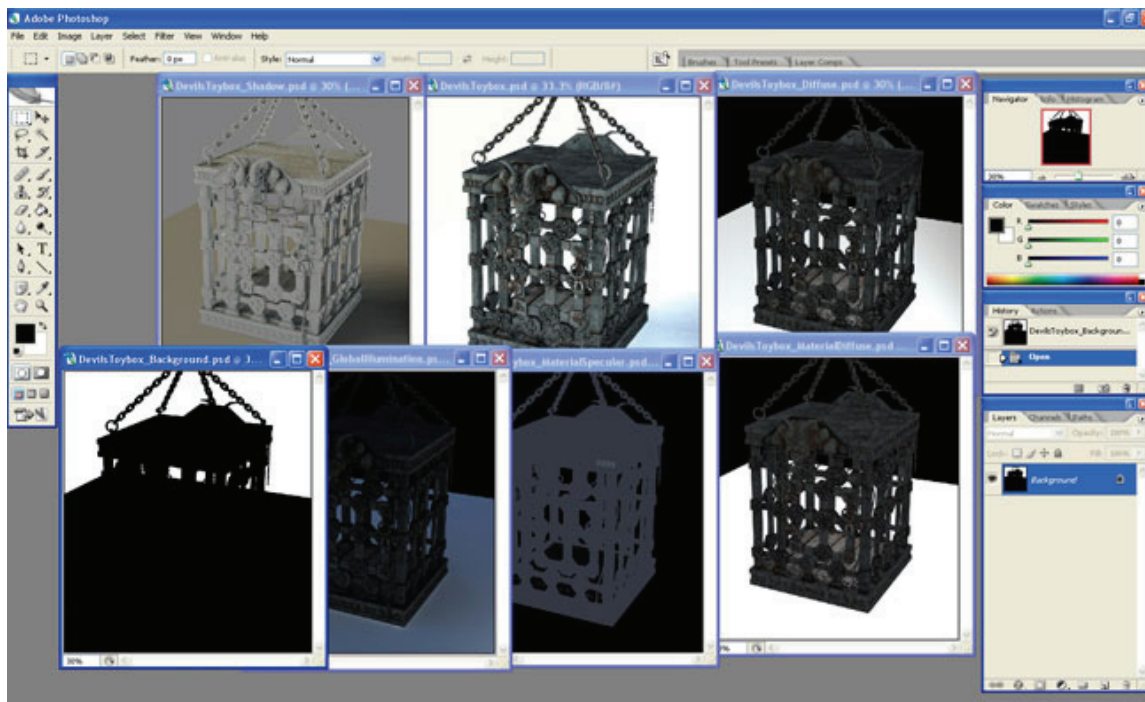
G-Buffers

Carrara can calculate and save several separate channels of geometric and lighting information in addition to your full color rendered image. These are called G-Buffers.
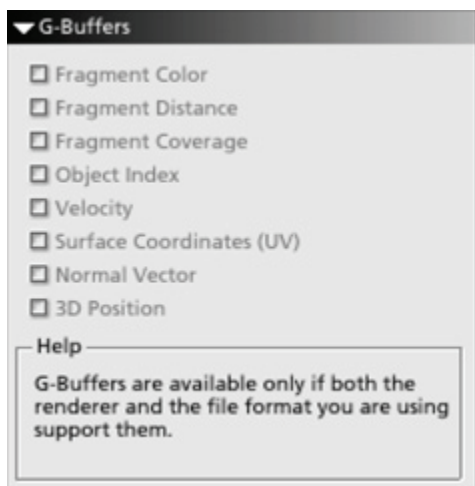
The data in each of these channels describes the point in your 3D scene that each pixel in the image represents. For example, if the scene shows a drinking glass on a wooden table, each pixel in the rendered image corresponds to a point:on the glass or on the table:in your 3D scene.

There are two types of G-Buffers:

- Image G-Buffers: These additional channels appear as grayscale images, where the gray values represent information.

- Actual G-Buffers: This buffers store values in a format that represent the actual value of the channel. (For instance, for the distance it would store a floating point value for each pixel.)

Not all file formats are capable of storing all of the channels possible.

To save Actual GBuffers, you should save to the RPF format.

To save image G-buffers you need to save to Adobe Photoshop or TIFF formats.

G-Buffer Options

The G Buffer options are:

- **Fragment color:** This is the color of object that is the closest to the camera.

- **Fragment Distance:** This is the distance of the object that is the closest to the camera. Lighter pixels are closest to the camera, while dark colors represent areas of the rendering that are farthest from the camera.

- **Fragment Coverage**: This is the percentage of the pixel that is covered by the object closest to the camera.

- **Object index** The object index relates each pixel in the image to the object its corresponding point belongs to. With an object index loaded in Adobe Photoshop, you can easily select individual objects with the Magic Wand, regardless of color. This works as long as the objects do not overlap.

- **Normal vector** The normal vector creates three channels describing the direction that each surface of the object faces. This information can be used after rendering to simulate additional light sources. For example, it's possible to add directional lighting or glows in Photoshop by loading the normal channel to select all the surfaces that face in a given direction, complete with information about how those facings fall off.

- **3D Position** The 3D position describes the coordinate of each point in the image. 3D position uses three channels, one for the x value, one for the y value, and one for z. An example of how you could use this information is to position low hanging clouds, making use of the Z, or height, channel.

- **Surface Coordinates** The surface coordinate is the location of the point on the object in relation to the object's surface coordinate system. This is two-dimensional information which allows adding, replacing, or repositioning texture maps on 3D objects in an image editing program.

To select G-Buffer channels:

1. Select the **Properties tray: Output Settings button** .

2. Select the G-Buffers panel.

3. Enable the checkbox or each feature you want.

Do not enable channels that you don't need. Each channel adds to file size considerably.

Choosing Movie Compression Settings

Rendering is computationally intensive, and depending on the render settings, jobs may take several hours to complete. If you have enough memory, you can continue working on other jobs and render in the background.

When using applications such as word processors or 2D drawing programs, the CPU is not using maximum capability and spends a lot of time idling. Carrara takes advantage of this by taking control of the CPU and performing its calculations in the background. Carrara's renderers are designed so that you can use other applications while rendering.

Rendering animation requires a significant amount of hard disk space. If you have disk space limitations, you will want to compress the movie that you render. If you have plenty of disk space, you may want to render without compression. This way you won't invest rendering time only to find out that you used too much compression. You can then re-open the movie file in Carrara and save it with compression.

The software compression and decompression algorithms (called codecs) that you can select in the Compression Settings dialog are provided with QuickTime (Macintosh or Windows (if the player is installed)) or AVI (Windows). Codecs compress data when you render an animation and decompress the data when you play the movie. Compression makes the file smaller, but degrades image quality. When you choose a compression option, you are always choosing between image quality and file size.

