

Learning Lightscape

Copyright © 1993-1999 Autodesk, Inc.

All Rights Reserved

This publication, or parts thereof, may not be reproduced in any form, by any method, for any purpose.

AUTODESK, INC. MAKES NO WARRANTY, EITHER EXPRESSED OR IMPLIED, INCLUDING BUT NOT LIMITED TO ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE, REGARDING THESE MATERIALS AND MAKES SUCH MATERIALS AVAILABLE SOLELY ON AN "AS-IS" BASIS.

IN NO EVENT SHALL AUTODESK, INC. BE LIABLE TO ANYONE FOR SPECIAL, COLLATERAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES IN CONNECTION WITH OR ARISING OUT OF PURCHASE OR USE OF THESE MATERIALS. THE SOLE AND EXCLUSIVE LIABILITY TO AUTODESK, INC., REGARDLESS OF THE FORM OF ACTION, SHALL NOT EXCEED THE PURCHASE PRICE OF THE MATERIALS DESCRIBED HEREIN.

Autodesk, Inc. reserves the right to revise and improve its products as it sees fit. This publication describes the state of this product at the time of its publication, and may not reflect the product at all times in the future.

Autodesk Trademarks:

Discreet is a division of Autodesk, Inc. Autodesk, AutoCAD, 3D Studio MAX, 3D Studio VIZ, DXF, and 3D Studio are registered trademarks, and Lightscape, LSnet, and Discreet are trademarks of Autodesk, Inc. in the USA and/or other countries.

Third-Party Trademarks:

OpenGL, Open Inventor and Silicon Graphics are trademarks or registered trademarks of Silicon Graphics, Inc. Microsoft, Windows and Windows NT are registered trademarks of Microsoft Corporation. LightWave 3D is a registered trademark of NewTek. ArchiCAD and Graphisoft are registered trademarks of Graphisoft R&D Rt.Lumen Micro is a registered trademark of Lighting Technologies, Inc. ARRIS is a trademark of ARRIS, LLC. form•Z is a trademark of auto•des•sys, Inc. PolyTrans and Okino Computer Graphics are trademarks or registered trademarks of Okino Computer Graphics, Inc. Softimage is a registered trademark of Softimage, a wholly owned subsidiary of Avid Technology, Inc. ACIS is a registered trademark of Spatial Technology Inc. Truevision and TARGA are registered trademarks of Truevision Inc. RealVR Traveler and RealSpace are trademarks of RealSpace, Inc. ImageCELS is a registered trademark of IMAGETECTS. LEADTOOLS is a trademark of LEAD Technologies, Inc. All other brand names, product names, or trademarks belong to their respective holders.

GOVERNMENT USE

Use, duplication, or disclosure by the U.S. Government is subject to restrictions as set forth in FAR 1 2.212 (Commercial Computer Software-Restricted Rights) and DFAR 227.7202 (Rights in Technical Data and Computer Software), as applicable.

Third Party Software Program Credits

The software program contains content files licensed by 3Name3D, Santa Monica, CA; Artbeats Software, Inc., Myrtle Creek, OR; Modern Medium, Inc. Eugene, OR; ImageCELS® (Texture Files) Copyright © 1987-1999 IMAGETECTS™ in addition to the manufacturers listed in the Acknowledgements at the end of this manual. Please note that the manufacturers reserve the right to discontinue or change any lighting or other products represented in the software, and the copyrights to these content files are the property of their respective owners.

Printed in the United States.

Title: Learning Lightscape
Item No.: LIGS3.2-01
Publication ID: 1.0
Date: April, 1999

Table of Contents

Introduction	1
Learning a New Approach	1
Creating Better Models	2
Using the Batch Utilities	2
Practicing the Workflow	2
Overview of the Lessons	4
Source Material	5
Conventions	5
Getting More Help	6
Readers' Comments	6

1 The Gallery	7
Stage 1: Importing Data	9
About the Lightscape Interface	10
Stage 2: Preparing Data	13
Viewing the Model	13
Defining Materials	17
Using Material Libraries	20
Displaying Textures	21
Aiming Luminaires	22
Stage 3: Creating the Radiosity Solution	26
Initiating the Preparation Model	26
Setting Processing Parameters	26
Processing the Solution	28
Stage 4: Fine-Tuning the Solution	32
Assigning Materials	32
Changing a Light Source	35
Stage 5: Outputting the Results	37
Creating a Radiosity Image	37
Creating a Ray-Traced Image	38
Viewing Image Files with LVu	41
Summary	42

2 The Sunroom 43

Stage 1: Importing Data	45
Objects Supported by Lightscape	45
Checking Import Units and Scale Factor	46
Stage 2: Preparing Data	49
Viewing the Model	49
Organizing Data into Layers and Blocks	57
Working with Luminaires	68
Working with Materials	73
Stage 3: Creating the Radiosity Solution	81
Processing Parameters	81
Setting the Processing Parameters	82
Initiating the Preparation Model	82
Processing the Solution	83
Stage 4: Fine-Tuning the Solution	87
Using Natural Lighting	87
Setting Up a Window	88
Turning Off Lights	92
Stage 5: Outputting the Results	93
Performing Lighting Analysis	93
Rendering	94
Creating a Single High-Quality Image Using Ray Tracing	95
Summary	98

3 The Lounge 99

Stage 1: Importing Data	101
Using Advanced DXF Import Features	101
Importing Geometry	102
Stage 2: Preparing Data	113
Orienting Surfaces	113
Smoothing Surfaces	117
Assigning and Aligning Textures	119
Stage 3: Creating the Radiosity Solution	124
Global Processing Parameters	124
Determining Optimum Meshing Parameters	126
Testing Direct Lights	127
Testing for Overall Light Levels and Artifacts	128
Preparing the Model for Initialization	128
Setting the Processing Parameters	129
Processing the Solution	130
Stage 4: Fine-Tuning the Solution	132

Improving the Radiosity Solution	132
Stage 5: Outputting the Results	134
Lightscape Animation	134
Creating an Animation	136
Creating Animation Images	142
Summary	144

4 The Exterior Model 145

Stage 1: Importing Data	147
Using Good Modeling Practices	147
Importing AutoCAD Files	149
Stage 2: Preparing Data	153
Assigning the Correct Textures to Surfaces	153
Assigning Procedural Textures	160
Loading and Placing Light Fixtures From a Library	164
Setting the Local Processing Parameters	167
Setting the Global Processing Parameters	172
Stage 3: Creating the Radiosity Solution	174
Creating a Night Solution for the Courtyard	174
Creating a Daytime Solution for the Full Model	178
Stage 4: Fine-Tuning the Solution	182
Creating Sun Studies	182
Stage 5: Outputting the Results	187
Ray Tracing a Region	187
Playing an Existing Animation	190
Summary	192

5 Working with Solution Files 193

Merging Files	194
Using the Mesh to Texture Tool	198
Exporting Solution Files	212
Exporting to Panoramic Format	212
Exporting to VRML Format	214
Summary	215

6	More Techniques with Blocks and Luminaires	217
	Working with Blocks and Layers	218
	Replacing Existing Blocks	218
	Using Layers to Organize Blocks	221
	Moving Block Surfaces between Layers	226
	Working with Luminaires	229
	Creating a Luminaire from Geometry	229
	Summary	242
7	Using the Batch Processing Utilities	243
	Using the Command Line	244
	Using Batch Files	246
	Location of Batch Utilities and Project Files	246
	Using LSnet	251
	Summary	258
	Going Further with Sample Files	258
	Index	259

intro

Introduction

Lightscape™ is a next generation visualization and lighting tool that produces realistic computer-generated environments and images through precise simulation of light and material behavior. It is the first application to integrate radiosity and ray tracing with physically based lighting.

Designers use Lightscape to prototype design projects under specific material and lighting conditions. Lightscape is used by producers of film, broadcast, special effects, interactive games, and web-based interactive worlds for accurate lighting of digital sets and environments.

This guide introduces the basic Lightscape concepts and teaches you the skills needed to use the software effectively. The lessons should take ten to fifteen hours of active work to complete.

Learning a New Approach

Radiosity is a fundamentally different approach to visualization than scan-line rendering and ray tracing. The most common mistake new users make, especially those well versed in 3D graphics, is thinking that Lightscape is just another renderer. Lightscape is unique because it subscribes more closely to physical laws of lighting than other renderers and, therefore, requires a somewhat different approach to modeling, material definition, and lighting.

On one hand, you can place lights as they would actually occur in the real world and get great results every time. On the other hand, you are not able to get away with certain shortcuts you may have taken in the past.

Creating double-sided surfaces is a good example for illustrating problems with the old approach. In the real world (and in Lightscape), there is no such thing as an infinitely thin, double-sided surface. In standard renderers, it is simple enough to point lights at both sides of such a surface and illuminate it. In Lightscape, however, because all surfaces reflect light, double-sided surfaces confuse the radiosity algorithm. It cannot tell which side is in front of or behind the other because they are coincident. When the radiosity algorithm tries to reflect the light off the surfaces, the surface in front may be lit by light reflected off the surface behind it!

By doing the lessons in this guide, you can learn the new approach to visualization in just a few days.

Creating Better Models

New users often expect miracles when they use their old models in Lightscape. For the best results, models should be built with Lightscape in mind. Controlling polygon count and how surfaces are formed and intersect is important for achieving efficient processing and artifact-free results. Lesson 4 includes a section on good modeling practices.

Using the Batch Utilities

Many Lightscape users are not aware of the batch utilities included with the software for doing radiosity processing and rendering offline. Ideally, you should only use the Lightscape application to set up and test the scene. Once your lights and materials are defined and you are pleased with the test results, you can save a lot of time by using the batch utilities. The lsrad, lsray, and lsrender utilities are faster than the Lightscape application because they eliminate the overhead of running Lightscape in a graphics window. Lesson 7 introduces you to the batch utilities and LSnet (a program that provides a graphical interface to the batch utilities).

Practicing the Workflow

The workflow is similar for most Lightscape projects. Although projects vary in complexity, you generally perform tasks in five stages:

- Stage 1: Importing Data
- Stage 2: Preparing Data
- Stage 3: Creating the Radiosity Solution
- Stage 4: Fine-Tuning the Solution
- Stage 5: Outputting the Results

Each of the first four lessons leads you through this workflow. The last three lessons do not follow the workflow. Instead, they present techniques related to specific stages of the workflow. For example, Lesson 5 includes tasks related to Stage 5, and Lesson 6 includes tasks for Stage 2.

Importing Data

Lightscape imports 3D models which are then rendered using the software. These models are generally created in specialized CAD or 3D modeling systems. Lightscape provides import filters for 3DS, DXF, DWG, and Lightwave formats. As additional filters become available, these filters will plug seamlessly into the existing Lightscape interface.

You can also export 3D Studio MAX to Lightscape Preparation files using the corresponding plug-in. You can open these Preparation files directly in Lightscape.

Preparing Data

To prepare geometric data for radiosity processing:

- Make sure the surfaces of the model are properly oriented.
- Define the materials that comprise the surfaces.

Because Lightscape is based upon the physics of lighting, accurately representing the material properties of the surfaces in your model is important for achieving meaningful results.

- Set special processing parameters for the surfaces in the model.

These special properties can affect the final appearance of the model and the speed and accuracy of the radiosity process. You should be selective when choosing which surfaces need extra meshing because they require additional processing.

- Define the photometric properties (lighting distribution, intensity, and color) of the luminaires (lighting fixtures) used in the model.
- Add, delete, and move entities and luminaires.

Note: The model structure used during this stage is called the Lightscape Preparation model and is saved as a Lightscape Preparation (.lp) file.

Creating the Radiosity Solution

Once you have defined the material and lighting characteristics of the model, you can start calculating the direct and indirect diffuse lighting of the model using the radiosity process. (For a good introduction to the concepts of radiosity and lighting, refer to Chapter 1, “Introduction” in the *Lightscape User’s Guide*.)

During this stage, you:

- Initiate the geometry to optimize the model for the radiosity processing.
- Define the processing parameters to control the speed and quality of the resulting radiosity solution.

Finer settings produce more accurate solutions and better quality images, but require more time and memory. Lightscape provides a wizard that lets you set the processing parameters quickly and easily.

- Process the radiosity solution.

Note: The model structure used during this stage is called the Lightscape Solution model and is saved as a Lightscape Solution (.ls) file.

Fine-Tuning the Solution

In Lightscape, you can interrupt the radiosity process at any time to alter or fine-tune the model’s appearance. You cannot change the physical shape of the model, but you can change the characteristics of a material and the photometric properties of a luminaire.

Outputting the Results

You can output results from a radiosity solution in many ways:

- Create single images. You can add specular (shiny) reflections and enhanced lighting effects using the post-process ray tracer.
- Create frames for use in animation. You can create camera paths in Lightscape. You can render the frames quickly with OpenGL using only the radiosity solution or using ray tracing to add specular lighting effects.
- Export the radiosity solution as a VRML model for use in other applications.
- Export the radiosity solution as a panoramic image with or without ray tracing.

Overview of the Lessons

The lessons include both basic and intermediate exercises. Some tasks assume you understand tasks presented in previous lessons, so you should complete the lessons in sequence:

- Lesson 1, “The Gallery,” introduces the Lightscape interface and workflow. You open and view a model, define materials, aim lights, process the solution, and render the radiosity solution.
- Lesson 2, “The Sunroom,” teaches how to import a DXF file from a CAD or modeling application. The tasks in this lesson include creating views, using layers, adding luminaires, defining realistic materials, and using daylight.
- Lesson 3, “The Lounge,” teaches advanced DXF import techniques and specialized preparation tasks such as orienting surfaces and placing and aligning textures. You also create an animation and produce frames of the first segment of the animation.
- Lesson 4, “The Exterior Model,” teaches how to create a good model for Lightscape and how to import an AutoCAD (DWG) model. Tasks in this lesson include assigning textures, creating procedural textures, and replacing surfaces with luminaires. You also set local processing parameters and set daylight options specifically for the exterior model.
- Lesson 5, “Working with Solution Files,” teaches how to join the Solution files from the first four lessons, convert surfaces into textures, and export panoramic and VRML files.
- Lesson 6, “More Techniques with Blocks and Luminaires,” teaches how to organize surfaces and blocks into layers, and how to load and create luminaires. Tasks include creating an alternate furniture layout and creating a luminaire from an imported geometry.
- Lesson 7, “Using the Batch Processing Utilities,” teaches how to save time by radiosity processing, rendering, and ray tracing by running utilities instead of the Lightscape interface. Tasks in this lesson also include running LSnet, the Lightscape network rendering system that offers a handy interface to the three main batch utilities.

Source Material

The files needed to complete the lessons are automatically copied to your hard drive when you install Lightscape. There is a separate folder in the *Projects* folder for each lesson.

For example, the files for Lesson 1, “The Gallery,” are found in:

C:\Program Files\Lightscape\Projects\Lesson1

If someone else has completed the lessons, you can install the original files from the Lightscape CD. However, the lessons are designed so that the original files are left unchanged. You can also make a backup copy of the files before you start.

Note: A few of the lessons also require that the Lightscape libraries are installed. Two of the exercises in Lesson 7 require that LSnet is installed.

Conventions

When the lesson prompts you to go to a menu item, the following convention is used:

- Choose Menu | Submenu | Submenu.

For example, View | Interactive | Orbit means you should choose the View menu, then choose the Interactive submenu, and then choose the Orbit submenu.

The names of files and folders are shown in italics. Example:

- Change the path to *C:\Program Files\Lightscape\Projects\Lesson1*.

The first time a toolbar button is introduced, the menu equivalent is given. Example:

- Choose Edit | Selection | Select or click the Select button .

For all other occurrences of the instruction, only the button is given. Example:

- Click the Select button .

To enable an option, you click in the check box and a check mark appears. When there is no check mark, the option is disabled. Example:

- Disable Occluding to make the surface non-occluding.

Dialogs often have several panels of options. You click the tab to show the corresponding panel.

Getting More Help

For a comprehensive reference to all the software features, see the *Lightscape User's Guide* and Online Help. If you are not able to find the answers to your questions, contact the New Media Call Center from 8:00 a.m. to 8:00 p.m. EST Monday to Friday at one of the following numbers. You can also send queries by e-mail.

North America:	1-877-DISCREET
(toll-free)	(1-877-347-2733)
Overseas:	(514) 954-7550
Fax:	(514) 954-7254
E-mail:	discreet.techsupport@autodesk.com
WWW:	http://www.lightscape.com
	http://www.discreet.com

You may want to join the Lightscape users e-mail group to keep in contact with the community of people who use Lightscape. For information on subscribing to the lvs-users group, visit our web site.

Readers' Comments

We would like to hear from you. Your comments can help us to improve the quality of the lessons and other documentation. Mail, fax, or e-mail your comments to:

Discreet
Documentation Department
10 Duke Street
Montreal, Quebec, Canada
H3C 2L7
Fax: (514) 954-7495
E-mail: docs@discreet.com

The Gallery

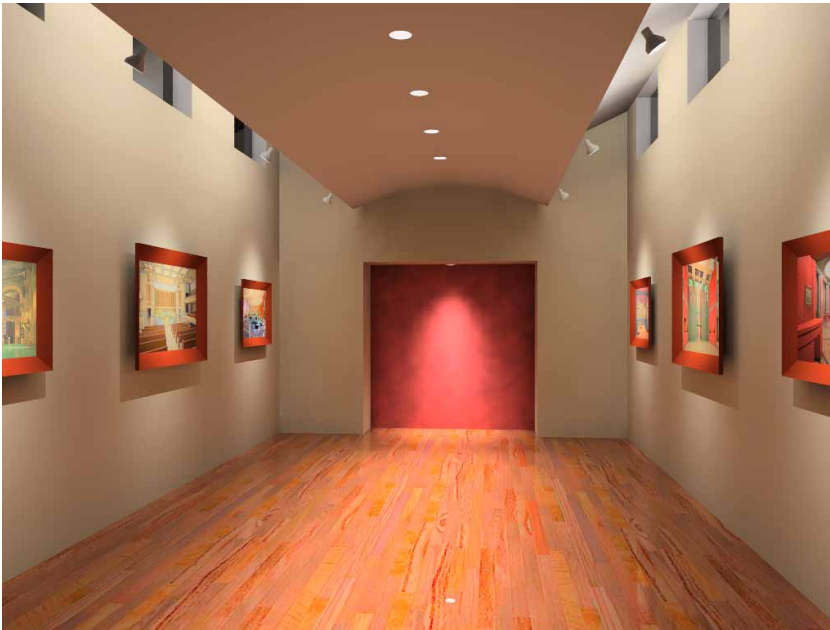
Because radiosity is fundamentally different from other visualization approaches, the best way to learn the new workflow is to work with simple models. Once you are familiar with the Lightscape interface and approach, you can proceed to more complex models. This lesson teaches you the Lightscape workflow using a model of a simple art gallery.

In this lesson:

- Start Lightscape and open a model
- Tour the components of the user interface
- Try the various display modes, view controls, and projections
- Define a new material with a texture
- Work with material libraries
- Aim spotlights to point at pictures
- Process and view the radiosity solution
- Assign a different material to a surface
- Change the photometric properties of a spotlight
- Create an image of the radiosity solution with and without ray tracing

1 The Gallery

At the end of this lesson, you create an image of the gallery that resembles the following illustration. This image was rendered with ray tracing after computing the radiosity solution.



Stage 1: Importing Data

When you import geometry into Lightscape, it is subsequently saved into a Lightscape Preparation file. The Preparation file for this lesson has already been set up for you.

Start Lightscape

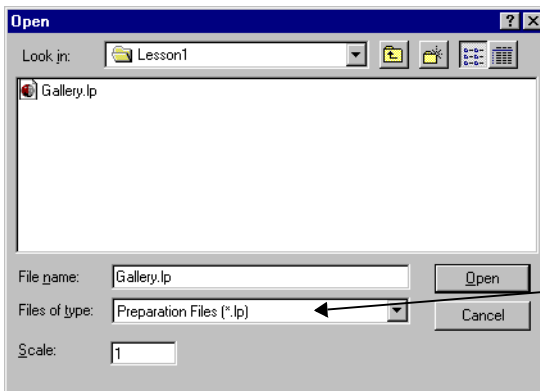
1. Click the Windows Start button.
2. Choose Programs | Lightscape | Lightscape.