Any Macintosh or Windows system with QuickTime or AVI software can play back a compressed movie.

If you are using hardware for MPEG compression, see the instructions that accompanied your board for a description of available compression options.

To select a movie compression option:

1. Display the Properties tray.

2. Click the Output Settings button.

3. Make sure Movie is selected and choose a Movie file type such as QuickTime or AVI.

4. In the File Format controls, click the Options button. The Compression Settings dialog appears.

5. If the movie type you selected doesn't support compression, other options will display.

6. Select one of the compressor options from the Compressor list.

NOTE! Your actual compression may vary, depending on the version of QuickTime or Video for Windows you have installed. For additional information, refer to your Apple or Microsoft documentation.

Macintosh Compression Options
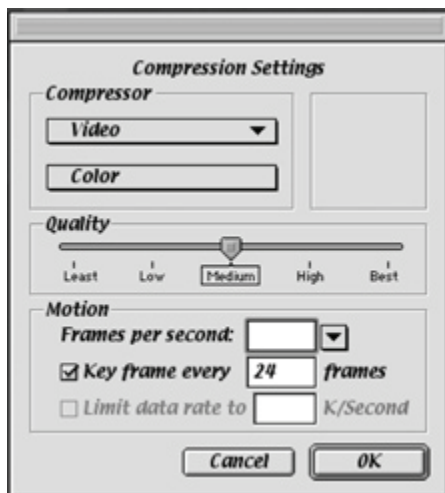
- **None**. This is a loss less form of compression with a 2 to 1 (or less) compression ratio. This codec converts an image's pixel depth from 32-bit to 24-bit image format with no loss in quality by combining the alpha channel byte. None is a good option for raw animation if you have plenty of hard disk space. You may want to use this as an intermediate step if you plan to edit your animation in Adobe Premiere because Premiere re-compresses the movie (avoid compressing twice). None is better for reachieving when the highest image quality is important.

- **Animation** . This codec works best for 8-bit animation and computer-generated screen images (as opposed to videotape images).

- It's often used for 3D animation. The image quality (especially at higher quality settings) is acceptable for informal movies. With this codec, you can create movies up to full-screen at any color depth. The Animation codec saves the first image and then saves only changes from one frame to the next. It also cuts out the alpha channel, resulting in increased speed and smaller files.

- **Cinepak**. This is a cross-platform codec for QuickTime and AVI movies that is best suited for video and CD-ROM playback. Movies take a long time to compress, but they decompress quickly at playback. Cinepak is best for 16- and 24-bit animations. Cinepak has better compression ratio, image quality, and faster playback than Apple's Video codec.

- **Component Video**. This is best suited for the archival or interim storage of digitized video compression. At a compression ratio of 2 to 1, Component Video requires lots of disk space. Use this codec when image quality is more import than compression ratio or real-time playback.

- **Graphic**. This is recommended for 8-bit still images. Useful for compressing pre-dithered animations in which decompression performance is not as important as the compression ratio.

- **Photo-JPEG**. This is recommended for compression of still images with continuous tones such as photographs. It works too slowly for animation, and discards some data (it's a lossy compression method). Photo-JPEG is recommended for archiving 16- and 24-bit source clips. This codec saves the image in JPEG format, which results in high-quality images with a high amount of compression.

  NOTE! You may have other JPEG options available.

- **Video**. This is Apple's QuickTime video compressor. It is best for high-quality video clips that play in real time on faster Macintosh machines. Its only option is color. Video is appropriate for rotoscoping. This codec can play back regular animations, but the image quality is not as high as when the Animator compressor is used.

AVI Compression Options
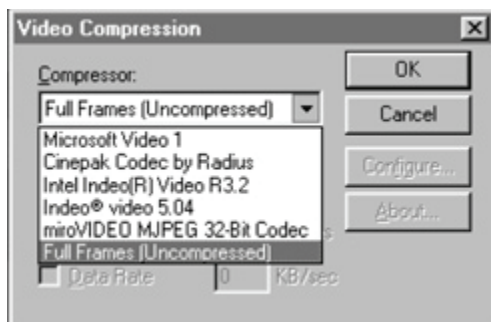
- **Microsoft Video 1** . This codec compresses each frame of a clip (and is used for analog video (8 and 16-bit). Some data is discarded (the codec is lossy).

- **Full Frames (Uncompressed).** This option is best for capturing full frames of analog video. No compression is applied, so use of this codec requires lots of disk space.

- **Cinepak Codec (Radius?).** This is a cross-platform codec for QuickTime and AVI. Movies take a long time to compress, but decompress quickly at playback. Cinepak is optimized for 16- and 24-bit animations. Cinepak has a better compression ratio, image quality, and playback than the Microsoft Video 1 codec.

- **Intel Indeo Video R3.2.** This codec uses higher compression ratios, and results in better image quality and faster playback than Microsoft Video 1.

- **Microsoft RLE** . Use this option for compressing animation and computer-synthesized 8-bit images.

- **Intel Indeo Video Raw** . Use this compression method for capturing uncompressed video. You will obtain superior image quality because no compression is applied.

TIP! For optimal compression, use the right frame size: 160 x 120 for Apple Video compressor and 240 x 180 for Cinepak. When using Cinepak, frame dimensions should be multiples of four for best performance.

To set compressor options for the QuickTime format:

1. Choose a color depth from the Color list. The depths available depend on the selected codec.

2. Use the Quality slider to set the level of compression.

   The compression ratio is inversely proportional to image quality. The Quality slider allows you to set an optimum balance between the amount of compression and image quality.

   - For a codec that uses key framing, enter a Key frame rate.

     The key frame is used in temporal compression methods. Each key frame is stored in its entirety. The next set of frames:up to the next key:are saved only as changes.

   - For a codec that supports the feature, you can use the Limit Data Rate option.

   - This allows you to set a maximum size for any frame. QuickTime automatically adapts the compression quality to maintain this rate. This feature is available for only a few codecs. When you are finished setting options, close the options dialog.

To select a file format for a movie:

1. Select the **Properties tray: Output Settings button** .

2. Select the File Format panel.

3. Select Movie.

4. Choose the movie file format you want from the list.
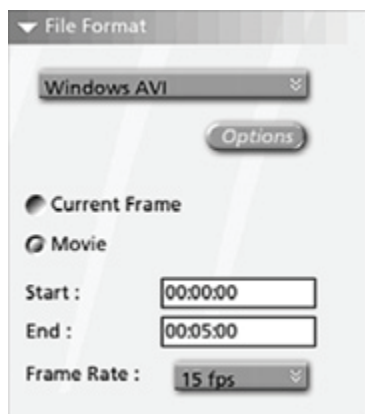
   - Movie formats include QuickTime Movie (Macintosh or Windows (if the player is installed)), AVI Movie (Windows), and a variety of sequenced image formats.

- The "Movie" formats (QuickTime, AVI, and animated GIF) generate a single file that includes all of the frames.

- The "sequenced" formats generate a single file for each frame. The files are numbered 000, 001, 002, and so forth, to keep them in sequence. Sequenced files are often used for transferring the animation to other programs, such as Adobe Premiere.

5. Set the start and end points of the render range.

6. Select the frame rate from the list (in set increments from 1 to 60 fps).

7. Click Options. Carrara displays a dialog that lets you set features for this file format.

Rendering Statistics

You may want to keep track of the rendering process for the current scene being rendered. You do this through the Statistics button in the Properties tray.

Scene Info

Scene Info gives basic information on the number of objects lights, master objects, and master shaders in your scene.

To view scene information:

1. Select the **Properties tray: Progress/Statistics button** .

2. Open the **Scene Info** panel.

3. Look at the list of items in the scene being rendered.

Rendering Progress

Rendering Progress reflects the start and estimated end times for the rendering, as well as the elapsed time, time left and disk space.

To view rendering progress:

1. Select the **Properties tray: Progress/Statistics button** .

2. Open the **Rendering Progress** panel.

3. Look at the progress of the rendering of your scene.

Animation Progress

Animation progress shows the frame number currently being animated, the time rendering takes for the average frame, and the number of frames left to be rendered.

To view scene information:

1. Select the **Properties tray: Progress/Statistics button** .

2. Open the **Animation Progress** panel.

3. Look at the progress of your animation.

Renderer Statistics

Renderer Statistics tells you the number of rays launched. Rays are used to compute the image. These statistics are renderer-specific; for example, Draft Renderer doesn't provide any statistics. Rendering statistics are used to estimate the complexity of your scene. More rays are launched when your scene contains more reflection and or transparency.

To view renderer statistics:

1. Select the **Properties tray: Progress/Statistics button** .

2. Open the **Renderer Statistics** panel.

3. Look at the number of rays launched.

Rendering Engines

Carrara offers three main rendering engines to choose from: Draft, Photorealistic and Non-Photorealistic Renderer (NPR).

Each engine has an advantage when you are looking for a particular result. You select your rendering engine from the Properties tray. Refer to "The Properties Tray."

- The **Draft renderer** is an excellent choice for fast proofing. For more details see "Draft Renderer."

- The **Photorealistic Renderer** calculates the effects of light rays from your light sources as they encounter the objects in your scene. Ray tracing shows most of the "real world" lighting effects, including transparency, shadow, reflection, and bump maps. It also supports Global Illumination and Caustics. Refer to "Photorealistic Renderer."

- The **Non-Photorealistic Renderer (NPR)** creates images that imitates drawing and painting effects. For more details see "Non-Photorealistic Renderer."

To speed up rendering you can also render across networks (Pro version only). See "Network Rendering".

To set a rendering engine and rendering properties:

1. Select a scene from the Render tray.

2. Display the Properties tray.

3. Select Renderer Settings**.**

4. Choose a Rendering Engine from the Renderer list.

5. Select or deselect the rendering preferences for this object.

Draft Renderer

Follow the instructions below for setting Draft Renderer options. The Draft Renderer provides fast rendering and should be used for animation previewing.

To set the Draft renderer options:

1. Select **Properties tray: Renderer Settings button: Renderer panel**.

2. Select **Draft** as the renderer.

3. Select a Rendering mode:

   - **WireFrame** : This is the default rendering mode. WireFrame is particularly useful for scene composition since it allows you to align objects precisely.

   - **Gouraud** : This method uses the colors at the corners of a polygon to determine the shading for the entire polygon. It then blends the color across the polygon from the corners.

   - **Textured: (Phong)** This method shows all of the details of the shape, color, and texture of your objects, but takes longer to calculate and draw. (Standard, Pro)

   - **Sketch**: This method shows a flat shaded image with pencil-sketched hard edges.

4. Select a lighting mode:

   - **Fast Lighting** : This mode uses a pre-existing lighting set and pre-calculated textures. It renders quickly but doesn't depict how the final scene will be rendered.

   - **Actual Lighting** : This mode uses the lighting information contained in the scene. This option matches the lighting you will see in the final rendered scene.

Photorealistic Renderer

The Photorealistic Renderer should be used for realistic rendering. It is based on the Raytracing technology and supports global illumination and caustic rendering.

To select the Photorealistic Renderer:

- Choose **Properties tray: Renderer Settings tab** then choose **Photorealistic** from the list.

The Photorealistic Renderer settings are divided in three different categories: Ray Tracing, Global Illumination and Caustics settings:

Ray Tracing Settings

You can set the following options for raytracing:
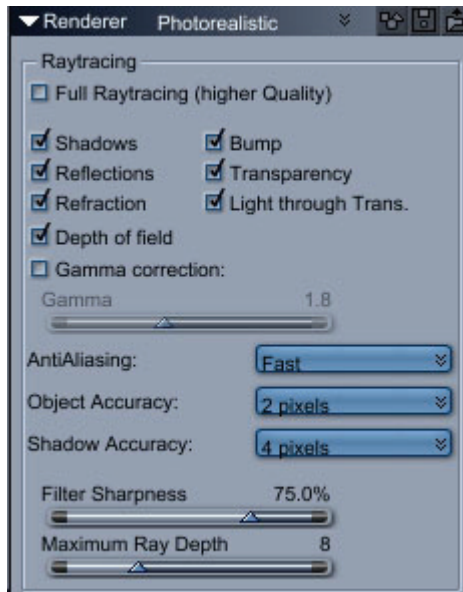
- **Full Raytracing**: Checking the option will ensure the best rendering quality but will usually slow down the rendering. (This corresponds to the difference between the old Ray Tracer and the Hybrid Ray Tracer of the previous versions).

- **Enable Shadows** to render shadows.

- **Enable Reflections** to render reflections on reflective surfaces.

- **Enable Refractions** to render refraction effects through transparent objects.

- Enable **Depth of Field** to render scenes that use depth of field settings.

- **Enable Bump** to render bump effects on objects.

- **Enable Transparency** to render transparent objects.

NOTE! Reflectiveness, transparency, refraction, and bump are Shader characteristics. Refer to "Shader Channels Reference."

- **Enable Light Through Transparency** to render lighting effects through transparent objects.

- Select **Gamma Correction** to turn on the gamma correction.