The Lightscape interface appears on the screen.

Load a Preparation file

1. Choose File | Open.

The Open dialog appears.



Note: The Files of Type box should be set to Preparation Files (*.lp).

2. Change the path to *C:\Program Files\Lightscape\Projects\Lesson1*.

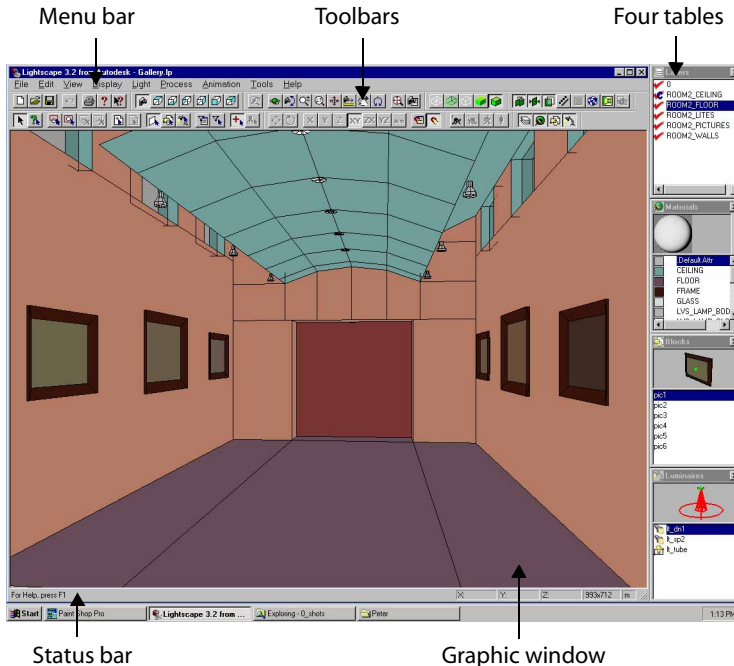
Note: This path assumes that you performed a typical installation. You must adjust the path if you installed the application to a different location.

3. Double-click the file *Gallery.lp*.

Lightscape displays a small art gallery with a number of light sources and picture frames on the walls.

About the Lightscape Interface

The Lightscape interface consists of the Graphic window, Layers table, Materials table, Blocks table, and Luminaires table.



The Graphic window is used to display and edit the geometry of the current model. By default, it is located on the left side of the screen and occupies the majority of the screen. The four tables are grouped together on the right side of the screen. You can reposition and resize all the windows as required.

Hint: The arrangement of the windows is stored when you exit Lightscape and used the next time you start the application.

Editing commands can be found on the toolbars, in the menus (on the Lightscape menu bar), and in the context menus (when you right-click in the four tables or Graphic window). Additional dialogs pop up when you choose some of the editing commands.

Hint: A tooltip shows the name of a toolbar button; position the cursor over any button and the tooltip appears.

Using the Mouse

Lightscape is designed for a two-button mouse. The left button is the action button and the right button is used to display a context menu.

When you move the mouse and press the left button in the Graphic window, Lightscape performs one of several actions, depending on the mouse mode selected. There are four mouse modes:

- **Select mode:** Use the mouse to select objects in the model.
- **Query mode:** Click an object in the Graphic window to display information about that object on the status bar.
- **Dynamic View mode:** When you select a view control such as Orbit or Rotate, dragging the mouse in the Graphic window dynamically changes your view of the model.
- **Special Selection mode:** Certain functions in Lightscape require special mouse operations. For example, you can use the Pick mode on the Transformation dialog to aim a luminaire. A special selection mode is usually started from the dialog related to the function being performed.

Clicking with the right mouse button in a window displays a context menu: a pop-up menu that contains commands appropriate to the current window or to the objects selected in it. For example, right-clicking in the Graphic window when a surface is selected displays a menu of commands that can only be performed on surfaces.

Layers Table

Each Lightscape object is assigned to a layer and the Layers table displays the names of all the layers. A check mark to the left of a layer name indicates that the layer is on and that the objects assigned to it are currently displayed in the Graphic window. Double-clicking a layer name toggles its state on and off. Right-click in the Layers table to display the Layers context menu, which contains functions appropriate to the layer selection set.

Materials Table

The Materials table lists all the materials currently available in the model. Double-clicking a material name activates the Material Properties dialog, which contains tools for changing the characteristics of the material. Right-click in the Materials table to display the Materials context menu, which contains functions appropriate to the material selection set.



← The material preview displays the material currently selected in the Materials table. A sample sphere with the material is shown on the left and a preview image of the material is shown on the right.

Blocks Table

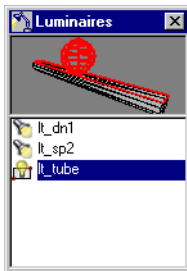
The Blocks table lists all the blocks available in the model. A block in Lightscape is a group of surfaces and/or other blocks that is given a common name and an insertion point. Double-clicking a block name isolates the block for display and editing in the Graphic window. Right-click in the Blocks table to display the Blocks context menu, which contains functions appropriate to the block selection set.



← The block preview displays the block currently selected in the Blocks table. You can change the view of the block (using the interactive view controls) and change the display mode. Right-click in the preview window to change the view control and display mode options.

Luminaires Table

The Luminaires table lists all the luminaires available in the model. A luminaire is a special type of block used to represent a light fixture in Lightscape. A luminaire includes the photometric characteristics that control how light energy is emitted from it. Double-clicking a luminaire name isolates the luminaire in the Graphic window and activates a dialog for editing the luminaire's photometric characteristics. Right-click in the Luminaires table to display the Luminaires context menu, which contains functions appropriate to the luminaire selection set.



← The luminaire preview displays the luminaire currently selected in the Luminaires table. This preview works the same as the block preview except it shows the fixture (block) and the luminous intensity distribution (LID).

Stage 2: Preparing Data

In this section, you perform the following tasks:

- Viewing the Model
- Defining Materials
- Using Material Libraries
- Displaying Textures
- Aiming Luminaires

Viewing the Model

You have complete control over the view of the model in the Graphic window. By using display modes, view controls, and projections, you can manipulate the view.

Turn on all toolbars

These lessons make extensive use of the toolbars to simplify operations. Check that all the toolbars are displayed.

1. Choose Tools | Toolbars.

The Toolbars dialog appears.

2. Double-click any toolbar name that does not have a check mark beside it.

3. Click Close.

View the full extents of the model in different projections

Often, you want to see everything on the screen at once. View Extents adjusts the view to fit the entire model in the Graphic window.

1. Choose Display | Show Axis to turn the axes display on.

The axes are displayed in the lower-left corner of the Graphic window.

2. Choose View | Projection | Perspective or click the Perspective button .

Hint: Buttons shown in the instructions in this guide are on the toolbars.

3. Choose View | Extents or click the View Extents button .

When the Perspective projection is displayed, View Extents orients the model so you view it from the front.

4. In addition to the Perspective projection, you can view the model in six orthographic projections. For example, choose View | Projection | Top or click the Top button .

The model changes to the plan or Top projection.

Note: In the orthographic views, View Extents simply zooms the model to fit in the Graphic window.

5. To return to the Perspective projection, click the Perspective button again.

Adjust the view interactively

The set of view controls makes it easy for you to interactively change your point of view.

1. Choose View | Interactive | Orbit or click the Orbit button .

You can only orbit in the Perspective projection.

2. Place the cursor anywhere in the Graphic window, press the left mouse button, and drag first horizontally and then vertically.

Notice the change of view in the model. This is called orbiting.

Set the display mode

Lightscape includes a variety of modes with which to display your model.

1. Choose Display | Wireframe or click the Wireframe button .

A wireframe representation provides the fastest display speed but can often become confusing with more complex models. Other modes provide better feedback but can be slower to display.

2. Choose Display | Solid or click the Solid button .

3. Continue to orbit the model.

Notice the orbiting is somewhat slower than in Wireframe mode but you can more easily understand the model.

4. To see the model more clearly, choose Display | Outlined or click the Outlined button .

The result is similar to Solid mode except the surface edges are outlined.

5. Click the Solid button.
6. You can further enhance both Solid mode and Outlined mode by adding some simple shading. This creates an effect as if you have a headlight mounted on the camera as you move around.
 - a) Choose Display | Enhanced or click the Enhanced button .
 - This option may already be turned on.
 - b) Click the Enhanced button again to turn the shading off.
7. As you orbit the model, notice that some of the surfaces such as the walls and ceiling disappear. This is a display technique called backface culling. In this display mode, surfaces that are oriented away from your viewing position are removed (culled) from the display.
 - a) Choose Display | Culling or click the Culling button  to turn off backface culling.
 - Notice the change in the display as you orbit.
 - b) Click the Culling button again to turn it back on.
8. Choose View | Display Original View to open the original view of the model.

It is easy to restore the view displayed when you first opened the model. The view when you last saved the model is the original view the next time you open it. For more information on views, see “Viewing the Model” on page 49.

Try other interactive view controls

In addition to orbiting, there are several other controls you can use to interactively set the viewpoint.

1. Try panning:
 - a) Choose View | Interactive | Pan or click the Pan button .
 - b) Place the cursor anywhere in the Graphic window, press the left mouse button, and drag first horizontally and then vertically.
 - Notice the change of view in the model. This is called panning.

2. You will often need to use a combination of view controls to navigate the model. The most convenient way to do this is to hold down the hot key as you move the mouse in the Graphic window.

a) Hold down the following hot keys as you drag the mouse:

- O for Orbit
- R for Rotate
- Z for Zoom
- P for Pan
- D for Dolly
- S for Scroll
- T for Tilt

The chosen view control is only set while you hold down the hot key. When you release the hot key, the mouse reverts to whatever mode it was in. You can permanently set the mouse mode by pressing the Shift key with the hot key.

b) Press Shift-O to select the Orbit control.

c) When you are finished experimenting, choose View | Display Original View to reset your view.

Note: For more detailed descriptions of the view controls, see “Viewing the Model” on page 49.

3. The Zoom Window view control works a bit differently:

a) Choose View | Interactive | Zoom Window or click the Zoom Window button .

b) Drag in the Graphic window to draw a Zoom rectangle.

c) To reset your view, choose View | Interactive | Undo Zoom Window or click the Undo Zoom Window button .

Defining Materials

Because Lightscape is based upon the physics of lighting, defining materials properly is important to obtain meaningful results.

Define a wood floor material

1. Create a new material:

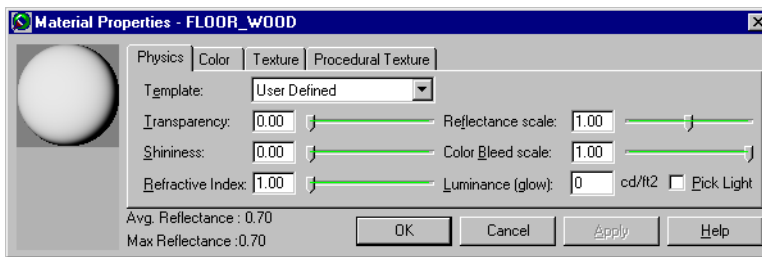
a) Right-click in the Materials table, then choose Create.

A new material, called “Item17”, is added to the table and highlighted in blue. You can now enter a name for the material.

b) Type FLOOR_WOOD and press Enter.

2. Double-click the FLOOR_WOOD material.

The Material Properties dialog appears.



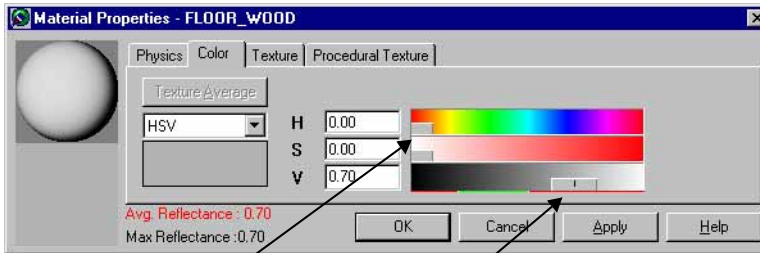
On the Physics panel, there are several parameters that you can set to define the characteristics of a material. You should start by selecting a template that most closely matches the type of material you are defining. This helps you select valid values for a specific type of material.

3. Select Wood Varnished from the Template list.

The Shininess and Refractive Index change. The various parameter sliders are marked with red and green lines. The green portions of these lines represent the valid range of values that you should use when defining a material.

4. Click the Color tab.

The Value slider controls the average and maximum reflectance of a material. The reflectance represents the amount of diffuse light energy that is reflected from a surface. A material with a reflectance of 0.20 reflects 20% of the energy and absorbs 80% of the energy that it receives.



Hue and Saturation sliders

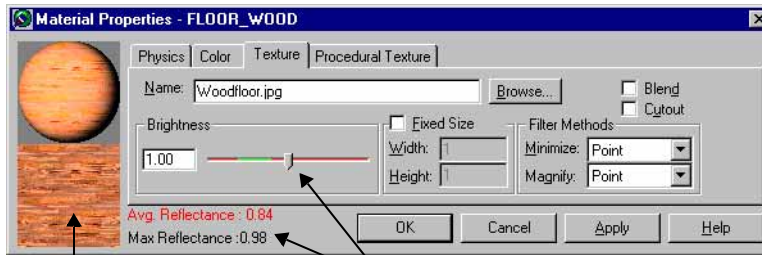
Value slider with green line that shows valid range

The green and red lines on the Value slider represent the valid range of values for varnished wood. Notice that the Hue (H) and Saturation (S) sliders of the material color have no lines. This means that you can use any values.

5. Move the Value slider and note how the average and maximum reflectance change. (The average and maximum reflectance for a color are always the same.)
6. Many materials contain patterns or textures. A wood floor is a good example of such a material. To define these materials, you can use image bitmaps (called texture maps) to modify the color of a material across a surface.
 - a) Click the Texture tab.
 - b) Click Browse.
 - c) Double-click *Woodfloor.jpg* in the File Name list to load this texture map.

Note: Since the image file for this texture is in the current folder, you do not have to add its path to the Textures list in the Document Properties dialog. If you select a file from a folder that is not included in the paths, a dialog appears to automatically add the path. For more information about paths, see Chapter 7, "Using Materials" in the *Lightscape User's Guide*.

Hint: Another way of loading the texture map is to drag the *Woodfloor.jpg* bitmap from Windows Explorer (or another utility like LVu) to the Name box.



The texture map appears in the material preview.

The Brightness slider controls the Avg. Reflectance and Max Reflectance.

7. The Brightness slider controls the average and maximum reflectance of the material. Since the Brightness of this texture is outside the recommended range of values, the average reflectance of the material (displayed at the bottom of the dialog) is displayed in red.

Set the Brightness slider to 0.25 so that the Avg. Reflectance is about 0.21 and the Max Reflectance is about 0.25. For texture maps, the average and maximum reflectance can differ. It is generally the average reflectance that is important.

8. Enable Fixed Size and type 3 (meters) in both the Width and Height boxes.

These numbers represent the real world dimensions of the texture being used.

9. Click the Color tab.

If a texture map is used, the material's reflectance is set using the brightness of the texture. If there is no texture, reflectance is set using the value of the color. Notice that there are no red and green lines on the Value slider since a texture map is loaded. If you turn off texture display in your scene, surfaces that use this material are displayed using the average color of the material's texture.

10. Click the Texture Average button to set the material color to the average color of the texture map being used.

Note: Regardless of whether a texture is displayed or not, when you run the radiosity process, the software will always use the texture to calculate the light reflectance if one is associated to the material.

11. Click the Physics tab.

12. Set the Shininess slider to 0.90.

This value determines how shiny the floor appears when the image is ray traced.

13. Click OK to set the new material.

You use the FLOOR_WOOD material later in this lesson.

Using Material Libraries

Now that you have defined a material, you can save it to a material library. When working on another project, you can load the material from the library.

Save the material definition to a material library

1. Right-click the FLOOR_WOOD material in the Materials table, then choose Save.

The Save As dialog appears.

2. Type Floors.atr in the File Name box.

If necessary, change the path to *C:\Program Files\Lightscape\Projects\Lesson1*.

3. Click Save.

In the warning dialog, click Yes to confirm the creation of a new library file. You can add additional materials to this library at any time.

Load material definitions from a material library

You can load materials from a material library to use in your project. If the name of the material you are loading already exists in the Materials table, then the new material replaces the existing definition. This is a convenient way of storing alternate material palettes for a project.

1. Right-click in the Materials table, then choose Load.

The Open dialog appears.

2. Select *Gallery.atr* in the File Name list.

If necessary, change the path to *C:\Program Files\Lightscape\Projects\Lesson1*.

3. Click Open.

The Available Materials dialog appears.

4. Click Select All to select all materials.
5. Click OK to load the material definitions.

In the warning dialog, click Yes to confirm the overwriting of all textures.

The appearance of the materials used for the ceilings, walls, and floor changes because these materials were predefined and stored in *Gallery.ctr* and they overwrote the materials of the same name when they were loaded.

Displaying Textures

In the Material Properties dialog, you can assign a texture to a material. After the material with a texture is assigned to a surface, you can turn on Textures to display them in the Graphic window. In this model, texture maps are applied to the areas inside the picture frames.

Turn on textures

1. Choose Display | Textures or click the Textures button .

Allow some time for the textures to load into memory. Check the status of texture loading on the status bar.

2. Click the Textures button again to turn off the textures.

If your graphic card does not provide texture memory, then the interactivity of the model improves when textures are not displayed.

You can see if a texture is loaded by looking at the Materials table. The colored square next to each material displays the material color. A colored texture symbol  next to the material name indicates the texture is loaded and displayed in the Graphic window. A black and white texture symbol indicates that a texture has been loaded but is not currently displayed. A green  indicates that the texture file associated to the material could not be found.

When you modify or replace a texture image file, you must reload textures to update the model. To reload all the textures used in the scene, choose Display | Reload Textures. To reload textures only for selected materials, right-click in the Materials table and choose Reload Texture.

Aiming Luminaires

Lightscape creates a special class of blocks called luminaires, which represent light fixtures in the model. You create a luminaire by associating photometric data with an existing block. As with materials and regular blocks, you can store luminaires in libraries and use them again in different models. In this lesson, the luminaires have been set up for you in the model.

Although Lightscape is not a general modeling program, several tools allow you to easily add and position objects and luminaires in your models.

Aim the spotlights so that they point at the picture frames

1. Select a spotlight:

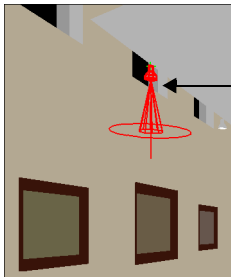
- a) Choose Edit | Selection | Select or click the Select button .

This sets the function of the left mouse button to Select mode, the normal operating mode.

- b) Choose Edit | Selection | Luminaire or click the Luminaire button .

This sets selectable objects to luminaires only.

- c) Click on or near one of the spotlights.

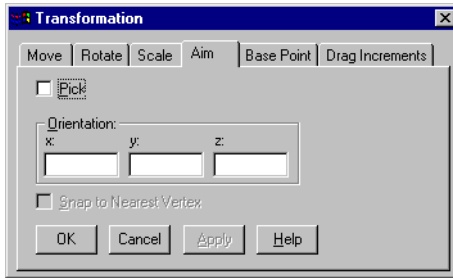


The spotlight closest to where you click is highlighted.

2. Choose Edit | Transformation or right-click anywhere in the Graphic window and choose Transformation.

The Transformation dialog appears.

3. Click the Aim tab.

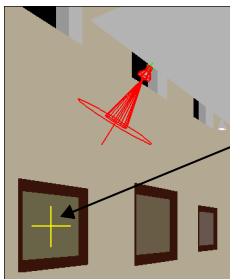


4. Use the Pick feature to aim the spotlight:

- a) Enable Pick.

This sets the function of the left mouse button to a special selection mode used for picking the focus point of the luminaire.

- b) Pick a point roughly in the center of the picture frame under the selected luminaire.



Lightscape aims the selected luminaire at the point you pick.

- c) Disable Pick to deactivate the Pick mouse mode.

5. Select another luminaire to aim.

If the first luminaire remains selected after you select a second one, the Accumulate Pick feature is turned on. To turn it off, click the Accumulate Pick button .