- Adjust the **Gamma** slider to set the value of the gamma. A value between 2.2 and 2.6 is usually correct for a standard computer monitor. Check your monitor's documentation for the actual gamma value. Note that some video cards actually perform gamma correction so you might not need to do anything. The Macintosh has a gamma correction of 1.4 so you might want to use a gamma of 1.8 to get the correct result on your monitor.

- Select None, Fast, Good, or Best for **Antialiasing** . (These options allow you to smooth object or shadow edges.)

- Select a **Object Accuracy** setting if the accuracy of some details are missing. This setting indicates the size of the smallest object or shadow in your scene.

- **Shadow accuracy**: Shadows usually require a lower accuracy than objects. A low accuracy (2-4 pixels will speed up the rendering) while a high accuracy (0.5 pixel) will assure that no detail is missed. The renderer automatically tries to pick up small details.

- Adjust the **Filter Sharpness** to control the sharpness of your image. Very sharp images will tend to show aliasing problems (stair case effect) whereas a low value will tend to create blurry images.

- Adjust the **Maximum Ray Depth** slider to set a limit on the number of interactions allowed for rays that reflect or refract.

Gamma Correction

The luminance of a typical monitor does not vary in a linear fashion. It follows a function that is characterized by a parameter called gamma. For most monitors the gamma value varies between 2.2 and 2.6.

Carrara allows you to "gamma correct" your image so that it matches the luminance of your monitor. This allows you to view much more realistic images. With Global Illumination the gamma correction performs more than to correct the image. It also assures that the textures are correctly taken into account in the lighting calculations.

Note that an image that is gamma corrected will tend to be much brighter than one that is not.

Global Illumination

Global Illumination is the simulation of advanced lighting effects such as indirect diffuse lighting (radiosity effects), extended light sources and sky illumination.

Sky Light

You can use the light of the sky to illuminate your scene. This is especially useful for outdoor scenes to simulate the effect of a cloudy sky for instance. The sky light is controlled by the Background and the Atmosphere applied to the scene. The image below shows a logo rendered with only the diffuse lighting of the sky.

To light a scene with the sky light you must:

1. Set up a Background or an Atmosphere in the scene Properties.

2. Check the Sky Light checkbox in the Global Illumination panel.

3. Select the Intensity of the Sky Light using the intensity slider.

4. Adjust these settings to control the quality of the rendering of the sky light: Lighting Quality and Interpolation.

Indirect Lighting

In real life, much of the lighting you see is caused by indirect illumination. For example, the sun coming through a window only directly lights at a small part of a room. If there were no indirect illumination, the rest of the room would remain black. That's what you see in images created with typical raytracing. However, in real life, the light bounces off of the ground, walls and ceiling, flooding the room with light.

The Indirect Lighting option of Carrara allows you to simulate the diffuse part of indirect lighting. The diffuse part corresponds to that part of the light that is reflected uniformly in all directions by each surface as opposed to the specular part of the lighting that is reflected by shiny surfaces (mirror effects) or by transparent objects. This is the main component of indirect lighting in most cases and is very useful in properly simulating indoor scenes.
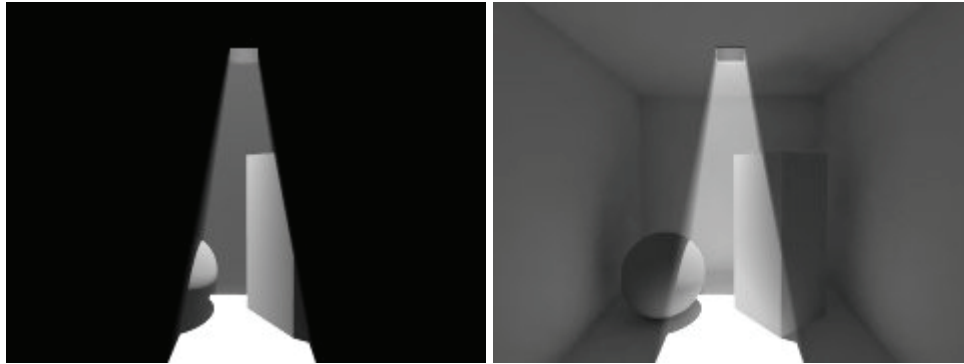
Notice how the light bounces off the ground to light up the entire room when the Indirect Lighting calculation is enabled.

Ambient Occlusion

Carrara provides a faster method to approximate the result of global illumination: ambient occlusion.

Ambient occlusion calculates the amount of light that is visible from a point. It uses a hemisphere centered on the point and calculates the percentage of light that would be visible from that point if the hemisphere was emitting light. This gives a much more realistic lighting for your scene even though it is only a rough approximation.

The radius of this hemisphere is very important. If you are rendering a room and you pick a radius bigger than the room itself everything will be black because the hemisphere will contain the walls. If you pick a radius that is too small, ambient occlusion will have only a small effect in corners.

To use Ambient Occlusion:

1. Select Indirect Lighting in the rendering properties.

2. Select Ambient Occlusion as the calculation method.

3. Select the radius used for the Ambient Occlusion calculation.

4. Make sure the ambient light of the scene is set to a proper value (the ambient light settings are accessible in the scene properties).

5. Render your scene.

NOTE! Contrary to the Full Indirect Lighting where you should probably set the ambient light to 0, you have to set the ambient light to a high value for ambient occlusion otherwise it will have no effect.

Area Light Sources

In typical raytracing programs, most light sources are punctual light sources. This means that they come from a single point. Although direct sunlight can be approximated by such a mechanism, this is clearly not the case for a tube like a neon.

The renderer uses the glow channel of the shader to simulate extended light sources. Without Global Illumination, the glow will only make the object brighter, but if you enable the Indirect Lighting option, a glowing object will act as a light source. This allows you to create light sources of any shape and size.

TIP! You can use the Scale shader (instead of the Value shader) to make very bright objects. If you want to make an object emit a very bright light of a given color (for instance, red), you can use a multiply operator between a Color and a Scale shader.

Irradiance Maps

Irradiance maps store the full solution of the global illumination. You can use them to speed up your animation renderings.

If you are rendering a static scene (where nothing is moving except the camera), you do not need to recalculate the global illumination for each frame. Just select the option called "one for all frames" and the rendering engine will assume that the scene is static and render animations much faster.

Load/Save Irradiance Maps

When working on a static scene, you can also save an irradiance map and then reuse the map over and over to render it from different point of view.

You can also use this feature for some more interesting effects. For instance, you could render the irradiance map for a scene without texture so that the Global Illumination solution is faster and less noisy and then use that irradiance map to render your scene with the texture applied.

Settings
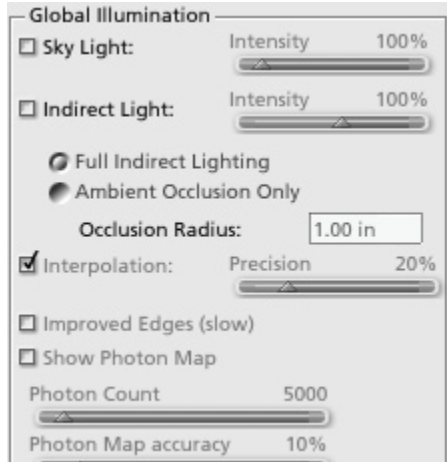
You can enable or disable the following options to add Global Illumination to your scenes:

- **Sky Light** activates the illumination by the sky dome. Note that it can be used with or without the Indirect Light option to speed up the rendering.

    o **Intensity** controls the intensity of the sky light.

- **Indirect Light** activates the calculation for indirect lighting. Note that the renderer will also take into account the light coming from glowing primitives.

    o **Intensity** controls the intensity of the indirect lighting, and corresponds 100% to a physically accurate effect.

- **Interpolation** speeds up rendering through the use of a caching mechanism to avoid recomputing the light for every pixel.

    o **Precision** controls the precision of the interpolation. A value of 0% can be used to get a quick preview of the lighting.

- **Improved Edges** triggers more accurate calculations of edges and corners. It removes most of the shadow and highlights that can appear in corners.

- **Show Photon Map** displays a quick approximation of the indirect lighting. This is useful to get quick feedback on the lighting within a scene. This approximation is also used when the final calculation is done. Use this setting to select the proper photon map precision.

    o **Photon Count** controls the number of photons emitted to sample the indirect lighting. When rendering, the memory used is proportional to this value. Use higher values only when necessary since this can significantly slow rendering performance. You need high precision when your scene contains many small illumination details or thin walls.

- **Lighting Quality** controls the quality of the final calculation. The rendering time increases quickly when you increase the quality. Since rendering can be very slow in order to obtain a high-quality result, increase the quality value carefully until the desired quality is obtained.

- **Accuracy** controls the accuracy of the rendering. Setting it to a small value ensures that you do not miss small details of illumination. Since the indirect lighting usually varies slowly, it is recommended to use a low accuracy for indirect lighting (8 - 16 pixels). Rendering time increases quickly as accuracy goes up. Even if you set a low accuracy, the renderer will, in most cases, still pick up small details. Use a high accuracy setting only when the details of the indirect lighting are incorrect.

Troubleshooting Global Illumination

You can correct the following types of problems.

- **Light filtering through corners** : If you use a low resolution photon map and very thin walls, the light tends to go through the walls at corners. To avoid this problem, make sure that your objects have a thickness. If this problem still occurs with a thin wall, you can either increase the photon count or increase the size of the wall.

- **White Objects** : When using reflection and transparency you might get very white objects. Usually the shading properties of your objects are not realistic (that is, your objects are creating lighting energy). To avoid this problem, make sure that the sum of the reflection, transparency, and color channels remains below 100%.

Speeding up Global Illumination

Here are some tips to improve rendering speed when using Global Illumination:

- **Use Ambient Occlusion**.

- **Use low accuracy settings**: Indirect lighting tends to vary very slowly so you do not need a lot of accuracy to get the correct image. An accuracy of 8 to 16 pixels is usually enough to get good results. Use high accuracy only if there are some small details of indirect lighting that are missing.

- **Use spot lights**: When you are lighting your scene, you can help the renderer by using a spot light that only lights the most important area of the scene. All types of lights work, but they might give you longer rendering times and lowered image quality.

- **Avoid huge scenes**: If you are creating an image of the interior of a house, limit the exterior world. Each object you add to a scene increases the rendering time.

- If your scene is static, you can avoid recalculating the lighting at every frame by using the correct irradiance map options.

Caustics (Standard, Pro)

Overview

When light is transmitted through transparent objects such as a glass or a bottle, or when the light is reflected by a mirror, it tends to create very specific illumination patterns called **caustics** . One typical example of caustics is the light effects you can see in a swimming pool.

Settings
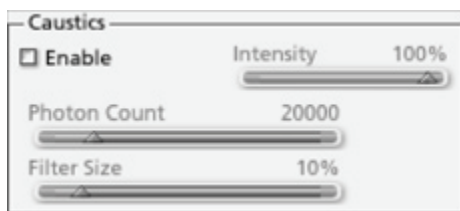
The caustic settings

- **Enable** activates calculation of the caustics.

- **(Caustics) Intensity controls the intensity of the caustics.** (100% corresponds to a physically accurate effect.)

- **Photon Count** controls the number of photons that are emitted to sample the caustics. Higher values mean more precise caustics but also longer rendering times.

- **Filter size** : This slider allows you to control how blurry the edges of the caustics will be. Higher values tend to make the edges very smooth and blurry.

NOTE! Global Illumination and Caustics are only available with the Photorealistic Renderer.

The images below show the same scene rendered without caustics (left) and with caustics (right). Notice how the light concentrates through the sphere.
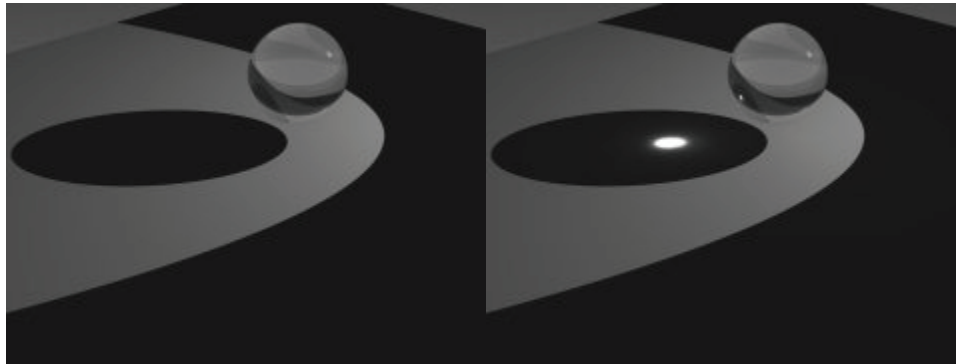
Misc Settings (Standard, Pro)

- **Enable Multi-Threading**: This option allows Carrara to use an extra thread for rendering. If you have a multi-processor computer, a thread will be created for each processor in order to take advantage of all of them for rendering.

- Select the **Enable Field Rendering** option if you are rendering an animation for video and you want to render the interlaced fields separately.

NOTE! If you use field rendering, choose 60 fps (NTSC) for the rendering frame rate.