Hint: You can move the Transformation dialog if it is blocking your view. If the area behind the dialog remains black, click the title bar of the Graphic window, and press F5 or choose Display | Refresh to refresh the view.

6. Enable Pick to reactivate the Pick mouse mode.

7. Pick a point roughly in the center of the picture frame under the selected luminaire.
8. You can greatly reduce the number of mouse movements you have to perform by using hot keys. In this step, use a hot key to temporarily deactivate the Pick mouse mode and activate the Select mode.

a) Hold down the 1 key to temporarily activate Select mode.

Notice that the Select button is selected.

b) While holding down the 1 key, select another luminaire.

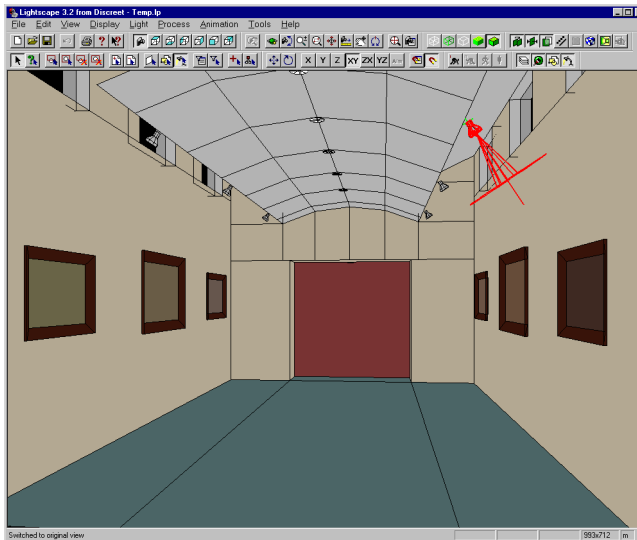
c) Release the 1 key, then select the point on the picture where you want the luminaire to aim.

9. Repeat step 8 until all six spotlights are pointing toward their respective paintings.

During this process, you can adjust the view to make it easier to pick points by holding down a view control hot key. For example, press the O key to orbit the model.

10. Click OK on the Transformation dialog when you are done aiming the luminaires.

The following illustration shows how the spotlights should be aimed prior to radiosity processing.



11. Choose View | Display Original View to reset the view.

12. Choose Edit | Selection | Deselect All or click the Deselect All button .

Save the file

Save the work you have done so far so you can return to this point later to modify the geometry or add lights.

Note: Lightscape's Undelete feature restores the most recently deleted object, if the model database has not changed in the interim. To use this feature, choose Edit | Undelete or click the Undelete button . In general, use the Save and Save As functions to give yourself a point to return to when necessary.

1. Choose File | Save As.

The Save As dialog appears.

2. Type Temp.lp in the File Name box, then click Save.

Hint: You should use a filename other than Gallery.lp so the original file for this lesson is not changed.

Stage 3: Creating the Radiosity Solution

In this section, you perform the following tasks:

- Initiating the Preparation Model
- Setting Processing Parameters
- Processing the Solution

Initiating the Preparation Model

You have now completed the Preparation stage and can proceed to the Solution stage for radiosity processing by initiating the Preparation model.

Begin the Solution stage

1. Click the Solid button .
2. Choose Process | Initiate or click the Initiate button .
3. If you made changes to the model since the file was saved, a dialog appears. Click No to proceed without saving changes.

After initiation, the model appears dark because the processing has not yet started and no energy has been distributed from the light sources. This is called the Zero Energy State (0 iterations).

Notice that the filename in the title bar of the Graphic window has the .ls extension, indicating that the model can now be saved to a Solution file.

Setting Processing Parameters

The most important part of the Solution stage is choosing processing parameters to control how accurately the light effects are calculated. These settings, more than anything else, determine the size of the final model (measured in polygonal elements), its quality, the accuracy of the light distribution, and the time it takes to process the model. For more information, refer to Chapter 11, “Radiosity Processing” in the *Lightscape User’s Guide*.

Set the processing parameters

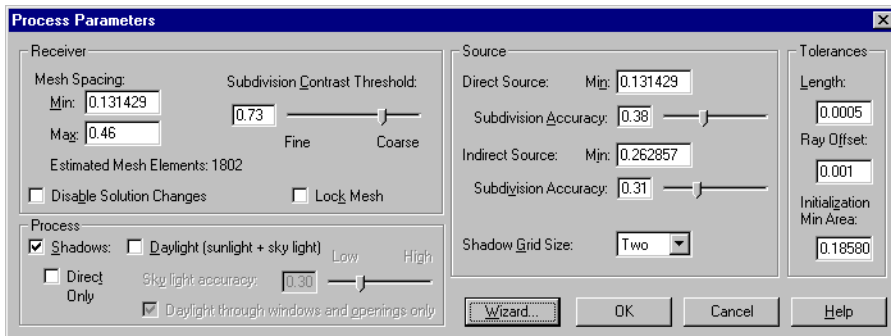
1. Choose Process | Parameters.

The Process Parameters dialog appears.

The numerical values in the various boxes of the dialog control the quality and speed of the Lightscape radiosity process. Although the values can be set individually, you use a wizard in this exercise to set them automatically.

2. Click the Wizard button to start the Process Parameters Wizard.
3. Select 2 for a moderate quality level, then click Next to show the next page of the wizard.
4. Select No so daylight is not considered in this model, then click Next to show the next page.
5. Click Finish to update the processing parameters based on the selections made in the wizard.

Notice the change in the numerical values in the various boxes of the dialog.



Note: Because various aspects of the model, such as the size and lighting conditions, are considered, answering the questions in the wizard in the same way for two different models can result in different values.

6. Click OK to close the Process Parameters dialog.

Processing the Solution

Lightscape uses a process called progressive refinement radiosity to compute the simulation in successive iterations. At each iteration, the software selects the brightest light source and computes its contribution to all the surfaces in the scene. For example, if your model includes daylight, the sun is usually processed in the first iteration.

Once it accounts for all the primary light sources, Lightscape starts computing the interreflections of light between surfaces. At each iteration, the system selects the brightest reflecting surface and computes its contribution to the rest of the environment.

Thus, the radiosity solution is refined at each iteration; it is a progressively better approximation of the final result.

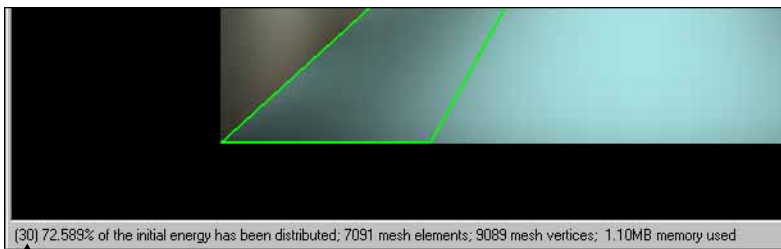
In principle, this refinement process continues until the system accounts for all interreflections of light. In practice, however, the simulation converges very rapidly toward the final result, so that visual differences between successive iterations become unnoticeable after only a fraction of the surfaces (but the most important of them) have reflected their light contribution back into the environment.

Start the radiosity processing

1. Choose Process | Go or click the Go button .

Lightscape calculates the light distribution in the given environment. It can take from several seconds to several hours to completely process the light energy distribution. Processing time depends on such factors as model size, number of light sources, meshing parameters, and hardware resources.

2. For the current model, let the process run for about 30 iterations, which should process approximately 75% of the available energy.



The number of iterations processed appears on the status bar in parentheses.

3. Choose Process | Stop or click the Stop button .

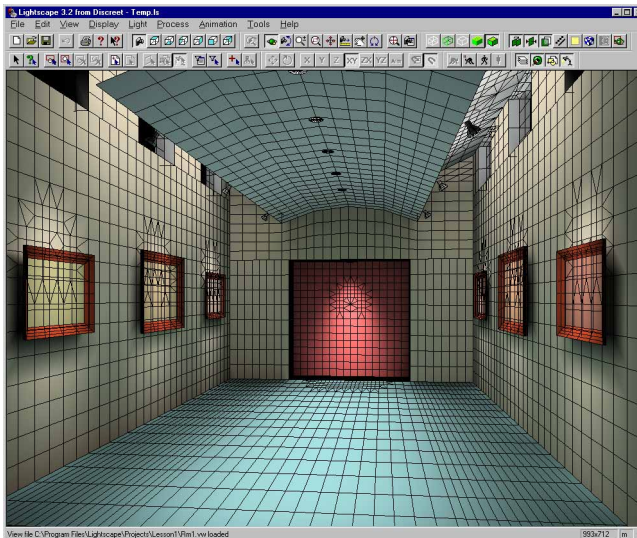
The process stops after the current iteration is complete, which may take a few seconds. The message “Preparing to stop processing” appears on the status bar. When processing completely stops, the message “Processing Stopped” appears.

Note: In this lesson, the scene is redrawn between each iteration so that you can better understand the process of progressive refinement radiosity. Each redraw slows down the overall processing. With a larger project, you would typically set up the model and then run the radiosity processing using the Israd batch processing utility for the fastest results. This procedure is described in Lesson 7.

Observe the model. Note the color bleeding on the ceiling and walls obtained as Lightscape distributes the reflected light energy from the floor. Also note how the brightness of the room evens out as the energy is distributed among the surfaces.

Observe the underlying radiosity mesh

1. Click the Outlined button .



Notice that the mesh elements are smaller and more dense around areas where there are light beams or shadows.

Lightscape uses adaptive subdivision to make sure that the sample points (the vertices of each mesh element) used for determining the energy transfer received at a surface are created only where required.

In areas where there is not a high contrast in lighting intensity, the samples can remain sparse (mesh element size is larger), whereas in areas where there are significant changes in intensity, such as along shadow boundaries and light beams, the sampling is more dense.

The key to controlling a radiosity solution lies in the processing parameters that you set. These parameters determine the meshing density and the resulting accuracy of the solution. For more information on adaptive subdivision, see “Global Processing Parameters” on page 124.

2. Click the Solid button  to return to Solid display mode.

Adjust the view using Level of Detail control

One of the most important features of Lightscape is the ability to interactively move around and explore a fully rendered model.

1. Click the Orbit button , then orbit around the model.

You can display different views of this model quickly because Lightscape stores the calculation for diffuse lighting distribution as an integral part of the 3D surfaces in the model in the form of a 3D radiosity mesh structure.

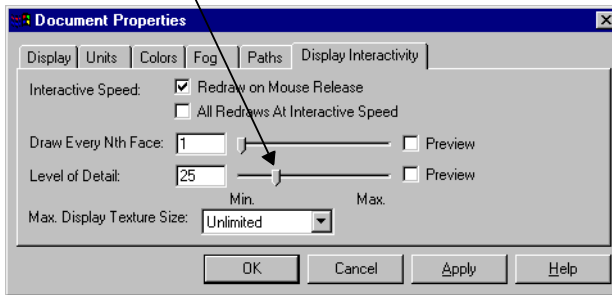
Once calculated, the system can display the radiosity mesh from any point of view by interpolating the colors that are stored at the vertices of each mesh element. This simple and fast display technique (often referred to as Gouraud shading) is supported by OpenGL and can be further accelerated using an OpenGL compatible 3D graphic acceleration board.

One way to increase the interactive display speed of a radiosity solution is to use Level of Detail control.

2. Choose File | Properties.
3. Click the Display Interactivity tab.

4. Try a medium level of detail:

a) Set the Level of Detail slider to approximately 25, then click Apply.



Hint: You can enable the corresponding Preview check box to see the result without applying the change.

b) Observe the model as you continue orbiting.

5. Try a low level of detail:

a) Set the Level of Detail slider to 1, then click Apply.

b) Orbit the model again.

6. When you are finished experimenting, reset the Level of Detail slider to 100, then click OK.

7. Choose View | Display Original View.

Note: The view that is currently the original view is also saved as *Rm1.vw*. You can choose View | Open to open a saved view.

Stage 4: Fine-Tuning the Solution

In this section, you perform the following tasks:

- Assigning Materials
- Changing a Light Source

Assigning Materials

An important feature of Lightscape is the ability to make changes to the surface materials and assign different materials to surfaces after starting a radiosity solution. This allows you to rapidly test various design alternatives and make adjustments to the materials and lighting to set the precise look you want.

Change the carpeted floor to a wood floor

1. To see the texture maps on the surfaces, click the Texture button .
2. To assign the wood floor material to the floor, drag FLOOR_WOOD from the Materials table to the floor surface. Release the mouse when the surface edges are highlighted with blue and yellow dashed lines.

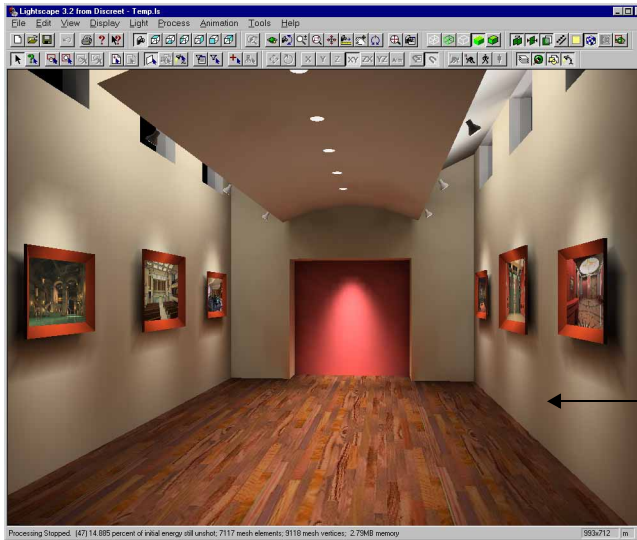
Note: There is also another way to assign a material. After you select one or more surfaces, right-click in the Graphic window and choose Assign Material. In the Assign Material dialog, select a material from the list, then click OK. Note that you must use this method to assign a material to several surfaces at the same time.

Notice that the color of the floor changes as soon as you assign the material, but the color of the interreflection from the floor to the ceiling and walls does not change. To update the interreflections, you need to continue the radiosity processing.

Continue processing a few more iterations

1. Click the Go button .
2. After a few iterations, click the Stop button  to interrupt the processing.

Notice the color change in the room.



The color of the floor
bleeds onto the walls.

After only a few iterations, you can see the effects of the material change on adjacent surfaces. The color of the walls and ceiling has changed significantly. This demonstrates color bleeding, an important effect that various materials and lights can have on the overall aesthetics of a room.

Perform a lighting analysis

The ability to simulate color bleeding is one of the key advantages that Lightscape offers over conventional rendering technologies. Another advantage is that you can perform a meaningful lighting analysis of your solution because luminaires can be specified using real-world photometric values (including specifications from manufacturers).

1. Choose Light | Analysis.

The Lighting Analysis dialog appears.

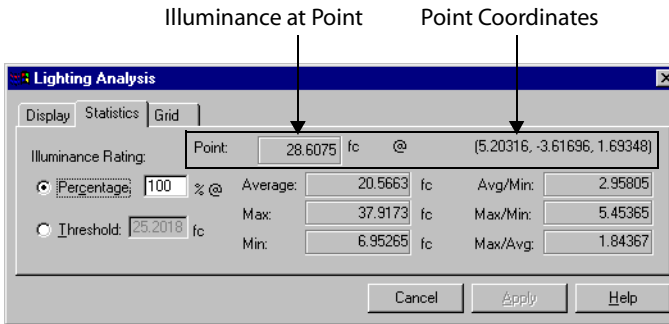
2. Click the Statistics tab.

You can now use the mouse as a “virtual light meter” in your model.

3. Check the illuminance on the pictures:

a) Pick a point in the middle of one of the pictures.

Information about the illumination level of the picture is displayed in the dialog.



Note: The Illuminance Rating and the Average, Maximum, and Minimum statistics are computed over the area of the selected surface.

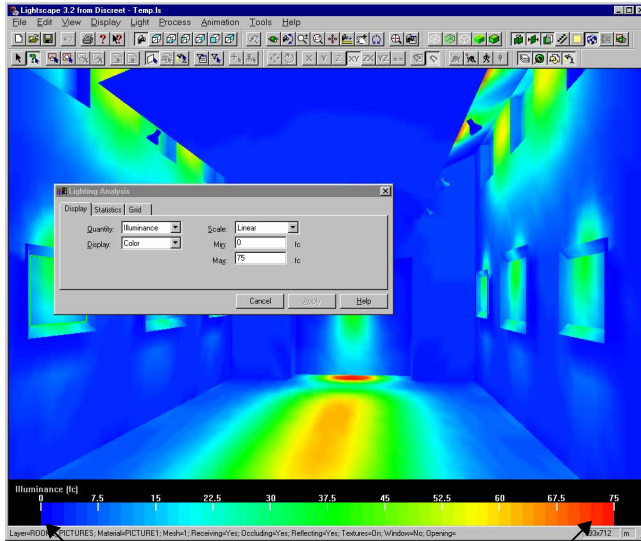
b) Check the illuminance at the selected point to see if the lighting is too bright; most galleries do not want more than 20 footcandles on a picture.

In the next exercise, you learn how to easily modify a luminaire to test different settings.

4. Click the Display tab.

5. Select Color from the Display list.

6. Type 75 in the Max box, then click Apply.



The linear Illuminance scale ranges from 0 to 75 fc.

The model is now displayed in a pseudo-color mode for easy visualization of the lighting conditions. Move around interactively to better evaluate the light levels of the room.

7. To exit this display mode, click Cancel.
8. Choose View | Display Original View.

Changing a Light Source

Just as you can change a material, you can also change the properties of a luminaire.

Note: You cannot position or aim a luminaire in the Solution model. Modifying the geometry is only possible in the Preparation model.

Change the properties of one of the luminaires

1. Click the Select button , then click the Luminaire button .

1 The Gallery

2. Pick a point on or near one of the spotlights aimed toward a picture.

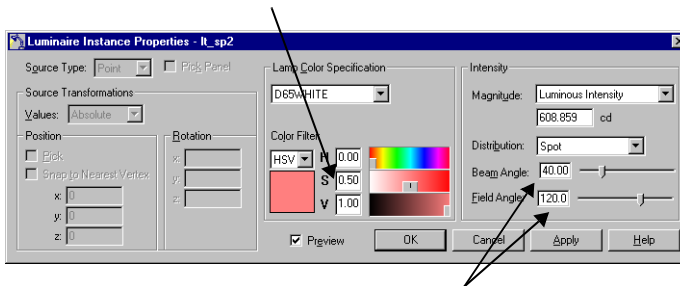
The selected luminaire is highlighted.

3. Right-click in the Graphic window, then choose Photometrics.

The Luminaire Instance Properties dialog appears.

4. Adjust the photometric properties for the selected luminaire:

a) Change the Saturation (S) of the color filter to 0.50. The Hue (H) and Value (V) should be set to 0 and 1.00, respectively.



b) In the Intensity group box, set the Field Angle slider to 120 and the Beam Angle slider to 40.

Note: It is important to set the field angle before setting the beam angle because the field angle acts as a maximum constraint for the beam angle.

The shape of the spotlight's distribution changes.

5. Click OK and confirm the overwrite.

6. Click the Deselect All button .

Continue processing a few more iterations

You must run the simulation for a few more iterations to allow Lightscape to process the new properties of the modified luminaire. The system first subtracts the original light energy from the environment and then adds the redefined light energy back into the environment.

1. Click the Go button .
2. After a few iterations, click the Stop button  to interrupt the processing.

The light emitted from the luminaire is red.

Stage 5: Outputting the Results

In this section, you perform the following tasks:

- Creating a Radiosity Image
- Creating a Ray-Traced Image
- Viewing Image Files with LVu

Creating a Radiosity Image

Once you are satisfied with the radiosity solution and the materials and luminaires are set the way you want, you can rapidly save images of the model using OpenGL display techniques.

Typically, OpenGL is used to interactively display the radiosity solution on your screen. Using OpenGL, you can render only the diffuse lighting effects calculated during the radiosity solution. You cannot render shiny specular reflections and true transparency effects, nor can you render the effects of procedural bump maps or intensity maps that may be part of a material definition.

If you are satisfied with the quality of the radiosity solution you see, then OpenGL offers the fastest method for creating images. This can be particularly important if you are rendering many images (for example, animation frames).