Non-Photorealistic Renderer

Overview

For many years the goal of computer graphics has been to create synthetic images as realistic as possible. This search for realism has led to images that are more and more realistic. However, very often what an artist really wants is to create a representation of reality. In this context realism is not the goal anymore and the artist is therefore given free expression which allows him to express himself in many different ways. It is this freedom of expression that the Non-Photorealistic Renderer (NPR) of Carrara is meant to give back to the artist.

The NPR renderer allows you to create images that imitate various painting and drawing styles. Yet it is not meant as much to imitate as to give you the freedom to create your own styles.

Basic Concepts

There are a few basics concepts that need to be understood before you can start using the NPR renderer:

- **Paper:** The paper is the background of your image. Everything that is going to be rendered will be drawn on top of that.

- **Layers:** Your image will be composed of different layers: the background layer, the diffuse layer, the highlight layer and the outline layer. These layers are drawn on top of each other (the background layer is drawn first on top of the paper and the outline layer is drawn last)

- **Brush Strokes**: The most important concept of the NPR renderer is the concept of brush strokes. Everything it renders is composed of a number of brush strokes. You can control the brushes at two different levels:

- **At the renderer level**: by changing the parameters of the renderer.

- **At the geometry level**: The brush size and shape depends on the triangle shapes so by editing your mesh you can influence your rendering. (Smaller brushes are used to paint smaller triangles).

NPR Parameters

Global Parameters
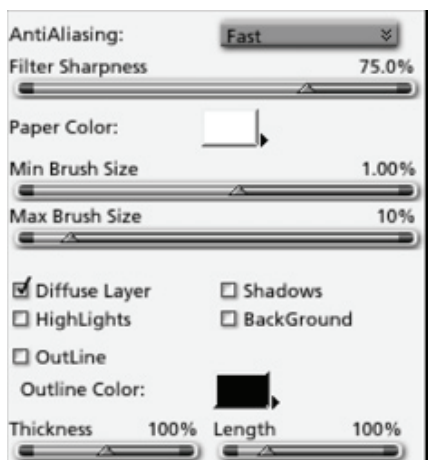
Global Settings for NPR Rendering:

- **AntiAliasing:** This parameter controls the amount of antialiasing that is performed on the image. Increasing the antialiasing will create smoother images by smoothing edges and removing noise. Be careful as a high level of antialiasing will slow down the rendering.

- **Filter Sharpness**: This parameter controls how sharp your final image will be. It complements the antialiasing parameter. A low value will create a blurry picture and a high value will tend to show aliasing.

- **Paper Color:** Controls the color of the paper.

- **Min Brush Size:** Controls the minimum size of a brush stroke as a percentage of the image size. Increasing this value will create a less accurate painting effect.

- **Max Brush Size:** Controls the maximum size of a brush stroke as a percentage of the image size.

- **Diffuse Layer:** Activates the diffuse layer. This will paint everywhere on the objects except in the shadow and highlight areas if they are activated.

- **Highlights:** Activates the highlight layer.

- **Shadows:** Activates the shadow layer. This will turn on shadow calculation.

- **BackGround:** Activates the background layer. It is important to understand how the background layer differs from the paper. The background layer is actually painted on top of the paper. If the brush strokes are sparse, you will be able to see the paper through it. The background layer will use the backdrop of the scene.

- **Outline:** Activates the outline layer which will draw the outline of objects.

The image below shows a render using the Outline and Shadow layer.

Layer Parameters

The NPR also contains layer tabs (Diffuse, Highlights, Shadows, and Background, for example). You can edit each layer separately. Each of the following parameters apply to one single layer at a time:

- **Object Clipping:** If checked, all brushes of the current layer will be clipped by the edges of the object. This allows you to create very clean outlines for painted objects.

- **Brush:** By click on the brush image, a dialog opens that offers a selection of brushes that can be used to paint in the current layer.

- **Brush Color** controls the color of the brush that is used for this layer. (For example you can select a dark grey color for shadows). Note that it will have no effect if "Use Object Color" is selected.

- **Use Object Color**: If checked, the color of the brush will be the color of the object at the brush location (including lighting), otherwise the color selected in the Brush Color field will be used.

- **Thickness/Length**: These sliders control the thickness/length of the brush strokes.

- **Intensity**: Controls the intensity of the random variations of the brush color. A value of zero will lead to a uniform color whereas a value of 100% will create a very random intensity distribution. Note that high values of this parameter might cause flashing effects in animations.

- **Transparency**: controls the transparency of the brush strokes relative to each other. This is a very important parameter since it really controls the final appearance of the rendering.

- **Orientation**: This panels controls the orientation of the brush strokes on each triangle. Selecting several orientations will combine them (If horizontal and vertical are selected, for example, you will get a diagonal orientation). Here is a description of the available orientations:

- **Horizontal**: TBD

- **Vertical**: TBD

- **U**: the brush strokes follow the U texture coordinates on the surface of the object.

- **V**: the brush strokes follow the V texture coordinates on the surface of the object.

- **Triangle**: The brush strokes are orientated according to the longest side of the triangle.

- **Normal**: The brush strokes are orientated according to the normals of the object.

- **Hue**: The brush strokes are orientated according to the angle which corresponds to the hue of the color. (very useful for backgrounds).

- **Random** controls the amount of randomization used for the brush.

Index

Glossary

2

2D (two-dimensional): An image, shape, path, or plane that exists in the dimensions of width and height only.

3

3D (three-dimensional): An object or volume that exists in the dimensions of width, height, and depth.

A

Absolute position: The X, Y, Z coordinates of an object's hot point in the Global Universe.

Alignment (relative): Setting two or more objects to some meaningful spatial relationship, such as centering them or distributing them evenly along a line.

Alignment (text): Text alignment affects text objects of more than one line. The lines of text may be aligned to their left edge, their center, or to their right edge.

Alpha Channel: The top byte of a 32-bit pixel that is used for data other than color. The channel may hold mask or transparency data.

Ambient Light: That light responsible for the overall, diffuse lighting of a 3D scene, similar to daylight in the real world.

Anti-aliasing: Intermediate colors (or shades of gray) in the pixels between contrasting colored regions. Anti-aliasing improves the appearance of objects in renderings by removing jagged, stair-step edges.

Aspect Ratio: The ratio of the width of an image to its height (x:y). For example, the aspect ratio of an image 640 x 480 pixels is 4:3.

Atmospheric Effects: Atmospheric conditions or phenomena that affect the clarity or mood of a scene. Fog and smoke are examples of atmospheric effects.

Attitude: See Orientation.

Axis: A hypothetical linear path. The X, Y, and Z axes (width, height, and depth, respectively) define the directions of the 3D universe. The axis along which an object is rotated is the axis of rotation. An object's axes are parallel to its bounding box.