Create an image showing the diffuse lighting of the gallery only

1. Choose View | Display Original View.
2. If necessary, click the Textures button  to turn on textures.
3. Choose File | Render.

The Rendering dialog appears.

4. In the Output File group box, select Targa (TGA) from the Format list.
5. Type Radiosity.tga in the Name box, then click OK.

Lightscape takes a snapshot of the current view and saves the image to disk. Rendering progress messages appear on the status bar.

Note: You can further improve the quality of any image you create with Lightscape by using antialiasing to remove jagged edges. The higher the level of antialiasing specified, the longer it takes to create the image.

Creating a Ray-Traced Image

For the highest quality image, use the ray tracer to add specular reflections and enhanced lighting effects. The ray tracing process used in Lightscape is faster than traditional ray tracing because the ray tracer does not have to calculate the direct lighting; this is already calculated with the radiosity process. For some light sources (such as sharp spotlights and sunlight), you may choose to ray trace the direct light to obtain better quality shadows.

Set a luminaire to be ray traced

1. Click the Select button , then click the Luminaire button .

2. Select one of the spotlights aimed toward a picture.

The selected luminaire is highlighted.

3. Right-click in the Graphic window, then choose Luminaire Processing.

The Luminaire Processing dialog appears.

4. Enable Ray Trace Direct Illumination, then click OK.

The other options are already enabled.

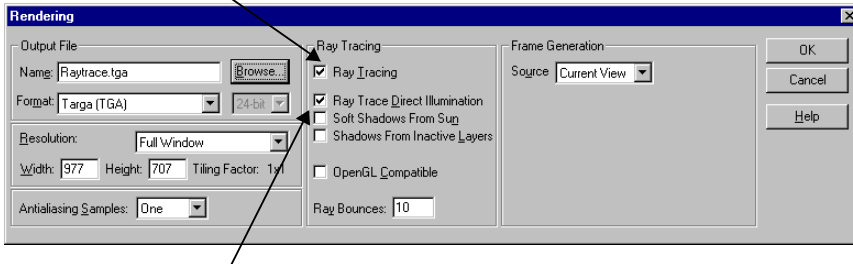
Create an image using ray tracing

1. Choose File | Render.

The Rendering dialog appears.

2. Set the Ray Tracing options:

a) Enable Ray Tracing.



b) Enable Ray Trace Direct Illumination.

This indicates that you want to ray trace any luminaires that have been set for ray tracing.

3. In the Output File group box, type Raytrace.tga in the Name box.

Note: If you leave the Name box blank, the system renders the image and displays it on the screen but does not save it to disk.

4. Select Targa (TGA) from the Format list.

5. Click OK, then confirm that some additional processing can occur before ray tracing starts.

Lightscape does an iteration for each light source set for ray tracing (in this case, only one) to remove the direct contribution to the surfaces in the model. Once this is complete, the ray tracing begins.

Soon, you see the specular reflections from the floor surface that were not visible in the diffuse radiosity model or in the OpenGL rendered image. You also see the results of the procedural intensity map, which gives an “aged plaster” effect to the red wall in the alcove (the recess in the wall).

You may notice that some of the surfaces appear brighter than they did when rendered using OpenGL. This is particularly true of texture mapped surfaces and surfaces that have a self-emitted illuminance. This difference occurs because the Lightscape ray tracer handles the interaction of light with these surfaces in a different (and more accurate) way than OpenGL does.

You can easily adjust the brightness of any material before ray tracing by adjusting the V (Value) slider on the Color panel of the Material Properties dialog. If the material has a texture, adjust the Brightness slider on the Texture panel. For a review of material properties, see “Defining Materials” on page 17.

Try the Ray Trace Area feature

To quickly test the materials in your model before committing to a long, high-resolution ray trace rendering or animation, you can use the Ray Trace Area feature to ray trace only a selected region on the screen and check if the materials require any adjustments.

1. To clear the current ray traced image, choose Display | Refresh or press F5.
2. Because one of the luminaires is set to Ray Trace Direct Illumination, you should turn on this option for the Ray Trace Area feature:

a) Choose Display | Ray Trace Area Options.

The Ray Trace Area Options dialog appears.

Note: All of these options are also available on the Rendering dialog.

b) Enable Ray Trace Direct Illumination, then click OK.

3. Choose Display | Ray Trace Area or click the Ray Trace Area button .

This sets the function of the left mouse button to a special selection mode used for the Ray Trace Area feature.

4. Drag in the Graphic window to select the area to be ray traced.

For example, draw a rectangle around the picture that receives light from the luminaire with the Ray Trace Direct Illumination option enabled.

5. After you release the mouse button, a warning appears. Click OK to confirm that some additional processing can occur before the ray tracing is started.

Note: This warning only appears when Ray Trace Direct Illumination is enabled (in the Ray Trace Area Options dialog).

Finish your session in Lightscape

1. Choose File | Save As to save this file.

The Save As dialog appears.

2. Type Gallery.ls in the File Name box, then click Save.

3. Choose File | Exit.

Viewing Image Files with LVu

The LVu utility allows you to view images, such as textures and Lightscape renderings, within a selected folder. You can display a full-sized image or all the images in a folder (as thumbnails). LVu supports the following file formats:

- .bmp (Windows native file format)
- .tga (Targa format)
- .tif (TIFF)
- .rgb (Silicon Graphics native file format)
- .jpg (JPEG)
- .gif (CompuServe 8-bit format)
- .png (Portable Net Graphics)

Start LVu and open the Lesson1 folder

1. Click the Windows Start button.
2. Choose Programs | Lightscape | LVu.

The LVu interface appears.

Hint: You can also start the LVu utility in Lightscape by choosing Tools | LVu.

3. Open all the images in the *Lesson1* folder:

- a) Choose File | Open Dir or click the Open Directory button .
- b) Change the path to *C:\Program Files\Lightscape\Projects\Lesson1*.
- c) Click OK.

Ten thumbnails of image files appear.

4. View the image that only includes radiosity:

- a) Double-click the *Radiosity.tga* thumbnail to display the full-sized image.

You produced this image in “Creating a Radiosity Image” on page 37. Notice how natural the diffuse surfaces appear, although the floor and the glass (which are specular surfaces) are less realistic.

- b) Double-click the image to return to the thumbnail images.

5. View the image that only includes ray tracing:

- a)** Double-click the *Rayonly.tga* thumbnail to display the full-sized image.

This image of the gallery was produced using only ray tracing. The quality is like that obtained using conventional scanline rendering and ray tracing programs. Notice that the floor and glass, which are specular surfaces, look compelling but the non-specular surfaces, such as the walls and ceiling, appear flat.

- b)** Double-click the image to return to the thumbnail images.

6. View the image that includes both radiosity and ray tracing:

- a)** Double-click the *Raytrace.tga* thumbnail to display the full-sized image.

You produced this image in “Creating a Ray-Traced Image” on page 38. It is the most accurate and, therefore, the most compelling of the three images.

- b)** Double-click the image to return to the thumbnail images.

7. Choose File | Exit to exit LVu.

Summary

Congratulations! You have reached the end of Lesson 1. At this point, you should:

- Feel comfortable with the Lightscape interface
- Have a better understanding of how Lightscape simulates the interaction between light and materials
- Understand the major steps in the Lightscape workflow.

The Sunroom

In Lesson 1, you started the workflow by opening a Lightscape Preparation file. In this lesson, you start by importing a DXF file of a sunroom. This type of file is output by a CAD or modeling application. Once the geometry is imported, you work with the model by defining realistic materials and lights.

In this lesson:

- Import a DXF file
- Create and save views of the model
- Organize the data into layers
- Define a block as a luminaire
- Add a luminaire from a block library
- Edit the properties of several materials
- Process the radiosity solution with different parameters
- Change surfaces into windows
- Turn on daylight and turn off interior luminaries
- Output the radiosity solution with ray tracing
- Perform a lighting analysis of the solution

2 The Sunroom

At the end of this lesson, you create a rendered image of the sunroom that resembles the following illustration.



Stage 1: Importing Data

In this section, you learn about:

- Objects Supported by Lightscape

You perform the following task:

- Checking Import Units and Scale Factor

Objects Supported by Lightscape

The Sunroom model for this lesson is a DXF file. This file format was initially defined by Autodesk and is now the A/E/C (Architecture/Engineering/Construction) standard for the exchange of geometric data. Most commercial CAD and modeling applications can export DXF files.

Lightscape currently imports most of the DXF objects that can be converted to polygons in 3D. These objects include:

- 2D Polyline (with width or height)
- 3D Polyline
- 3D Face
- 3D Polygon Mesh
- 3D Polyface Mesh
- Arc (with height)
- Circle (with height)
- Line (with height)
- Solid Trace

ACIS solids, lights, and cameras are not supported. To import these DXF objects, use the DWG or 3D Studio (.3DS) formats. For more information on importing, see Chapter 5, “Importing Geometry” in the *Lightscape User's Guide*.

Checking Import Units and Scale Factor

Many of the problems encountered by new Lightscape users occur because units are not set correctly when the file is imported.

Because Lightscape is a physically based lighting renderer, correct physical measurements are vital to getting accurate results. For example, the seat of a chair should be 16 inches from the floor, not 16 feet, and a wall should be 8 feet high, not 8 inches or 8 meters high.

- You can choose millimeters, centimeters, meters, kilometers, inches, feet, or miles as the units in which the model was created.
- You can apply a scale factor by entering a number in the Scale box, if required.

Load the DXF file named Sunroom.dxf

Lightscape is not a general modeling program, so you must import the geometry. A DXF file is one of several types of geometry files you can import.

1. Start Lightscape if the application is not already running.

2. Choose File | Import | DXF.

The Import DXF dialog appears.

3. Click the Browse button beside the Name box.

The Open dialog appears.

4. Change the path to *C:\Program Files\Lightscape\Projects\Lesson2*.

5. Double-click *Sunroom.dxf*.

6. Select Feet from the File Units list.

This model was created with one unit in the CAD program set to equal one foot in the real world. It is important to indicate the units of measurement when you import a model.

Note: The Import DXF dialog includes several other options for converting XYZ coordinates, translating geometry, mapping materials, and using block and luminaire libraries. For this exercise, you can use the default settings for these options. For more information, see “Using Advanced DXF Import Features” on page 101.

7. Click OK.

A dialog reports that the model measures 30.0622 x 17.15 x 9.50054 feet.

Note: This is the first way you can check if the units you specified were correct. These measurements seem reasonable for this model.

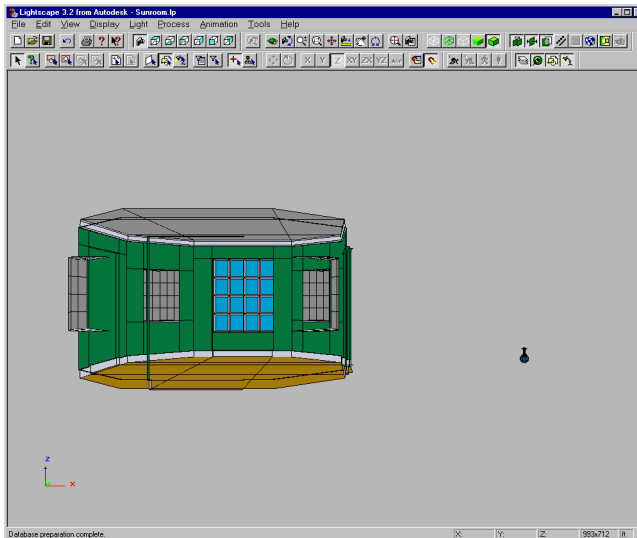
8. Click Yes to open and display the file.

Check the units and scale of the geometry

It is important to confirm the scale of the model before you start to work on it.

1. To see the surfaces and their edges, click the Outlined button .
2. If necessary, click the View Extents button  to see everything on the screen at once.

When you imported the model, the view was probably set to View Extents automatically. Because the Perspective projection is displayed, View Extents orients the model so you view it from the front.



3. The display unit used for the model is shown on the right end of the status bar. Check that this unit is set to feet (ft).

Note: To change the display unit, choose File | Properties and click the Units tab.

4. Choose Tools | Measure Distance.

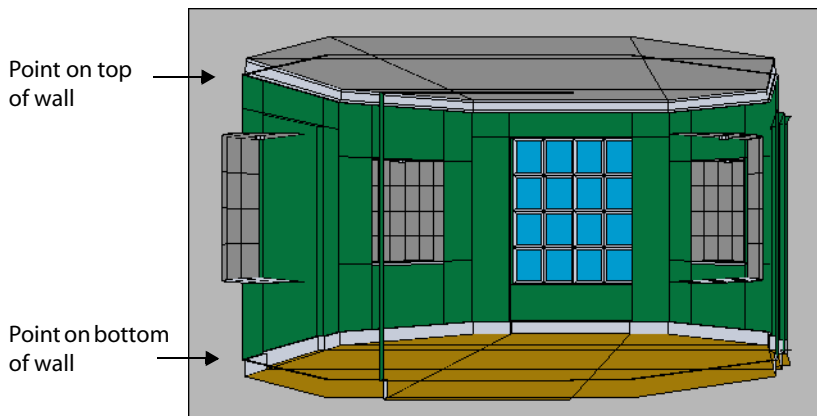
The Measure Distance dialog appears.

5. If necessary, click the Select button so you can pick points.

6. Enable Snap to Nearest Vertex.

7. Pick two points to measure the distance of a wall:

a) Click on a surface near the upper-left corner of the side wall.



b) Click on a surface near the lower-left corner of the same wall.

The dialog shows the X, Y, and Z coordinates for Point 1 and Point 2. The distance between the two points is about 8.5 feet. This indicates that the room is 8.5 feet high, a reasonable size.

Note: If you had selected inches instead of feet when you imported the DXF file, this measurement would show 8.5 inches, which is obviously incorrect. You would have to import the DXF file again using the proper units.

8. Click Close on the Measure Distance dialog.

Stage 2: Preparing Data

In this section, you perform the following tasks:

- Viewing the Model
- Organizing Data into Layers and Blocks
- Working with Luminaires
- Working with Materials

Viewing the Model

To become familiar with the shape of the sunroom, you can view several projections. In Lightscape, there are six orthographic projections and the Perspective projection.

In the Perspective projection, you can use the View Setup feature and dynamic view controls to create the exact view that you want.

View different projections of the model

1. Click the Top button .

The model changes to the plan or Top orthographic projection.

2. Click the Perspective button .

The model changes back to the Perspective projection.

3. Choose View | Projection | Left or click the Left button .

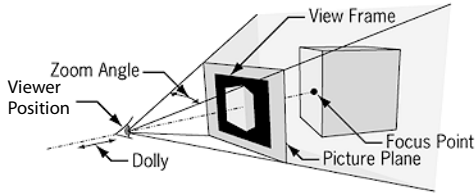
The model changes to the Left orthographic projection.

4. Experiment with the other four orthographic projections: Bottom, Right, Front, and Back.

About the View Setup feature

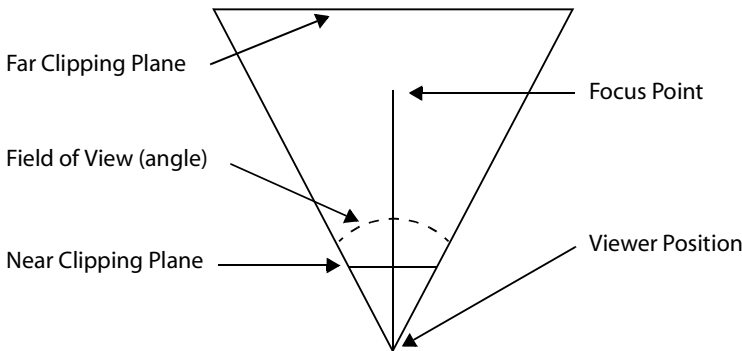
In Lightscape, the Perspective projection uses a viewer position, a focus point, and a picture plane to create views. Both the View Setup feature and the dynamic view controls are built around these conventions.

When you choose View | Setup, Lightscape displays the model in the Top projection with a red triangle. This triangle is known as the view frustum. It describes, in two dimensions, the currently active Perspective projection.



The bottom vertex of the triangle represents the viewer position, or the point where the viewer is standing. (This point is called the camera position in some programs). The opposite end point of the line protruding out from the viewer position is the focus point, or the point the viewer is looking at.

The side of the triangle opposite the viewer position is the far clipping plane. The near clipping plane is the line parallel to the far clipping plane between the viewer position and the far clipping plane. Only the portions of the model that fall within the trapezoid formed by the frustum and the near and far clipping planes are visible in the display.



The view frustum shows five of the settings in the View Setup dialog:

- Viewer Position (except for the Z coordinate)
- Focus Point (except for the Z coordinate)
- Near Clipping Plane
- Far Clipping Plane
- Field of View

Set up an explicit view

1. Choose View | Setup or click the View Setup button .

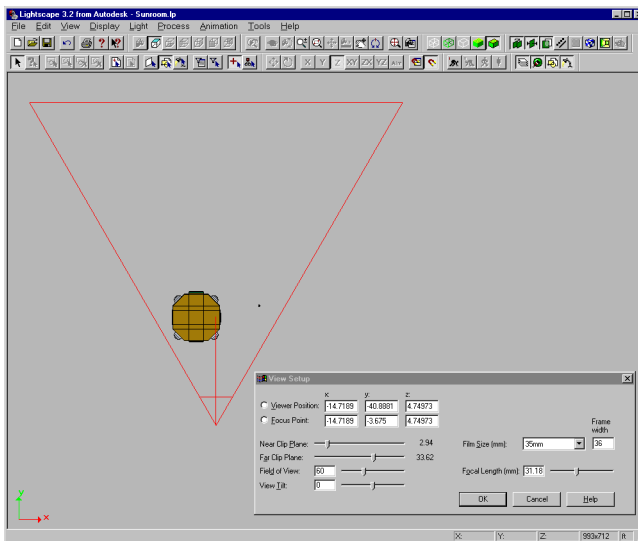
The View Setup dialog appears and the view of the model switches to the Top projection. You can move the dialog out of the way if necessary.

2. Make the Graphic window active by clicking on its title bar.
3. To show the entire view frustrum, use the view controls for the Top projection:

a) With your cursor at the top of the Graphic window, hold down the Z key and drag downward.

Note: Z is the hot key for the Zoom view control.

b) Zoom out until the red triangle is fully visible.



4. Select a new viewer position:
 - a) In the View Setup dialog, enable Viewer Position.

This sets the function of the left mouse button to a special selection mode used for Viewer Position.

b) Pick a point inside the lower-left corner of the model (approximately $X = -26$, $Y = -9$).

The red triangle is redrawn.

5. Select a new focus point:

- a) In the View Setup dialog, enable Focus Point.

This sets the function of the left mouse button to a special selection mode used for Focus Point.

- b) Pick a point near the upper-right corner of the model (approximately $X = -17$, $Y = 2$).

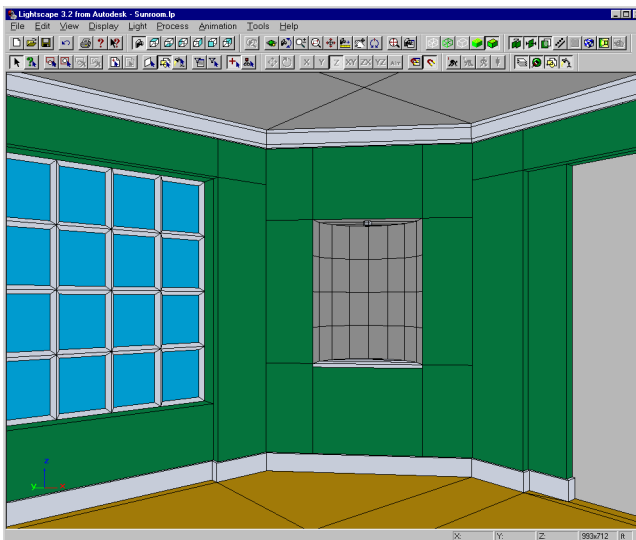
The red triangle is redrawn.

Note: Picking the viewer position and the focus point in the Graphic window only sets the X and Y coordinates of these positions. Since it does not set the height (Z), you must set the heights explicitly by entering Z values.

6. Move the Near Clip Plane slider all the way to the left.

7. Click OK.

Look at the new view of the model.



About the View Controls

Lesson 1 introduced the eight interactive view controls. When you select a view control such as Orbit or Rotate, dragging the mouse in the Graphic window dynamically changes your view of the model.

To set the mouse mode to a view control, press the Shift key with the hot key (or click the toolbar button). In this case, the view control remains in effect until you select another view control or another mouse mode (Select, Query, or a special selection mode).

To set the mouse mode to a view control temporarily, hold the hot key as you drag the mouse. When you release the hot key, the mouse returns to its previous mode.

You can change the Perspective projection with the View Setup dialog or the view controls; both techniques affect the same parameters. An effective way to achieve a specific view is to use both techniques together.

You should learn how the view controls affect the Perspective projection settings in the View Setup dialog:

- When you Orbit the view, you move the viewer position around the stationary focus point.
- When you Rotate the view, you move the focus point around the stationary viewer position.
- When you Zoom the view, the Field of View changes. This is equivalent to using a zoom lens on a camera.
- When you Pan the view, both the focus point and viewer position move up, down, left, and right parallel to the picture plane.
- When you Dolly the view, you move the viewer position in and out along the axis connecting the viewer position and the focus point.
- When you Tilt the view, you rotate about the axis connecting the viewer position and the focus point. This changes the View Tilt, which is measured in degrees.

Experiment with the view controls

1. Click the Orbit button , then orbit the model 45 degrees either way.
2. See how the Perspective projection settings changed:
 - a) Click the View Setup button .
 - Notice how the viewer position moved.
 - b) Click OK.
3. Click the Rotate button , then rotate the model to look at the opposite corner of the room.

4. See how the Perspective projection settings changed:

- a) Click the View Setup button .

Notice how the focus point moved.

- b) Click OK.

5. Try the other view controls and check the changes in the View Setup dialog.

6. Stop when you have a good view of the model.

Save the current view

After you set up a specific view, it can be saved so that you can load it again later. A saved view is a file containing only the information described previously, such as viewer position and focus point. It does not contain information about the geometry seen in the view when you save it. In other words, the view is related to the coordinate system, not the geometry of the specific model that is open.

1. Choose View | Save As.

The Save As dialog appears.

2. If necessary, change the path to *C:\Program Files\Lightscape\Projects\Lesson2*.

3. In the File Name box, type View1.

You do not need to type the .vw file extension; the program adds it automatically.

4. Click Save and confirm the overwrite if necessary.

Open different views

Opening a view file does not change anything contained in the model you are currently using.

1. Choose View | Display Original View to reset the view.

It is easy to restore the view displayed when you first opened the model. The view you had when you last saved the model is the original view the next time you open it.

2. Choose View | Open.

The Open dialog appears.

3. Double-click *View1.vw*.

The view you saved earlier reappears on the screen.

Hint: At the bottom of the View menu, there is a Most Recently Used List. When a view is saved or opened, the name of the view is added to this list of views. Choosing View | View1.vw is the same as opening it from the hard disk.

Working with Display Controls

There are five different modes for displaying the model:

- Wireframe mode displays only the edges of the surfaces in the model.
- Colored Wireframe mode displays the surface edges in their material color.
- Hidden Line mode displays only those edges not blocked by other surfaces closer to the viewer.
- Solid mode displays the surfaces of your model in their material color.
- Outlined mode displays the model in Solid mode with the addition of the surface edges being outlined.

In addition to these modes, there are several controls that affect the display:

- When Double Buffer is on, double buffering is used to eliminate screen redraw artifacts when you interactively view a model.
- When Culling is on, the surfaces that are oriented away from the viewer position are not displayed.
- When Blending is on, OpenGL transparency is used to display transparent surfaces. You can only see a difference if there are transparent surfaces in the model.
- When Antialiasing is on, lines in all modes except Solid are antialiased to appear smoother.
- When Enhanced is on, the Preparation model is displayed using some simple shading.

Try several display modes and controls

1. Choose Display | Hidden Line or click the Hidden Line button .

2. Click the Outlined button .

View the outlined, flat-shaded version of the model. This mode is often more useful than viewing the model in the Wireframe or Hidden Line modes.

3. Click the Enhanced button .

In Enhanced mode, the Preparation model is displayed using simple shading to help differentiate surfaces.

4. Click the Enhanced button again to turn the shading off.
5. Experiment with the other display controls such as Culling, Antialiasing, and Double Buffer. When you are finished, make sure Culling is on.

Change the background color

You can change the colors of items such as the background color, wireframe color, and mesh outline color.

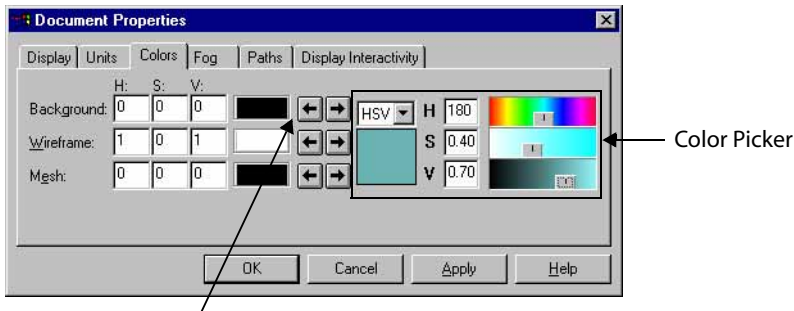
1. Choose File | Properties.

The Document Properties dialog appears.

2. Click the Colors tab.
3. Set color values in the color picker:

Note: On this dialog, you can set a color two ways: enter values directly, or use the color picker and then transfer the color values to the boxes. When you use the color picker, you can use the sliders instead of entering values.

- a) Set H (Hue) to 180.
- b) Set S (Saturation) to 0.40.
- c) Set V (Value) to 0.70.



- d) Click the left arrow button that corresponds with Background to assign the HSV values to the background color.

Hint: You can use the right arrow buttons to transfer color values from the boxes (for Background, Wireframe, or Mesh) to the color picker.

4. Click OK.

Organizing Data into Layers and Blocks

Using layers and blocks are two powerful techniques for organizing data in Lightscape. You can isolate layers or blocks to work on only part of the model at any given time. This can simplify the task of working on large and complex models.

Each object in Lightscape is assigned to a layer. You use layers to manage the large amounts of data that make up a model. For example, you can assign all the surfaces that define the walls of a model to a layer named “walls”.

All the layers that exist in a Lightscape model are listed in the Layers table. In Lightscape, a layer is either on or off. Only objects assigned to layers that are on appear in the Graphic window.

Turn off all layers except the layer named ROOM3_WALLS

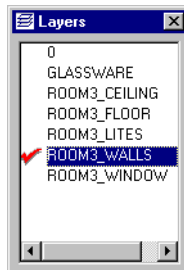
1. Display the model in Outlined mode and view the full extents of the model.

This helps you view the model correctly for the next few exercises.

2. Right-click in the Layers table, then choose All Off.

3. Right-click the ROOM3_WALLS layer in the Layers table, then choose On.

A red check mark to the left of a layer name indicates that the layer is on.



All layers are now turned off except the ROOM3_WALLS layer.

Hint: Double-clicking a layer name toggles the layer on and off.

4. Using the Orbit view control, adjust the view so that you are looking down at the model.

Notice that only the walls are visible. Some of the walls seem to appear and disappear when you orbit because Culling is on.

Turn layers on and off

1. Right-click in the Layers table, then choose All On.

2. Turn off the layer named ROOM3_WALLS:
 - a) Select ROOM3_WALLS in the Layers table.
 - b) Right-click in the Layers table, then choose Toggle.

Hint: You can right-click the layer name to select it and display the context menu in one step.

The walls disappear from the view.

3. To turn on the layer named ROOM3_WALLS, double-click its name in the Layers table.

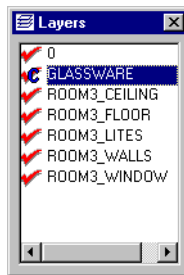
The walls reappear.

Set the GLASSWARE layer to be the current layer and turn off all other layers

Setting a layer to be current makes it the layer with which all new geometry is associated. For example, if you create a new surface in the model, it is associated with and displayed on the current layer.

1. Right-click the GLASSWARE layer in the Layers table, then choose Make Current.

A blue C appears to the left of the GLASSWARE layer, indicating it is the current layer.



2. Right-click in the Layers table, then choose All Off.
3. Double-click GLASSWARE in the Layers table to turn on this layer.

The check mark reappears beside the layer name.

4. Click the View Extents button .

A single vase is now the only object in the Graphic window.

Select a single surface

You can select surfaces one at a time or you can select a group of them. The selection tools are also used for blocks and luminaires.

1. Click the Select button , then click the Surface button .

Now the selection filter only selects surfaces (not blocks and luminaires).

2. Click a surface on the vase.

The edges of the selected surface are highlighted in red.

3. Click another surface on the vase.

The first surface is deselected as the second is selected.

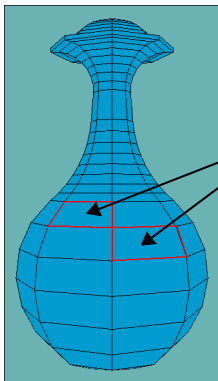
Add other surfaces to the selection

1. Choose Edit | Selection | Accumulate Pick or click the Accumulate Pick button .

With this tool, you can add objects to the current selection set.

Hint: You can also hold down the Ctrl key to add objects to the selection set.

2. Click another surface on the vase.



All the surfaces in the selection set are highlighted in red.

3. Click one of the selected surfaces on the vase.

The red highlight disappears, indicating the surface has been removed from the selection set.

4. Click the Accumulate Pick button again to turn off this feature.
5. Click the Deselect All button .

The current selection set is cleared.

Select a group of surfaces

1. Choose Edit | Selection | Select Area All Vertices or click the Area All button .
2. Drag a selection rectangle to enclose all the surfaces of the vase.

All the surfaces are selected.

Note: When you use Select Area All Vertices, the entire surface (all vertices) must be inside the rectangle for the surface to be selected. When you use Select Area Any Vertex, a surface is selected if any vertex of the surface is inside the rectangle.

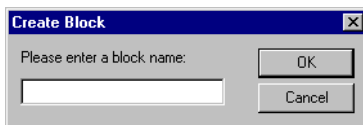
Create a block from the selected surfaces

A block is a group of objects (surfaces and/or other blocks) that has a specific name and an insertion (origin) point. A block can be inserted, or instanced, repeatedly in the model in various positions and orientations. All instances of a block, however, refer to the same geometric description. If you make a change to the geometry or any attribute of a block, every instance of that block in the model inherits the change.

To insert several instances of the vase into the model, you need to group the individual surfaces into a block.

1. Make sure that all surfaces of the vase are still selected.
2. Right-click in the Graphic window, then choose Create Block.

The Create Block dialog appears.



3. Type VASE in the name box, then click OK to create the block.

Notice that a new block named VASE is added to the Blocks table. The existing geometry in the Graphic window has been grouped together to form a new single block object. The block was also given an insertion point that provides a ‘handle’ for moving and rotating instances of the block. By default, the insertion point is always created at the world-coordinate origin (0, 0, 0). The VASE block was created at the origin, so the insertion point is at the bottom center of the block. For more information about insertion points, see “Creating a Luminaire from Geometry” on page 229.

Note: The block instance is assigned to the current layer (GLASSWARE). The surfaces are also assigned to this layer. For details, refer to “Moving Block Surfaces between Layers” on page 226.

Move the VASE block to one of the niches in the model

1. Turn on all the layers in the model.

To do this, right-click in the Layers table, then choose All On.

2. Click the View Extents button  to show the full extents of the model.

3. Orbit to look down on the model.

4. Select the VASE block:

a) If necessary, click the Deselect All button .

b) Click the Select button , then click the Block button .

Now the selection filter only selects blocks.

c) Click a surface on the vase.

The entire vase is highlighted in red, indicating the block is selected.

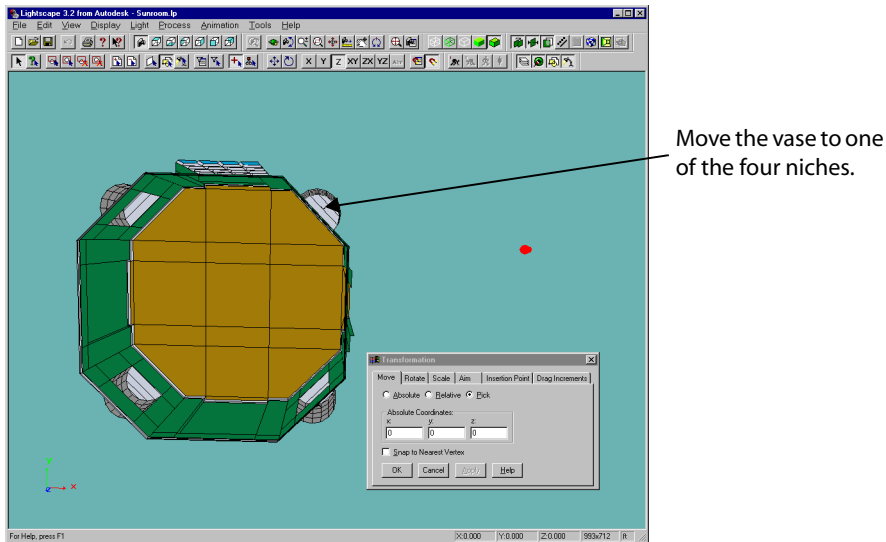
5. Choose Edit | Transformation or right-click in the Graphic window and choose Transformation.

The Transformation dialog appears.

6. On the Move panel, enable Pick.

This sets the function of the left mouse button to a special selection mode used for Pick.

- Click in the center of the shelf within one of the four niches in the model. (You may need to modify your view first.)



The vase moves to the new location. You can pick again to move to another location.

- Click Cancel to close the Transformation dialog.

Hint: You can also leave the dialog open because you will use it in the next few exercises.

Duplicate the VASE block and move the duplicate instance

- Right-click in the Graphic window, then choose Duplicate.

This creates a duplicate instance of the vase in exactly the same location as the first vase. You can now move this instance.

- Switch to the Top projection.

If necessary, zoom in or click the View Extents button.

3. Move the instance using interactive Move:

- a) Make sure that the Use Drag Increments button  is off.

You disable this feature just in case drag increments were set for interactive Move.

- b) Click the Move button  to switch to interactive Move mode.

- c) Click the XY Constraint button  to move along the XY plane.

- d) Drag in the Graphic window until the vase is in the center of one of the other niches.

Hint: You can undo an interactive Move before you release the left mouse button by right-clicking.

- e) Click the Deselect All button .

Add a block instance from the Blocks table using drag and drop

One way to create an instance of a block is to select the block name in the Blocks table, right-click and choose Create Single Instance. The new instance is placed at the origin (0, 0, 0), so you need to move it to another position. A fast way to perform these steps is to drag the instance from the Blocks table directly to the new position.

1. Click VASE in the Blocks table and drag the mouse into the Graphic window.

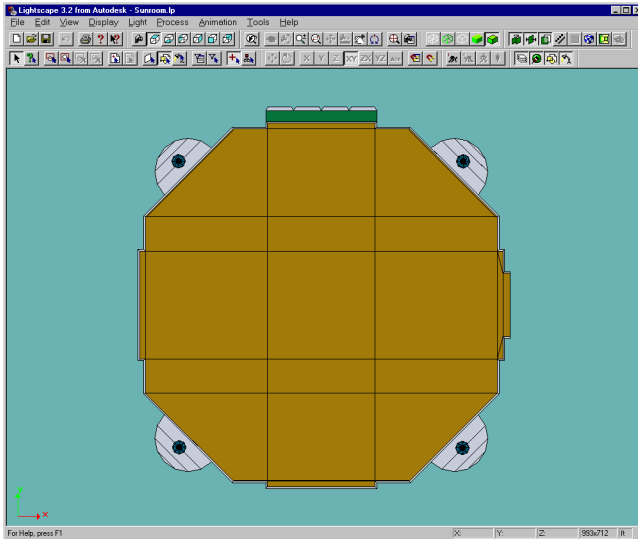
As you drag, the mouse cursor is a blue arrow with a block symbol.

2. Move the cursor over one of the empty niches and release the mouse.

An instance of the VASE block is added at that location.

Hint: If you use the Accumulate Pick feature when dragging an instance from the Blocks table, the new instance will be automatically selected. Before dragging the instance out, make sure that nothing else is selected. To clear the selection set, click the Deselect All button.

3. Repeat steps 1 and 2 to place a vase in the remaining niche.



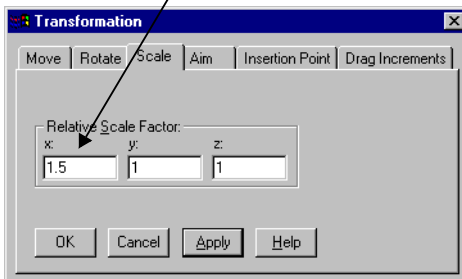
Now there is a vase
in each niche.

If necessary, use the interactive Move feature to position the new vases more precisely.

Scale and rotate the blocks

1. Click the Deselect All button .
2. Select one of the VASE blocks.
3. Change the scale of the selected vase:
 - a) Right-click in the Graphic window and choose Transformation.
 - b) Click the Scale tab.

- c) Change the Relative Scale Factor X to 1.5, then click Apply.



Note: Each time you click Apply, objects are scaled relative to their current size (not relative to their original size). Rotations, on the other hand, are relative to their original angle.

The vase is now oval in shape.

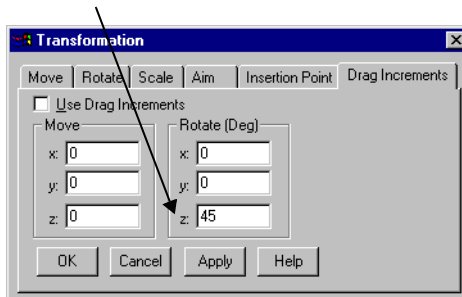
4. Set the drag increments to help you use the interactive Rotate feature:

With the interactive Move and Rotate features, you can transform a block without using the Transformation dialog.

- a) Click the Drag Increments tab.

Hint: When the Transformation dialog is closed, you can click the Edit Drag Increments button  to quickly display the Drag Increments panel.

- b) Type 45 in the Rotate Z box.



- c) Enable Use Drag Increments.

Hint: You can also toggle this option by clicking the Use Drag Increments button .

- d) Click Apply.

5. Rotate the selected vase:

- a) Click the Rotate button  to switch to interactive Rotate mode.
- b) Click the Z Constraint button  to rotate around the Z axis.
- c) Drag in the Graphic window until the longer side faces into the room.

Depending on which vase you selected, the rotation around the Z axis might be 45 degrees or 135 degrees.

6. Vary the scale and rotation of the three remaining vases:

- a) Select one of the block instances.
- b) Scale it as explained in step 3.

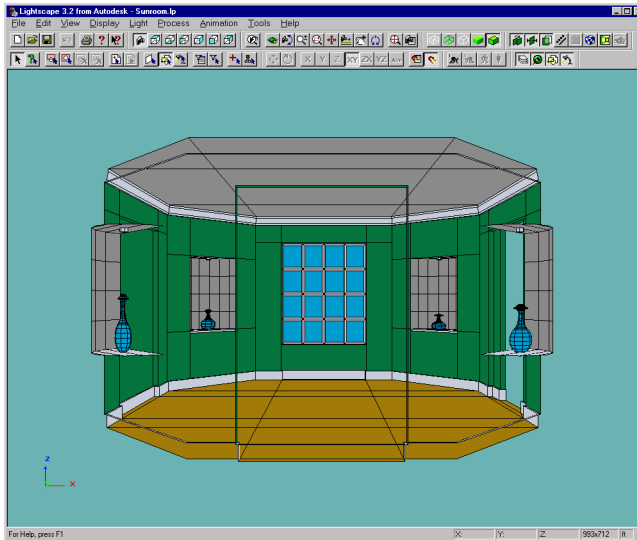
Experiment with different values for the Relative Scale Factor X, Y, and Z boxes.

- c) Rotate the instance using interactive Rotate or the Rotate panel of the Transformation dialog.
- d) Click the Deselect All button .

7. Click Cancel to close the Transformation dialog.**8. Click the Perspective button .**

9. Click the View Extents button .

The following illustration shows the view with the four vases.