B

Backdrop: A picture that is automatically composited behind a 3D scene. The matte paintings used in traditional movie making are an example of backdrops.

Background: 2D images that show up on reflective surfaces (metal, glass, etc.). Reflected backgrounds increase the realism of such surfaces. Reflected backgrounds are also knows as environment maps.

Bézier Curve: A path defined by the position of four control points (at the ends of the tangents of the vertices). The length and angle of the tangents describe the deviation from a linear path that the curve follows between vertices.

Bit Depth: The number of bits used to define the shade or color of each pixel in an image. A 1-bit image is black and white. An 8-bit grayscale image provides 256 shades of gray (2 to the 8th power is

256). An 8-bit color image provides 256 colors. A 24-bit image provides over 16 million colors: 8 bits are used for red, 8 are for blue, and 8 for green.

Bitmap: A pixel-based image.

Boolean: An object created by combining two objects using mathematical operators (Booleans). The two objects may be subtracted one from another, merged, or intersected to form the Boolean object.

Bounding Box: A hypothetical box drawn around an object or group of objects. A bounding box is the smallest rectangular box in which the object (or group of objects) fits completely. The bounding box is parallel to the axes of the object. The bounding box is shown (around the preview of selected objects and groups) in the 3D view, and it is the bounding box—not the object itself— that casts projections onto the working box grid.

C

Child: An object linked to another object (its parent) in the hierarchy. When the parent is moved, the child and all grandchildren go with it. The parent-child link is used to enable articulation of complex objects.

CMYK Color: The subtractive color model, used in printing. Colors are created by assembling different densities of cyan, magenta, yellow, and black pigments on a surface. When white light strikes the surface, only specific bandwidths are reflected— depending on the density of the specific pigments. The reflected bandwidths create the perceived color. The CMYK model is called the subtractive model because the pigments subtract (by absorption) the bandwidths of white light that do not contribute to the specified color.

Colinear: Two or more objects that are in the same line.

Color Depth: See Bit Depth.

Complex Object: An object constructed of several simple objects that are linked or grouped. For example, a telephone— comprised of the grouped, simple objects: cord, handset, and cradle— would be considered a complex object.

Component (shader): An elementary building block of a shader tree.

Concentric: Having the same center.

Constrain: To restrict object movement to a particular plane, axis, or angle. The working box is the primary tool for constraining an object.

Control Points: The knobs at the ends of Bézier tangents used to adjust a Bézier curve.

Coordinate System: Groups and individual objects have their own local coordinate systems. The origin of a group or object's coordinate system is the center of its bounding box. The axes are parallel to the sides of the bounding box.

Coplanar: Occurring in the same plane.

Cross-Section: One of the planes on which 2D shapes are drawn in the Spline modeler to create a 3D object. Objects have two or more cross-sections.

D

Deformers: See Modifiers.

Dithering: The process of approximating pixel colors when reducing the color depth of an image or mapping to a different palette. Dithering can improve transitions between colors when reducing a 24-bit image to 8-bit format.

Dolly: The computer equivalents of the real world camera movement commands. With Dolly, the camera moves around in 3D space as if gliding on the surface of a sphere that has the object of interest as its center.

DXF: A standard 3D file format originally developed by Autodesk, makers of AutoCAD, for the purpose of exchanging CAD data between various 3D software applications. Widely used as a poor man's exchange file format, DXF only offers support for basic geometric information (no textures).

E

Extrusion: The method of creating a 3D object by moving a 2D cross-section along a path (the sweep path). The Spline modeler uses this method.

F

Face: The back (first cross-section shape) or front (last shape) of an extruded object.

Formula: A mathematical expression that, with input, returns a meaningful set of values. Formulas may be used to create objects, define a tweener curve, or create a color pattern in a shader, gel, background, or backdrop.

G

G-Buffer (Geometry Buffer): A type of information carried on optional channels of a rendered image. The G-Buffer carries an aspect of three dimensionality with the 2D rendering. This can assist 2D filters and paint tools during retouching.

Global Mixer: A shader component that mixes all channels of its component shaders.

Global Universe: A Cartesian coordinate system with the origin of the X, Y, and Z axes (0, 0, 0) at the center of the universe. The Global Universe is fixed and is not affected by changes made to the working box.

Glow Channel: A shading channel that holds data describing an object's luminescence or glow.

Gouraud: An object preview mode in the Assemble room which shows objects with shading at a low level of detail.

Grayscale: An image in which the pixels are defined with 8 bits, which provides 256 levels of gray.

Group: A set of collected objects. Grouping enables a set of objects to behave as one.

H

Hierarchy: The tree structure in the Sequencer tray that lists the objects in the scene and shows their logical relationships (links and groups). The Hierarchy also includes listings for the animatable properties of each object.

Highlights: Regions on an object where light reflects directly into the viewpoint. Highlights appear as bright spots on smooth! shiny objects and are almost nonexistent on rough!dull objects.

Hot Point: A special, active, movable point located by default at the center of an object or group. The hot point is used as the reference for rotation and for some positioning and alignment operations.

I

Icon: A pictorial representation of a tool, object, file, or other program item. An item is selected by clicking once, or sometimes double-clicking, on its icon.

J

Jaggies: The appearance of stair-step, jagged edges in a pixmap image. Jaggies can be reduced with anti-aliasing.

K

Key Frame: A moment in the animation where an object's characteristics change. Key events are marked on the time track for each animatable property.

L

Leading: A text property that determines the vertical space between lines of text. The default leading value is 120% of the font's point size. Decreasing the percentage makes vertical spacing more compact, while increasing the percentage expands it.

Letter Spacing: A text property that determines the horizontal spacing between characters in an entire word, line, or text block. A lower value decreases spacing, while a higher value increases it.

Links: (Also known as Constraints.) A variety of child/parent relationships and constraints on relative motion to simulate traditional, real world mechanical links. Shafts, axis, and ball joints are examples of constraints.

M

Mask: The mask is a grayscale image, stored in channel 4 of a rendering, which describes where objects are versus where they are not. In a 2D image editing program, the mask is used to generate a selection, which helps you composite your rendered scene with other background imagery.

Master Objects: The mold from which object instances are created. If a master object is modified, all of the objects derived from it are modified similarly.

Master Shader: The shader assigned to an object. Each object can have only one Master Shader. However, that Master Shader may be composed of multiple sub-shaders.

Metaball modeler: A modeler that uses blob primitives to create 3D objects.

Modifiers: Properties that can be applied to any object (or group of objects) to modify its geometry. Asymmetric scaling, bend, twist, and shatter operations are examples of modifiers available in Carrara.

Motion Path: A curve, visible in the scene window, that shows where an object will move during the course of an animation.

N

Non-Photorealistic Renderer: (NPR) creates images that imitates drawing and painting effects.