Save your work

1. Choose File | Save As.

The Save As dialog appears.

2. Type Sunroom.lp in the File Name box.

If necessary, change the path to *C:\Program Files\Lightscape\Projects\Lesson2*.

3. Click Save.

If necessary, click Yes to confirm the overwrite.

Working with Luminaires

A luminaire is the basic light source; it represents both the physical appearance and the photometric characteristics of a lighting fixture—that is, it includes the geometry of the fixture, the color intensity, and the directional distribution of light energy emitted from the fixture.

A luminaire is a block that has photometric attributes associated with it. It is made by associating photometric data with an existing block definition. When you create a luminaire, the block is removed from the Blocks table and placed in the Luminaires table. All instances of the block are changed into luminaires.

You can set the following photometric properties of a luminaire:

- Light Source Type: Point Light, Linear Light, or Area Light
- Lamp Color Filter
- Intensity Magnitude: Luminous Intensity, Luminous Flux, or Illuminance at a Distance
- Intensity Distribution: Isotropic, Spot, Diffuse, or Photometric Web

In this section, you convert an existing block into a luminaire and add a predefined luminaire from a block library.

Note: An exercise in Lesson 6 goes through the entire process of creating a luminaire from a geometry. You prepare and import a geometry file, define the block as a luminaire, and then set the photometric properties. For details, refer to “Creating a Luminaire from Geometry” on page 229.

Define the LITE_NARROW1 block as a luminaire

The luminaire in this exercise is set up as a spotlight (a directional point light). The LITE_NARROW1 will contain a R-40 150 watt wide beam bulb. According to the manufacturer, this bulb is a 5400 cd point light with a Spot distribution (beam angle is 22, field angle is 50). The lamp color specification is halogen, and the HSV values for the color filter are 0, 0, and 1.

1. Right-click LITE_NARROW1 in the Blocks table, then choose Query Instances to see where the block instances are.

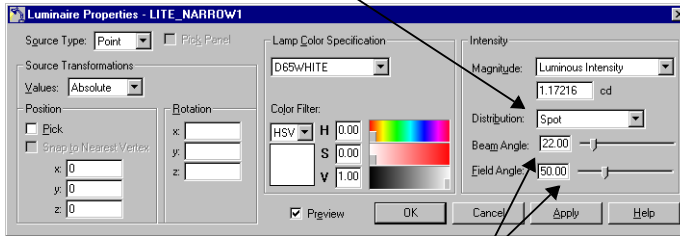
The queried blocks are highlighted in green.

2. Right-click the block name in the Blocks table, then choose Define as Luminaire.
3. Click Yes to confirm that you want to continue this action.

The Luminaire Properties dialog appears and the view shows the isolated luminaire.

4. Define the luminaire using the manufacturer's specifications:

- a) Select Point from the Source Type list.
- b) Select Spot from the Distribution list.



Note: It is important to set the distribution before the intensity magnitude.

- c) In the Intensity group box, set the Field Angle slider to 50 and the Beam Angle slider to 22.

Note: It is important to set the field angle before setting the beam angle because the field angle acts as a maximum constraint for the beam angle.

- d) Type 5400 in the Magnitude box. (Use Luminous Intensity.)
- e) Select Halogen from the Lamp Color Specification list.

The Hue (H), Saturation (S), and Value (V) of the color filter should be set to 0, 0, and 1, respectively.

5. Click OK and confirm the overwrite.

The Graphic window shows the full model. LITE_NARROW1 is removed from the Blocks table and added to the Luminaires table. All instances of LITE_NARROW1 are now defined as luminaires.

About the Lightscape Libraries

A set of Lightscape libraries is included with the Lightscape application. The libraries include hundreds of blocks, luminaires, and materials that you can easily add to your models.

Note: Depending on your Lightscape installation, the libraries may already be installed in the *C:\Program Files\Lightscape\Lib* folder. Otherwise, perform a Custom installation and select only Lightscape Libraries from the list of items to install.

You can find a pictorial index of all the library contents in the Online Help for the Lightscape libraries. To load the luminaire, you need to know the name of the luminaire and the library where it is stored.

The luminaires in the libraries are predefined for use in Lightscape and the photometric properties are already set accurately. After you load a luminaire into Lightscape, you can simply drag and drop it into the model.

Add ERCO luminaires to the sunroom

One of the lighting manufacturers featured in the Lightscape libraries is ERCO Leuchten GmbH. One of the luminaires ERCO manufactures is the Trion 33487 uplight. In this exercise, you load this luminaire from a library and add four instances of it. You need to know the name of the luminaire and the library where it is stored. In the *Lesson2* folder, the *ERCO-Luminaire-33487.blk* library includes the ERCO-33487 luminaire.

Note: You can also load this luminaire from the Lightscape libraries if they have been installed.

1. Click the View Extents button .
2. Load the ERCO-33487 luminaire from the *ERCO-Luminaire-33487.blk* library:

a) Right-click in the Luminaires table, then choose Load.

The Open dialog appears.

b) If necessary, change the path to *C:\Program Files\Lightscape\Projects\Lesson2*.

c) Double-click *ERCO-Luminaire-33487.blk*.

The Available Luminaires dialog appears. You can use the block preview to examine the luminaire.

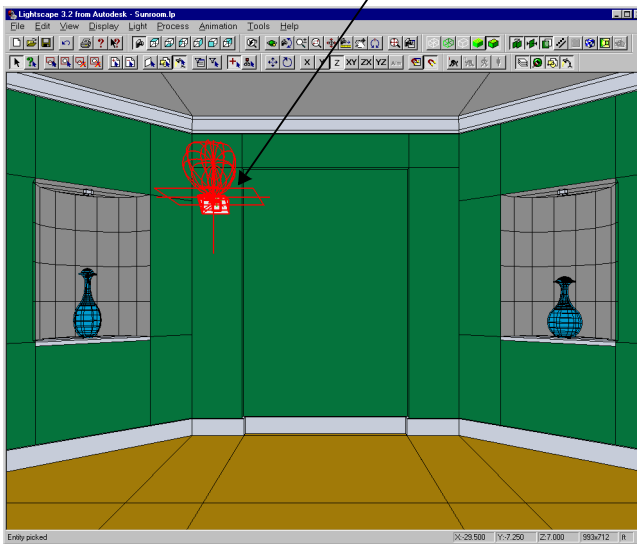
d) Select ERCO-33487 and click OK.

The luminaire is added to the Luminaires table, a layer with the same name is added to the Layers table, and materials used for the block surfaces are added to the Materials table.

Note: To view the photometric properties of this luminaire, double-click ERCO-33487 in the Luminaires table.

3. Right-click ERCO-33487 in the Layers table, then choose Make Current.
4. Open the view named *Wall.vw*.
5. Click the Accumulate Pick button  to turn on this feature.

6. Drag an instance of the ERCO-33487 from the Luminaires table to the wall.



As you drag, the mouse cursor is a blue arrow with a luminaire symbol.

7. Set the exact position of the luminaire:
- Right-click in the Graphic window, then choose Transformation.
 - Click the Move tab.
 - Enable Absolute and enter these coordinates: $X = -29.5$, $Y = -7.25$, $Z = 7$.
 - Click Apply.
8. Duplicate the luminaire and move the new luminaire instance:
- Right-click in the Graphic window, then choose Duplicate.
 - On the Move panel, enter these coordinates: $X = -29.5$, $Y = -0.75$, $Z = 7$.
 - Click Apply.

9. Duplicate the luminaire and move the new luminaire instance:

- a) Right-click in the Graphic window, then choose Duplicate.
- b) On the Move panel, enter these coordinates: $X = -13.5$, $Y = -0.75$, $Z = 7$.
- c) Click Apply.
- d) Orbit the model to see the opposite wall.

10. Rotate the selected luminaire to face into the sunroom:

- a) On the Rotate panel, enable Relative.
- b) Type 180 in the Offset Z box.
- c) Click Apply.

11. Duplicate the luminaire and move the new luminaire instance:

- a) Right-click in the Graphic window, then choose Duplicate.
- b) On the Move panel, enter these coordinates: $X = -13.5$, $Y = -7.25$, $Z = 7$.
- c) Click OK.

The new luminaires are ready for processing.

12. Click the View Extents button .

13. View all the luminaires:

- a) Click the Select button , then click the Luminaire button .

Now the selection filter only selects luminaires.

- b) Choose Edit | Selection | Select All or click the Select All button .

Notice that four of the light distributions point up and four point down.

- c) Click the Deselect All button .

14. To save the changes to the *Sunroom.lp* Preparation file, choose File | Save.

Working with Materials

A material defines the appearance of a surface. Defining a material in Lightscape requires that you set both the specular and diffuse behavior of the material.

You set the specular behavior of a material with the Refractive Index, Shininess, and Transparency sliders:

- The Refractive Index slider controls the amount of light reflected from a material. Typical values range between 1.0 (no specular reflection) and 2.5 (the Refractive Index for diamond). More light is reflected as the refractive index increases.
- The Shininess slider controls the character (how shiny or dull the reflection). If the surface is perfectly shiny, reflected images appear as they would in a mirror. A less shiny surface scatters the light in many directions and reflected images are more indistinct and appear as highlights.
- The Transparency slider controls how much of the incident energy that is not reflected from the surface passes through the material. As it passes through the material, the energy is modulated by the reflective color of the material.

You set the diffuse behavior with the reflective color for the material. This behavior includes the amount and character of the absorption and diffuse reflection. Generally, you change the reflective color by setting the HSV values:

- Hue controls the color of the absorbed and reflected light. For example, a blue material reflects more energy in the blue wavelengths and thus appears blue in color, while a red material reflects more energy in the red wavelengths and appears red in color.
- Saturation controls the degree of colorization of the reflected and absorbed light. Increasing the saturation deepens the color of the material and increases the amount that reflected light is colored by the surface.
- Value controls the amount of light reflected diffusely versus the amount of light absorbed. A value of 0.7 indicates that 70% of the incident energy is being reflected and 30% is being absorbed. Increasing the value increases the amount of light reflected by the material. It is almost always set lower than 0.8.

Note: You can also set the reflective color using a texture map.

To obtain a valid radiosity solution, it is important that the reflectance of the material be set to within the recommended range of values for the specific type of material being defined. Reflectance, by default, is set either by the Brightness value of a texture map (if one is used) or by the Value of the color (if a texture map is not used). A green line on these parameters displays the recommended range of values. In addition, if you are outside the recommended range, the average reflectance of the material (displayed at the bottom of the dialog) appears in red.

Change the name and properties of the material assigned to the walls

In this exercise, you rename the materials from the AutoCAD color index numbers to names. In general, you should give your materials names that are meaningful.

1. Adjust the view so that you are “standing” inside the room.

To do this, open the view named *View1.vw* (if it shows the inside of the room) or create a new view.

2. Click the Outlined button .

3. Choose Edit | Selection | Query or click the Query Select button .

This sets the function of the left mouse button to Query mode. Clicking an object in the Graphic window displays information about the object on the status bar.

4. Click the Surface button .

5. Pick a point on a wall.

The wall is highlighted in green and material 115 is highlighted in the Materials table.

6. Rename material 115:

- a) Right-click the material name in the Materials table, then choose Rename.

A blinking cursor appears at the end of item 115.

Note: You must right-click the material name to rename the selected material. If you right-click elsewhere in the Materials table, material 115, for example, will not be renamed.

- b) Type PAINT_FLAT_DARKGREEN and press Enter.

7. Edit the properties of the wall material:

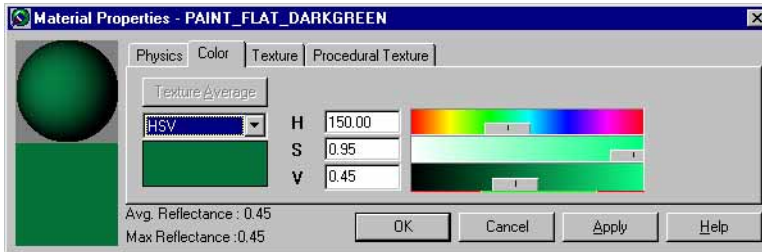
- a) Double-click PAINT_FLAT_DARKGREEN in the Materials table.

The Material Properties dialog appears.

- b) On the Physics panel, select Paint Flat from the Template list.

The sliders are marked with red and green lines. The green portions of these lines represent the valid range of values that you should use for this type of material.

c) Click the Color tab.



Look at the current values in the boxes. The Saturation is set to 0.95, indicating that light reflected from the surface will be extremely green, changing the color of the rest of the room.

d) Set S (Saturation) to 0.60.

The V (Value) box is set to 0.45, indicating that 55 percent of the incident energy will be absorbed and 45 percent reflected.

e) Set V (Value) to 0.40.

Since there is no texture map for this material, the Avg. Reflectance and Max Reflectance are both equal to the Value. These reflectance values are displayed at the bottom of the dialog.

f) Click OK.

Notice the slight change of color on all the walls.

Change the name and properties of the material assigned to the trim

1. The left mouse button should still be in Query mode. Pick a point on the trim.

The trim is highlighted in green and material 7 is highlighted in the Materials table.

2. Rename material 7:

a) Right-click the material name in the Materials table, then choose Rename.

A blinking cursor appears at the end of item 7.

b) Type PAINT_OIL_WHITE and press Enter.

3. Edit the properties of the trim material:

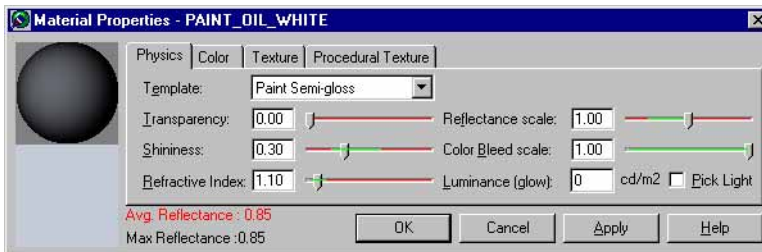
- a) Double-click PAINT_OIL_WHITE in the Materials table.

The Material Properties dialog appears.

- b) Click the Physics tab.

- c) Select Paint Semi-gloss from the Template list.

The sliders are marked with red and green lines and the Shininess and Refractive Index are changed.



- d) Set the Shininess slider to 0.50 and the Refractive Index slider to 1.20.

This makes the material slightly specular (shiny) and makes the bounced light appear as highlights rather than reflections.

- e) Click the Color tab.

The Value is set to 0.85, indicating that only 15 percent of the incident energy will be absorbed and 85 percent reflected.

- f) Set V (Value) to 0.75.

Do not change the Hue or Saturation.

- g) Click OK.

Change the name and properties of the material assigned to the floor

1. Pick a point on the floor.

The surface is highlighted in green and material 43 is highlighted in the Materials table.

2. Rename material 43:

a) Right-click the material name in the Materials table, then choose Rename.

A blinking cursor appears at the end of item 43.

b) Type WOOD_WHITEASH and press Enter.

3. Edit the properties of the floor material:

a) Double-click WOOD_WHITEASH in the Materials table.

The Material Properties dialog appears.

b) On the Physics panel, select Wood Varnished from the Template list.

c) Set the Shininess slider to 0.70 and the Refractive Index slider to 1.20.

This makes the material slightly specular (shiny) and makes the bounced light appear as highlights with some reflective effect.

d) Click the Color tab.

The Saturation is set to 0.95, indicating that light reflected from the surface will be extremely yellow, changing the color of the rest of the room.

e) Set S (Saturation) to 0.25 and set V (value) to 0.50.

f) Click OK.

Change the name and properties of the material assigned to the ceiling

1. Pick a point on a ceiling.

You may need to change your view to do this. The surface is highlighted in green and material 253 is highlighted in the Materials table.

2. Rename material 253:

- a) Right-click the material name in the Materials table, then choose Rename.

A blinking cursor appears at the end of item 253.

- b) Type `PAINT_FLAT_WHITE` and press Enter.

3. Edit the properties of the ceiling material:

- a) Double-click `PAINT_FLAT_WHITE` in the Materials table.

The Material Properties dialog appears.

- b) On the Physics panel, select Paint Flat from the Template list.

- c) Click the Color tab.

- d) Set V (Value) to 0.70.

- e) Click OK.

Change the properties of the material assigned to the vases and windows

1. Pick a point on one of the vases.

The surface is highlighted in green and material 140 is highlighted in the Materials table.

2. Rename material 140:

- a) Right-click the material name in the Materials table, then choose Rename.

A blinking cursor appears at the end of item 140.

- b) Type `GLASS_BLUE` and press Enter.

3. Edit the properties of the vase material:

- a)** Double-click GLASS_BLUE in the Materials table.

The Material Properties dialog appears.

- b)** On the Physics panel, select Glass from the Template list.

The template sets the specular sliders to appropriate values for a particular class of materials. In this case, the material becomes quite specular (shiny) and the bounced light appears as reflections rather than highlights.

- c)** Click the Color tab.

Look at the current values in the boxes. The Saturation and Value are both set extremely high.

- d)** Set S (Saturation) to 0.30.

- e)** Set V (Value) to 0.80.

For transparent materials like glass, Value controls what is generally called the transmissivity of the material, or the amount of energy absorbed as the light passes through the body of the material.

- f)** Click OK.

Change the name and properties of the material assigned to the spotlights

1. Pick a point on one of the LITE_NARROW1 luminaires.

The fixture is highlighted in green and material 9 is highlighted in the Materials table.

2. Rename material 9:

- a)** Right-click the material name in the Materials table, then choose Rename.

A blinking cursor appears at the end of item 9.

- b)** Type PAINT_METAL_WHITE and press Enter.

3. Edit the properties of the fixture material:

- a)** Double-click PAINT_METAL_WHITE in the Materials table.

The Material Properties dialog appears.

- b)** On the Physics panel, select Metal from the Template list.

This disables the Refractive Index and Transparency sliders. The Refractive Index of metals is approximated using the color of the material, and metals cannot be transparent.

- c)** Set the Shininess slider to 0.50.

This makes the bounced light appear as weak highlights rather than reflections.

- d)** Click the Color tab.

Look at the current values in the boxes. The Value is set to 0.85, indicating that only 15 percent of the incident energy is absorbed and 85 percent reflected. As metals tend to be more highly reflective than other classes of material, do not change the Value.

- e)** Click OK.

4. To save the changes to the *Sunroom.lp* Preparation file, choose File | Save.

You have finished the Preparation stage and are about to enter the Solution stage. The model you are currently working on is going to be radically transformed during the initiation process. It is a good idea to save your work at this time.

Stage 3: Creating the Radiosity Solution

In this section, you learn about:

- Processing Parameters

You perform the following tasks:

- Setting the Processing Parameters
- Initiating the Preparation Model
- Checking the Solution
- Processing the Solution

Processing Parameters

During the radiosity process, the surfaces of your model are subdivided into a mesh of energy sample points. The number of mesh elements required to capture the illumination of a surface depends upon the complexity of the illumination on that surface. The greater the lighting detail, the greater the number of mesh elements that are needed to capture it accurately.

Adaptive Subdivision

To represent variations of illumination across a surface, the surface is automatically broken down into smaller pieces (elements). The radiosity process then computes the illumination from a light source to each of the corners (vertices) of each element. To maintain as efficient a solution as possible, the system starts with a coarse mesh. It automatically refines the mesh locally where large differences in light level are detected. For more information on adaptive subdivision, see “Global Processing Parameters” on page 124.

Initiation

Initiation is the process that simplifies the geometry, materials, and lights, and produces a model that is ready for the radiosity calculations.

Processing

After initiation, your model is ready for the radiosity lighting simulation, which you start by choosing Process | Go. The simulation is computed in successive iterations.

At each iteration, the system selects the brightest light source and computes its contribution to all the surfaces in the scene. Once all the primary light sources are accounted for, the system starts computing the interreflections of light between surfaces. At each iteration, the system selects the brightest reflecting surface and computes its contribution to the rest of the environment.

This process is called progressive refinement because, at each iteration, the radiosity solution is refined—that is, it is a better approximation of the final result.

Setting the Processing Parameters

Set the processing parameters

1. Choose Process | Parameters.

The Process Parameters dialog appears.

The numerical values in the various boxes of the dialog control the quality and speed of the Lightscape radiosity process. Although the values can be set individually, you use a wizard in this exercise to set them automatically.

2. Click the Wizard button to start the Process Parameters Wizard.
3. Select 2 for a moderate quality level, then click Next to show the next page of the wizard.
4. Select No so daylight is not considered in this model.
5. Click Finish to update the processing parameters based on the selections made in the wizard.

Notice the change in the numerical values in the various boxes of the dialog.

6. Click OK to close the Process Parameters dialog.

Initiating the Preparation Model

Begin the Solution stage

1. Click the Solid button .
2. Click the Initiate button .

When the warning appears, click Yes to save changes.

This updates the *Sunroom.lp* model with the latest changes and causes the program to create the Solution model. Notice that the filename changes to *Sunroom.ls* when the initiation process completes. The model is dark because no light is distributed to the scene.

Processing the Solution

Start the radiosity processing

1. Click the Go button .

Lightscape calculates the light distribution in the given environment. The green highlight on the screen indicates which light source is currently being distributed.

Notice the various statistics shown on the status bar such as the number of iterations and percentage of the total energy distributed.

2. When the percentage of energy shot exceeds 75%, click the Stop button .

The processing stops after the current iteration finishes. This may take a while (depending on the speed of your computer).

Change the brightness and contrast settings

Because monitors are calibrated differently, Lightscape lets you adjust brightness and contrast. These settings help to fine-tune the way you perceive the solution.

Note: These changes can only be seen in the Solution model.

1. Choose File | Properties.

The Document Properties dialog appears.

2. On the Display panel, set Brightness to 65.

Note: The brightness of the image does not affect the lighting calculations.

3. Set Contrast to 50.

4. Click OK.

Change the display mode to see the radiosity mesh

Lightscape gives you complete control over the quality of a solution. A quick design study does not require the same degree of quality as a final production graphic. The next few exercises compare the various processing parameter settings to demonstrate the differences in processing speed and image quality.

1. Click the Outlined button .

Notice how the mesh was formed.

2. Click the Solid button .

3. Adjust the view to get a feel for the interactive display speed.

For example, orbit around the focus point or rotate around the viewer position.

4. Before you reset the solution, note the quality level of the present solution:

a) Choose Process | Statistics.

The number of mesh elements in your model is shown on the status bar. Write this number on a piece of paper. (You should have approximately 12,700 mesh elements.)

b) Notice how the light is distributed on the surfaces.

After you re-process the solution, you will look at the light distribution again.

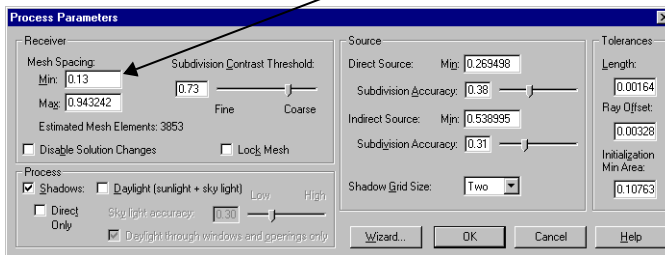
Reset the solution, increase meshing, and start processing again

1. Choose Process | Reset or click the Reset button .

When a warning appears, click Yes to continue resetting the radiosity solution.

2. Choose Process | Parameters.

3. In the Receiver group box, set Mesh Spacing Min. to 0.13.



This will halve the size of the smallest mesh element, increasing the quality of the solution, but also increasing the amount of meshing.

4. Click OK.

5. Click the Ambient button .

Ambient approximation takes any 'unshot' energy and distributes this evenly over all the surfaces of the model. This lets you see the model during the early stages of the solution.

6. Click the Go button .

Notice the difference in the length of time it takes to process the solution. The processing time increases proportionally to the increase in mesh density.

7. When the percentage of energy shot exceeds 85%, click the Stop button .

It is likely that the solution took a lot longer to process and looks only a little better. Notice that the boundaries between the lighter and darker areas look better. This is because more mesh elements were used to represent a smoother transition.

8. Click the Ambient button again to turn off this feature.

Change the display mode to see the radiosity mesh

1. Click the Outlined button .

Notice where the geometry was meshed more finely.

2. Choose Process | Statistics.

Note the number of mesh elements in the model. (It should be approximately 18,300.)

3. Click the Solid button .

Increase the display speed of the solution

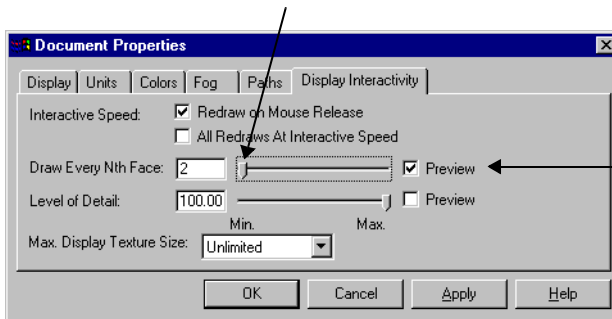
The radiosity mesh adds considerable additional geometry to the solution, slowing down the interactive display speed.

In Lesson 1, you learned that one way to increase the interactive display speed is to decrease the Level of Detail control. With this technique, only part of the mesh information for each original polygon is displayed. Where surfaces are far from you in the model, only the top level lighting data is displayed. Although the surfaces appear more crude, they are displayed faster.

The Draw Every Nth Face control is another way to increase the speed and smoothness of redraws when you are moving through the model. By increasing this control, only a percentage of the polygons in the model is displayed. For example, setting this control to 2 draws every second polygon, thus effectively doubling the display speed.

These settings only affect the display during interactive movement; the screen will redraw in full display mode once you release the mouse button.

1. Click the View Extents button .
2. Choose File | Properties.
3. Click the Display Interactivity tab.
4. Draw every second polygon:
 - a) Set the Draw Every Nth Face slider to 2, then click Apply.



Hint: You can enable the corresponding Preview check box to see the result without applying the change.

- b) Notice the increased smoothness and speed of the display as you orbit the model.
 - c) Reset the Draw Every Nth Face slider to 1.
5. Try a lower level of detail:
 - a) Set the Level of Detail slider to approximately 50, then click Apply.
 - b) Orbit the model again.

Notice that the lighting detail from the uplights is now displayed much more crudely.

 - c) Reset the Level of Detail slider to 100, then click OK.

Stage 4: Fine-Tuning the Solution

In this section, you learn about:

- Using Natural Lighting

You perform the following tasks:

- Setting Up a Window
- Turning Off Lights

Using Natural Lighting

For a more efficient and accurate interior daylight calculation, you should indicate which surfaces act as windows or openings (for light to enter through). When you define a surface as a window, the properties of the material assigned to the window affect the amount and color of the light that enters the space. The window surface's material should, therefore, be transparent.

When you define a surface as an opening, light enters through it as if there is no surface. In other words, the opening's surface is ignored and not displayed. It acts only as a placeholder.

You can modify the following sun and daylight parameters:

- Direction of north
- City
- Latitude and longitude
- Time zone
- Day of the year
- Time of day
- Daylight savings time on/off
- Sun color
- Sky color
- Sky conditions

Setting Up a Window

Define the window surfaces as windows

1. Click the View Extents button .
2. Select all the window surfaces:
 - a) Click the Select button , then click the Surface button .
 - b) Click the Accumulate Pick button  to turn on this feature.
 - c) Select the 16 window pane surfaces.

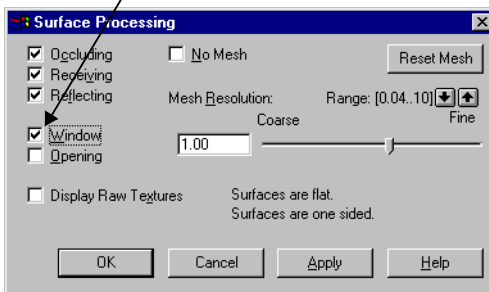
Note: You could use Select Area Any or Select Area All if you first add GLASS_BLUE to the selection filter. After you turn Use Selection Filter on, you could select the glass surfaces without selecting the window panes.

A red highlight appears as you select each of them.

3. Right-click in the Graphic window, then choose Surface Processing.

The Surface Processing dialog appears.

4. Enable Window, then click OK.



The surfaces are now defined as windows.

5. Click the Deselect All button  to clear the selection set.

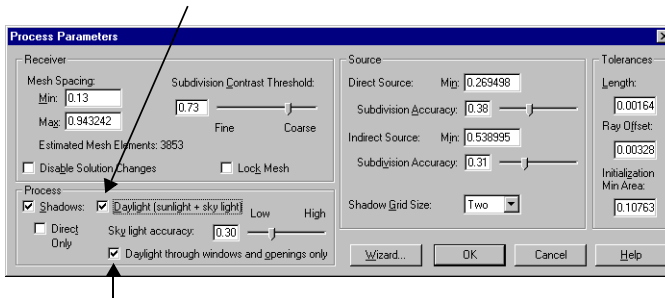
Turn on Daylight and then set the daylight parameters

1. Choose Process | Parameters.

The Process Parameters dialog appears.

2. Turn on the Daylight processing parameters:

a) In the Process group box, enable “Daylight (sunlight + sky light)”.



b) Enable “Daylight through windows and openings only”.

If there are no external elements being illuminated in your solution, you should enable this option for a more efficient daylight calculation.

c) Set the Sky light Accuracy slider to 0.30.

d) Click OK.

3. Click the Top button .

The model changes to the Top projection. This projection helps you set the North pointer in the next step.

4. Choose Light | Daylight.

The Daylight Setup dialog appears.

5. Set the daylight parameters:

- a)** On the Sun and Sky panel, set the Sky Conditions slider to medium.

You will use the default Sun and Sky colors.

- b)** Click the Place tab.

- c)** Select New York from the Location list.

The latitude and longitude are automatically set.

- d)** Set the Direction of North to 90 degrees.

To do this, type 90 in the North box or adjust the North pointer. The direction of the North pointer is relative to the plan view of the model.

- e)** Click the Time tab.

The Time Zone is set automatically because the location (on the Place panel) was chosen from the list of predefined cities.

Hint: If you enable Direct Control on the Daylight Setup dialog, the Place and Time panels are replaced with the Direct Control panel. Having direct control over the sun is useful when you want sun shadows in specific places. For example, it is useful for game development.

- f)** Set the date to June 21 and the time to 5:00 p.m.

- g)** To use daylight savings time, enable Daylight Savings.

- h)** Click OK.

6. Click the Perspective button .

Process the solution for 25 more iterations

1. Click the Go button .

As the processing continues, the direct energy from the sun will be distributed first. Next, the diffuse skylight component will be calculated for each window pane.

2. Click the Stop button  after about 25 more iterations (or until the solution is about 75% complete).

Note: The number of iterations processed appears on the status bar in parentheses.

The model appears quite bright because the display settings are currently optimized for a model without daylight. This is like the situation of having a real camera adjusted for an interior exposure and then using the same settings outside. You would need to adjust your camera settings to accommodate the brighter light.

3. Adjust the brightness:

- a) Choose File | Properties.

The Document Properties dialog appears.

- b) On the Display panel, change the Brightness to 50 and click Apply.

- c) Click OK.

Turning Off Lights

Although you cannot add or move lights in the Solution stage, you can change or delete them. With the sunlight added to the scene, you can turn off the four ERCO luminaires. Once turned off, the solution must run for several iterations to remove the light energy that was emitted from the modified light sources.

Turn off the ERCO luminaires

1. Click the View Extents button .

A view from the front is displayed because the Perspective projection is selected. This view shows the sunlight falling on the floor.

2. Right-click ERCO-33487 in the Luminaires table, then choose Photometrics.

The Luminaire Properties dialog appears.

Hint: You can also double-click a luminaire in the Luminaires table to display this dialog.

3. In the Intensity group box, change the intensity to 0 cd.
4. Click OK and confirm the overwrite of the existing luminaires.

Process the solution for a few more iterations

1. Click the Go button .

As the processing continues, the energy from the four lights is subtracted, creating a dark patch on the walls and ceiling.

2. Click the Stop button  after about ten iterations.

Stage 5: Outputting the Results

In this section, you learn about:

- Rendering

You perform the following tasks:

- Performing Lighting Analysis
- Creating a Single High-Quality Image Using Ray Tracing

Performing Lighting Analysis

The radiosity solution contains all the information necessary for performing lighting analysis:

- You can visualize the distribution of light over the surfaces in the environment by using pseudo-coloring techniques.
- A grid of illumination values can be superimposed over selected surfaces.
- Statistical data such as averages, minima and maxima, and criterion ratings are presented interactively for any surface in the scene on the Statistics panel in the Lighting Analysis dialog.

Check the lighting of the solution

1. Change the lighting units to American:
 - a) Choose File | Properties.
 - b) Click the Units tab.
 - c) Select American from the Lighting list, then click OK.
2. Choose Light | Analysis.
The Lighting Analysis dialog appears.
3. Click the Display tab.
4. View the model in pseudo-color mode:
 - a) Select Color from the Display list.
 - b) Set Scale Max to 185 fc, then click Apply.

5. Click the Statistics tab, then pick a point on a wall.

Notice the values that appear on the Statistics panel.

6. View the model using a grid:

- a) Show the Top projection.
- b) Click the Grid tab.
- c) Enable Grid.
- d) Set the Spacing X, Y, and Z boxes to 3.
- e) Set Grid Labels Precision to 4 digits, then click Apply.
- f) Pick a point on the floor to display the grid of illuminance values.

You may need to zoom in to see the numbers.

7. Click Cancel on the Lighting Analysis dialog.

8. Click the Perspective button .

Rendering

When you render, you create a 2D image of the 3D radiosity solution. With Lightscape, you can perform this operation in two ways:

- Rendering the displayed radiosity solution (diffuse lighting only)
- Adding specular reflections and refining shadows using ray tracing

To render an image, you select File | Render and set the appropriate rendering options. The resulting images can be of any resolution.

Rendering Options

Lightscape can produce images of any resolution that can be output in a variety of standard file formats. The following options control the rendering process:

- **Format Type:** You can select the file format of the image from the Format list. Most standard formats are supported.
- **Resolution:** The size of the image can be set by selecting a standard image size from the Resolution list or by entering values in the Width and Height boxes.
- **Antialiasing Samples:** The level of antialiasing is used to eliminate image artifacts such as the “stepping” or “jaggies” of polygon edges.
- **Ray Tracing:** This option is used to add specular lighting effects to the radiosity solution. This more accurately shows transparency effects and refines the primary shadows from light sources.
- **Ray Trace Direct Shadows:** This option recomputes the direct energy contribution of the sun and selected light sources using the ray tracer. It eliminates the meshing artifacts that can create ‘jaggie’ shadow boundaries.
- **Ray Bounces:** This box sets the number of ray bounces (levels of reflection) to calculate when ray tracing an image.

Creating a Single High-Quality Image Using Ray Tracing

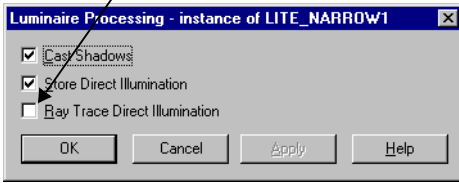
To add specular lighting effects to the radiosity solution, you render using the ray tracer. In the Rendering dialog, enable Ray Tracing and set the related options. For example, if you enabled Ray Trace Direct Illumination for any of the luminaires in the view, you should also enable Ray Trace Direct Illumination in the Rendering dialog.

Create a ray-traced image

1. Select a view of the inside of the sunroom that includes the window, some sunlight on the floor, and at least one of the vases.
2. Click the Select button , then click the Luminaire button .
3. Select the LITE_NARROW1 luminaire that is directly above the vase in one of the niches.
4. Right-click in the Graphic window, then choose Luminaire Processing.

The Luminaire Processing dialog appears.

5. Enable Ray Trace Direct Illumination to turn on ray tracing for this luminaire, then click OK.



6. Choose File | Render.

The Rendering dialog appears.

7. Set the Rendering options:

- a) Select Targa (TGA) from the Format list.
- b) Type Ray.tga in the Name box.
- c) Select Two from the Antialiasing Samples list.

Antialiasing over-samples pixels to remove “jaggies” along the edges of objects.

- d) Select Full Window from the Resolution list.
- e) Enable Ray Tracing.
- f) Enable Ray Trace Direct Illumination.

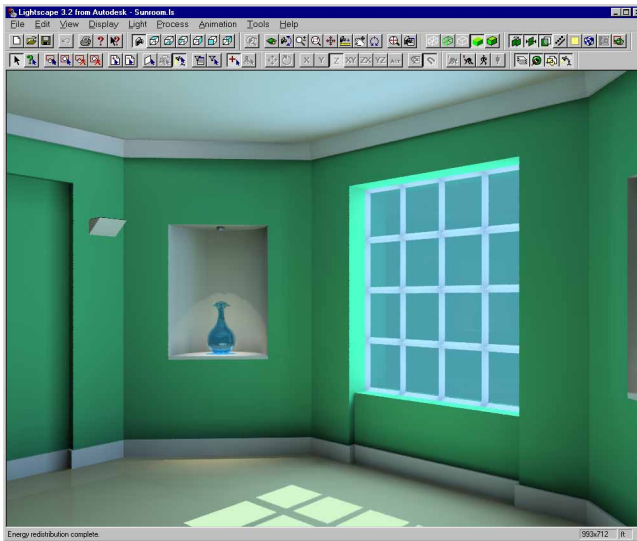
This will create better direct shadows by ray tracing the sun and the luminaire selected previously.

- g) Make sure Ray Bounces is set to 10.

This is the default setting.

- h) Click OK and confirm the additional processing.

Lightscape now ray traces the model. Soon you see the specular reflections from the vases that were not visible in the diffuse radiosity model.



The ray-traced image shown in the Graphic window is saved to *Ray.tga*. You can use the LVu utility to view this file.

Finish your session in Lightscape

1. Choose File | Save As to save this file.

The Save As dialog appears.

2. Type Sunroom.ls in the File Name box, then click Save.
3. Choose File | Exit.

Summary

Congratulations! You have reached the end of Lesson 2. At this point, you should understand:

- How to import DXF files
- How to define a block as a luminaire
- How to add luminaires from libraries
- Basics of setting materials
- Adaptive meshing and how to control the quality of a radiosity solution
- How to set interior daylight options
- How to perform an analysis of the lighting data in the model

3

The Lounge

In Lesson 2, you imported a DXF file using minimal options (filename and units). In this lesson, you start the Lightscape workflow by importing another DXF file, but this time you use some of the advanced import techniques. You assign a material map and block libraries, and learn how to merge a DXF file. After the Lounge model is imported, you learn about preparing surfaces and textures. Using the processed radiosity solution, you create an animation and produce frames of the first segment of the animation.

In this lesson:

- Import a DXF file and familiarize yourself with it
- Open material and block libraries and examine them
- Import the DXF file again using advanced import techniques
- Orient surfaces to point towards the viewer
- Align textures on surfaces
- Process the radiosity solution
- Create an animation path
- Render animation frames

3 The Lounge

At the end of this lesson, you create rendered images of the lounge that resemble the following illustration.



Stage 1: Importing Data

In this section, you learn about:

- Using Advanced DXF Import Features

You perform the following task:

- Importing Geometry

Using Advanced DXF Import Features

Lightscape contains a number of tools to optimize models that you import using the DXF translator. In part, this is because the DXF file format is missing support for material definitions, but you may also need to import revised versions of the model. In this case, preparing material maps and assigning libraries minimizes your preparation tasks. For example, you do not need to assign materials the second time you import the model.

Material Maps

By default, Lightscape assigns a material to a surface whose name and color match the color number in the DXF file. Not only are the names of these materials nondescript (usually 0 through 255), but the colors are generally not suitable for producing acceptable Lightscape results.

With the material map technique you can:

- Associate Lightscape materials to the color numbers used in the DXF file.
- Load a material map in conjunction with a DXF file. This substitutes all the colors defined in the DXF file with their associated Lightscape materials. For example, you can map a material called “wood” onto every surface that is drawn with color 1.

Block Libraries

By default, Lightscape creates a new block definition for each block when you import a DXF file. If you have pre-existing Lightscape blocks defined, you can use these definitions instead. This saves considerable time and effort in the Preparation stage.

While you are importing:

- Pre-existing Lightscape block and luminaire definitions can be mapped onto incoming DXF blocks of the same name.
- Lightscape searches through all the loaded block libraries and replaces each block encountered in the DXF file with a block of the same name stored in the libraries.