Normal Vector: A ray perpendicular to an object's surface at a given point.

O

Object: Any 3D volume or other item that appears in the universe, including cameras and lights. When objects are grouped, their group is also described as an object.

Operator: A shader component that combines other shaders to form a sub-shader.

Orientation: The direction an object faces as defined by the compound effect of the object's rotation around the X, Y and Z axis. Usually, the most important aspect of an object's orientation is its relation to other objects. An object's relation to the viewpoint can be changed by moving the current camera.

Oversampling: Rendering at a higher resolution, then bringing the resolution back down to minimize artifacts like aliasing.

P

Pan: The computer equivalents of the real world camera movement commands. With Pan, the camera acts as the center and rotates at a fixed position in space to track an object or view a scene, much like a movie camera can rotate on a tripod to follow a moving object.

Penumbra: The region of partial shadow at the edge of the full shadow. In Carrara, you can achieve nice penumbras with the Soft DRT Shadows option for lights.

Phong: An object preview mode in the Assemble room which shows objects with shading at a low level of detail.

Pipeline: An extrusion in which the cross-section is always perpendicular to the sweep path.

Pitch: The aspect of an object's attitude that describes its angular deviation along its vertical (top-to-bottom) axis.

Pixel (picture element): One dot in a 2D image. Computer images are created as an array of such dots, each having a specific color. See also Resolution and Bit Depth.

Plane: A hypothetical, 2D construct that may exist at any attitude in space. A plane can be envisioned as a flat sheet of invisible paper that stretches infinitely in two dimensions. In Carrara, planes are used to constrain the direction of translation of an object. The working box shows the angle at which the constraint planes are set. By default, the planes of the working box are set parallel to the axes of the current local universe.

Point: As a unit of measure, a point is 1/72 of an inch. Point also refers to Bézier vertex and control points.

Point of View (also viewpoint): The position and angle from which you view a scene. The point of view (POV) is always through a camera. You may add several cameras, positioned and angled differently, and switch the POV between them. When you render an image, you choose the POV from which the image should be taken.

Preview: The display of an object in the 3D View or Modeling window or the color chip of a shader in the Browser or Sequencer. Also refers to a rendering with a low resolution setting.

Primitive: An object created by definition, rather than modeling. A primitive object cannot be dismantled into components. The Sphere, Cube, Cylinder, Cone, and Icosahedron are the basic primitives in Carrara. The environmental primitives include Fountain, Fog, Cloud, and Fire.

Production Frame: The computer equivalent of the viewfinder in a regular camera. The production frame can be thought of as describing the area of the 3D scene that will be rendered into an image.

Projection: The silhouette of an object's bounding box on one of the three visible planes of the grid. The hot point of the selected object or group also casts a projection. Projections are used for translation and rotation.

Properties: The collection of characteristics applied to (or available for) an object. Properties for the selected object appear in the Properties tray.

R

Ray Tracing: A procedure for generating a rendering. The ray tracer sends hypothetical rays of light from the sources in the scene and calculates the visual effects, for each pixel in the rendering, as the rays encounter and reflect from the objects in the scene.

Reference: A fixed point or plane used as the starting point for some operation. The hot point, the center of the bounding box, and the corners of the bounding box are common reference points.

reflected: 2D images that show up on reflective surfaces (metal, glass, etc.). Reflected backgrounds increase the realism of such surfaces. Reflected backgrounds are also knows as environment maps.

Reflection: The phenomenon of light bouncing off objects.

Refraction: The phenomenon of light deflecting as it passes through a translucent object, like glass or fluid.

Relative position: The placement of an object in relation to another object. For example, a book might be on top of a table.

Render: The process of capturing a 2D image from a 3D scene.

Resolution: For an image, the number of pixels in each dimension. For devices, like a printer or the monitor screen, resolution is given in dots per inch (dpi) or dots per centimeter.

RGB Color: The additive color model, used in computer monitors. Colors are created by adding varying degrees of red, green, and blue light. For information on the varying degrees, refer to Bit Depth.

Root: The highest level of the Sequencer Hierarchy. When you look at a group, the group box is the highest level of the Hierarchy, and therefore, the local root.

S

Scaling (object): The percentage an object is resized. Each object instance may be scaled from the dimension of its master, which has a scaling of 1.

Scaling (text): Scaling changes the width of characters without affecting their height. A value below 100% results in characters that are narrower than usual, while a value above 100% results in characters wider than usual.

Shape: A 2D path that may be open (a line) or closed, such as an oval or a polygon. Shapes are used in cross-sections when modeling. In shading, paint shapes are used to specify regions for shading.

Skinning: The process of applying a vertex object to a skeleton for the purpose of animating the vertex object.

Spline modeler: A modeler that uses cross-sections, sweep paths, and scaling envelopes to create 3D objects.

Sub-shader: Any shader used as a component within a shader tree.

Sweep Path: The curve or line along which shapes are extruded when modeling. Called the sweep or extrusion path, the path is defined by one line on the bottom plane and one line on the back plane in the modeling window. The compound curvature of these two lines defines the path itself.

T

Texture Map: A 2D image used as a shader.

TIFF (Tagged Image File Format): An image file format often used for transfer between applications or platforms. Carrara opens TIFF images in RGB format, but not in CMYK format.

Tiling: The technique of repeating a small image across a larger surface to cover it.

Track: The computer equivalents of the real world camera movement commands. Track moves the camera in a plane perpendicular to the direction in which the camera is pointing.

Translation: Any manipulation of the position or attitude of an object. Also, an extrusion in which the cross-section remains at one angle, regardless of the curvature of the sweep path.

Translucence: The characteristic of an object that allows light to pass through it.

Tweener: A formula used to control the transition between two key events.

U

Universe: The 3D root of the Hierarchy. The Universe's coordinate system is absolute (does not move or rotate).

URL: Uniform Resource Locator, the address of an item on the internet.

UV: A 3D coordinate mapping system. UV coordinates are used in parametric mapping to align points on the shader with points on the object.

V

Vertex: A control point on a path. Paths begin, change angle, and end at vertices.

Viewpoint: See Point of View.

W

Word Spacing: A text property that determines the horizontal spacing between words. A lower value decreases spacing, while a higher value increases it.

Working Box: The three visible grid planes in the scene window. Called the working box because you'll move it as you work to constrain operations to certain planes.

Working Box System: The working box's own coordinate system. The attitude of its axes and the position of its origin (at the center of the working box) change as you move and reorient the working box.

Y

Yaw: The aspect of an object's attitude that describes its angular deviation along its linear (front-to-back) axis.

Z

Z-Buffer: A rendering technique that uses z (depth) information in sorting object facets.