Overwrite/Merge to the Current Model

When importing geometry, you can choose either to overwrite or merge with the current model:

- Overwriting creates a new model from the imported surfaces. If you choose to overwrite, the existing model displayed in the Graphic window is replaced by the model you import.
- Merging adds the imported model to the current model displayed in the Graphic window.

Importing Geometry

Although you will eventually use advanced tools for loading this DXF file, import it now using basic settings and familiarize yourself with it.

Load the DXF file named Lounge.dxf

1. Start Lightscape if the application is not already running.

2. Choose File | Import | DXF.

The Import DXF dialog appears.

3. Click the Browse button beside the Name box.

The Open dialog appears.

4. Change the path to *C:\Program Files\Lightscape\Projects\Lesson3*.

5. Double-click *Lounge.dxf*.

6. Choose Feet from the File Units list.

This model was created with one unit in the CAD program set to equal one foot in the real world.

7. Click OK.

A dialog reports that the model measures 26 x 29 x 25.1667 feet.

8. Click Yes to open and display the file.

Examine the Lounge model

1. To see the surfaces and their edges, click the Outlined button .

The material colors are taken from the default AutoCAD color scheme. You can see that the names of these materials in the Materials table are simply DXF color numbers.

2. Look at the inside of the model:

a) Adjust the view so that you are inside the room.

For example, set the viewer position to [11, -11, 7], the focus point to [-3, -2, 7], and the near clip plane to about 0. Notice that many of the surfaces of the piano are missing (because they are oriented incorrectly and backface culling is on).

b) Save this view as Inside_Lounge.vw.

3. You can examine blocks in the block preview or in Isolate mode. To use Isolate mode, right-click CHAIR in the Blocks table, then choose Isolate.

Hint: You can also double-click CHAIR to enter the Isolate mode.

4. Orbit the block.

The shape is a simple cube (and not a chair).

5. Right-click in the Blocks table or the Graphic window, then choose Return to Full Model to end the Isolate mode.

Repeat steps 3 to 5 for the LITE_FIRE1 and LITE_SPOT1 blocks. Notice that these two light fixtures are just blocks and not luminaires.

Hint: Another way to look at blocks that are in the model is to query the instances of a block. To do this, select the block name in the Blocks table, right-click, and choose Query Instances. All instances of the block are highlighted in green.

6. Choose File | New to open a new, empty Lightscape file.

A message appears asking if you want to save changes to the model.

7. Click No because you are going to import the model again in a future exercise.

Load materials from an existing material library

Determine which materials you want to use initially in your model. There should be one material for each DXF color number that is used. (For example, the lounge model uses 14 DXF numbers.) You can define new materials or load them from a library. Lightscape includes an extensive set of material libraries that you can use. For this lesson, a library of materials is supplied.

1. Right-click in the Materials table, then choose Load.

The Open dialog appears.

2. If necessary, change the path to *C:\Program Files\Lightscape\Projects\Lesson3*.

3. Double-click *Lounge.atr*.

The Available Materials dialog displays 12 materials.

4. Choose Select All to select all the materials, then click OK to load them.

A message appears asking if you want to overwrite one or more of the existing materials.

5. Click Yes. (In this case, the only material that you overwrite is Default Attr.)

All the materials appear in the Materials table.

Assign the loaded materials to a material map and save the map

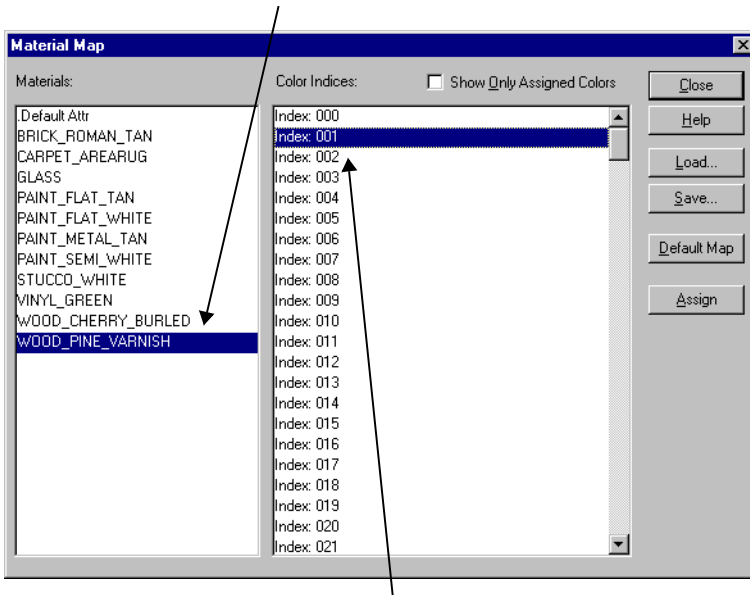
You need to assign materials to color index numbers. A material will be used for all surfaces in the imported DXF file with the color set to its assigned index number.

1. Choose Tools | Material Map.

The Material Map dialog appears.

2. Assign a material to a color index:

a) In the Materials list, select WOOD_PINE_VARNISH.



b) In the Color Indices list, select Index: 001.

c) Click Assign.

The WOOD_PINE_VARNISH material is now assigned to index 001.

3. Repeat step 2 for the following Index/Material pairs:

- 002: STUCCO_WHITE
- 003: BRICK_ROMAN_TAN
- 004: PAINT_FLAT_WHITE
- 005: PAINT_METAL_TAN
- 006: GLASS
- 007: CARPET_AREARUG
- 009: VINYL_GREEN
- 010: WOOD_CHERRY_BURLED

Note: Do not assign a material to index 008. If you make a mistake, leave the assigned color for 008 and continue with 009 and 010. (The mistake will not affect the results because 008 is not used in the DXF file.) You can also click Default Map to clear the material map and start over.

4. Enable “Show Only Assigned Colors”.

You can use this feature to quickly check the materials that you assigned.

5. Save the new material map:

a) Click Save.

The Save As dialog appears.

b) If necessary, change the path to *C:\Program Files\Lightscape\Projects\Lesson3*.

c) Type Lounge.mm in the File Name box.

d) Click Save, then click OK when the confirmation message appears.

6. Click Close to close the Material Map dialog.

Open the block library and examine the predefined Lightscape blocks

As a DXF file is imported, the blocks in the file can be replaced with other predefined blocks or luminaires that are stored in a library. You list the libraries to search when you set up the import. In this exercise, you simply examine the predefined blocks in the library that will be searched when you import.

A block library can contain both blocks and luminaires. The *Lounge.blk* library contains two blocks (CHAIR and PIANO1) and two luminaires (LITE_FIRE1 and LITE_SPOT1). You must load the blocks from the Blocks table and the luminaires from the Luminaires table.

1. Load blocks from the library:

- a) Right-click in the Blocks table, then choose Load.

The Open dialog appears.

- b) If necessary, change the path to *C:\Program Files\Lightscape\Projects\Lesson3*.

- c) Double-click *Lounge.blk*.

The Available Blocks dialog lists two blocks.

Note: You can use the block preview to examine each selected block (and then click Cancel) or continue loading the blocks and look at them in Isolate mode.

- d) Choose Select All to select both blocks, then click OK to load them.

The CHAIR and PIANO1 blocks appear in the Blocks table.

2. Load luminaires from the library:

- a) Right-click in the Luminaires table, then choose Load.

The Open dialog appears.

- b) If necessary, change the path to *C:\Program Files\Lightscape\Projects\Lesson3*.

- c) Double-click *Lounge.blk*.

The Available Luminaires dialog lists two luminaires.

Note: You can use the block preview to examine each selected luminaire (and then click Cancel) or continue loading the luminaires and look at them in Isolate mode.

d) Choose Select All to select both luminaires, then click OK to load them.

The LITE_FIRE1 and LITE_SPOT1 luminaires appear in the Luminaires table. In the Lounge model, these two blocks are not defined as luminaires. When you import the model with the library of predefined blocks, they will be replaced with luminaires that you just loaded.

Note: The LITE_SPOT1 luminaire uses the *lt_spot.ies* photometric web. This file is located in the *Lesson3* folder.

3. Select CHAIR in the Blocks table.

When you examined the Lounge model, you used Isolate mode to look at the blocks in the Blocks table. Use Isolate mode again to look at the blocks that you loaded from the library.

4. Right-click in the Blocks table, then choose Isolate.

A properly defined Breuer chair appears. When you import the Lounge model with the library of predefined blocks, the simple cube in the model will be replaced with this chair.

5. Orbit the block, then return to the full model.

6. Look at the PIANO1 block and the luminaires that you loaded from the library.

7. Choose File | New to open a new, empty Lightscape file.

When a message appears asking if you want to save changes to the model, click No.

Import the Lounge.dxf file again using the material map and block library

1. Choose File | Import | DXF.

The Import DXF dialog appears.

2. Click the Browse button beside the Name box.

The Open dialog appears.

3. If necessary, change the path to *C:\Program Files\Lightscape\Projects\Lesson3*.

4. Double-click *Lounge.dxf*.

5. Choose Feet from the File Units list.

This model was created with one unit in the CAD program set to equal one foot in the real world.

6. Load the material map that you created for this import:

- a) Click the Browse button beside the Material Map box.

The Open dialog appears.

- b) Double-click *Lounge.mm*.

7. Load the library that includes the blocks and luminaires for this import:

- a) Click the Add button beside the Block and Luminaire Libraries list.

The Open dialog appears.

- b) Double-click *Lounge.blk*.

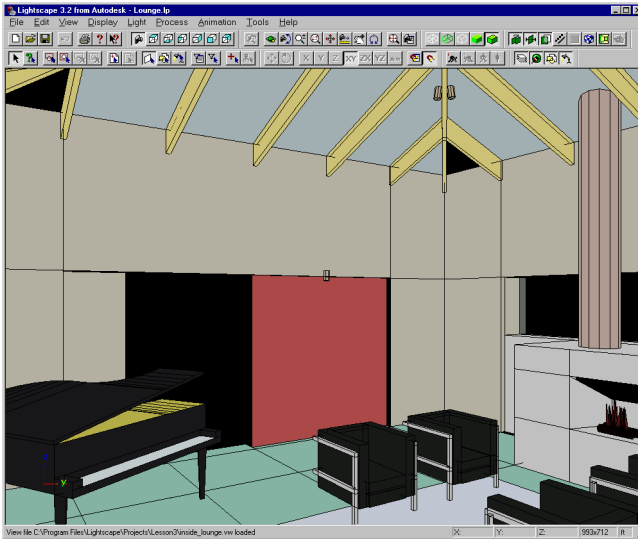
- c) For this model, the Block Creation mode should remain As Is. (The block structure of the DXF file will not be altered during the import.)

8. Click OK.

A dialog reports that the model measures 26 x 29 x 25.1667 feet.

9. Click Yes to open and display the file.

10. Load the view of the lounge that you saved earlier in this lesson (*Inside_Lounge.vw*).



Notice the advantages this model has over the first import:

- The model has the correct material names and definitions.
- The blocks in the DXF were replaced by blocks from the library. Now the chair does not look like a cube and the piano does not have “missing” surfaces.
- The blocks used as “placeholders” for the lights have been replaced with predefined luminaires.

Merge the file named Lounge_A.dxf

The file you are going to merge contains some additional uplights located in the four corners of the room. Rather than re-importing the entire model for a design revision, you can simply import the changes into Lightscape.

To import the *Lounge_A.dxf* file, you use all the same options as before except you select Merge instead of Overwrite.

1. Choose File | Import | DXF.

The Import DXF dialog appears.

2. Click the Browse button beside the Name box.

The Open dialog appears.

3. If necessary, change the path to *C:\Program Files\Lightscape\Projects\Lesson3*.

4. Double-click *Lounge_A.dxf*.

5. Select Merge from the list beside the Browse button.



This option causes the new geometry to be added to the existing model.

6. Choose Feet from the File Units list.

This model was created with one unit in the CAD program set to equal one foot in the real world.

7. Load the material map that you created for this import:

- a) Click the Browse button beside the Material Map box.

The Open dialog appears.

- b) Double-click *Lounge.mm*.

- 8.** Load the library that includes the blocks and luminaires for this import:

a) Click the Add button beside the Block and Luminaire Libraries list.

The Open dialog appears.

b) Double-click *Lounge.blk*.

c) For this model, the Block Creation mode should remain As Is. (The block structure of the DXF file will not be altered during the import.)

- 9.** Click OK.

A message appears asking if you want to save changes to *Lounge.lp*, the current Preparation file that includes the model that you just imported.

- 10.** Click No because you will save changes after the new DXF file is merged.

A dialog reports that the model measures 23.3 x 23.3 x 3.83084 feet.

- 11.** Click Yes to merge the file.

The additional LITE_SPOT1 luminaires appear in the four corners of the room.

Stage 2: Preparing Data

In this section, you perform the following tasks:

- Orienting Surfaces
- Smoothing Surfaces
- Assigning and Aligning Textures

Orienting Surfaces

Surface orientation determines which side of a surface is considered for calculating its interaction with light. For example, to simulate the lighting in a room, the walls' surfaces are oriented toward the inside of the room.

The orientation of a surface is defined by the surface normal. In some modeling systems, you can set the surface normal during the modeling process and preserve that information when you export the model to Lightscape. However, if your modeling system does not consider surface orientation or does not preserve the orientation when exporting files, you can set the orientation of surfaces in Lightscape.

When you are in Solid mode with backface culling on, surfaces with the wrong orientation are not visible (because they are facing the wrong direction). When you open the Surface Orientation dialog, backfacing surfaces are no longer culled and are bright green. This helps you identify the surfaces with incorrect orientation.

The Surface Orientation dialog offers four ways of changing a surface's orientation:

- By automatically orienting all surfaces visible to the current viewer position to face that direction using the Auto-Orient tool
- By reversing a selected surface's orientation
- By orienting a surface toward or away from a specific (focus) point in the model
- By making surfaces double-sided

Note: Use this last option sparingly because it is computationally expensive. For a material such as glass, it is better to model both sides of the plate of glass with the correct thickness between them to create accurate refraction effects.

Look at the model in Surface Orientation mode

You can orient surfaces from inside or outside the model. Notice which surfaces are green and which ones are the correct color.

1. Turn off Enhanced mode if it is on.

You cannot see the correct surface colors when this display mode is on.

2. To start Surface Orientation mode, choose Tools | Orient Surface.

You are in Surface Orientation mode when the Surface Orientation dialog is open. This mode turns off culling and displays the backfaces of all the surfaces in green.

3. Look at the model from the outside:

a) Click the View Extents button .

b) Orbit around the model.

Notice that most of the surfaces are green.

4. Look at the model from the inside:

a) Adjust the view so that you can orbit around the center of the room to look at all the inside walls. For example, set the viewer position to [0, 7, 4], the focus point to [0, -2, 4], and the near clip plane to about 0.

b) Save this view as Orbit_Inside.vw.

c) Orbit around the model.

Notice which surfaces should have their orientations reversed.

d) Load the view you just saved.

Orient a few wall surfaces using the Reverse technique

As you orient surfaces, it helps to hide some of the layers. In this exercise, you display only the walls of the lounge.

1. Turn off all layers except 0 and ROOM1_WALLS:

a) Right-click in the Layers table, then choose All Off.

b) Double-click 0 and ROOM1_WALLS in the Layers table to turn these layers on.

2. Click the Select button , then click the Surface button .

3. Click the Accumulate Pick button  to turn on this feature.

4. Select a few of the wall surfaces that are not oriented correctly.

5. On the Surface Orientation dialog, click Reverse to reverse the orientation of the selected surfaces.

The selection set remains active so that you can reverse the operation you just performed or perform another operation on the same set of surfaces.

6. Click the Deselect All button .

Orient the surfaces of another wall using the Auto-Orient technique

1. Orbit the model to show another inside wall.
2. On the Surface Orientation dialog, click Auto Orient.

The surfaces that are green (due to Surface Orientation mode) are oriented towards the current viewer position. Notice that you did not have to select any surfaces for this technique.

3. Repeat steps 1 and 2 for the other walls.

Hint: When you increase the Field of View, more surfaces become visible. Thus, more surfaces that are incorrectly oriented can be fixed at the same time. One way to increase the Field of View is to zoom out.

Orient the surfaces of the ROOM1_FLOOR and ROOM1_CEILING layers

1. Turn off the ROOM1_WALLS layer and turn on the ROOM1_FLOOR layer:

a) Select ROOM1_WALLS and ROOM1_FLOOR from the Layers table.

b) Right-click in the Layers table, then choose Toggle.

Now the 0 and ROOM1_FLOOR layers are on.

2. Using either the Reverse or Auto-Orient technique, orient the doors that are part of the ROOM1_FLOOR layer.
3. Turn on the ROOM1_CEILING layer and orient the surfaces.

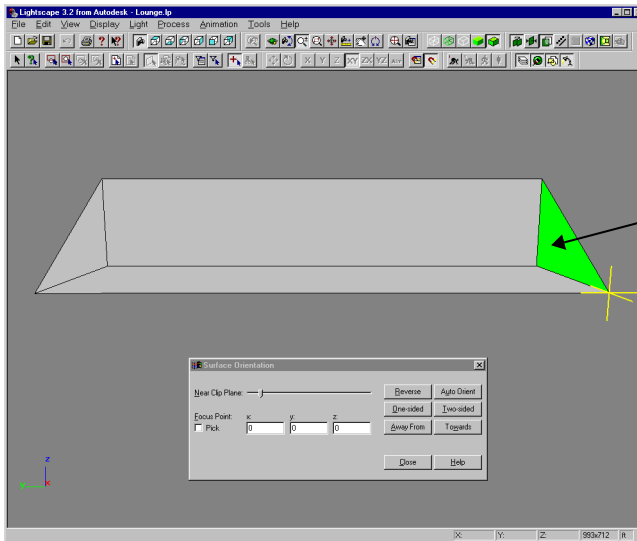
There are 11 surfaces on the ceiling that are oriented incorrectly. (Six of them are tiny surfaces on the beams.)

4. Turn on all the layers.

Orient the surfaces of blocks in Isolate mode

In this exercise, you orient the surfaces of the fireplace elements. The FIREPLACE1 block contains a stairwell made of TREAD1 blocks. You can orient surfaces of the blocks in Isolate mode or in the full model.

1. Look at the TREAD1 block in Isolate mode:
 - a) Double-click TREAD1 in the Blocks table.
 - b) Zoom out and orbit to see the block.



Notice that one of the four stair tread surfaces is oriented incorrectly.

2. Click the Select button .
3. Select the surface that is not oriented correctly.

Note: In Surface Orientation mode, you can select surfaces that are facing towards or away from you.

4. On the Surface Orientation dialog, click Reverse.
5. Right-click in the Blocks table, then choose Return to Full Model to end the Isolate mode.

6. Look at the FIREPLACE1 block in Isolate mode:

- a) Double-click FIREPLACE1 in the Blocks table.
- b) Orbit the block to see the orientation of the surfaces.

Notice that the TREAD1 blocks (in the stairwell) are oriented correctly.

7. Correct the orientation of a few of the surfaces that face into the lounge and several of the surfaces inside the stairwell. (A total of 16 surfaces need to be reversed.)

To do this, use the Reverse technique. You cannot use the Auto-Orient technique in this case because most views of the block show both inside and outside surfaces.

8. Right-click in the Blocks table, then choose Return to Full Model to end the Isolate mode.
9. On the Surface Orientation dialog, click Close to end Surface Orientation mode.

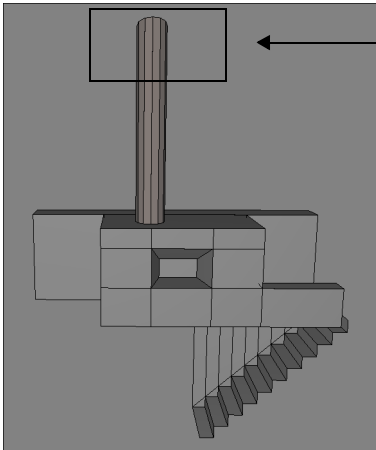
Smoothing Surfaces

In this exercise, you smooth the surfaces of the fireplace chimney. It currently is made of 12 facets. Because the chimney is a round tube, you want to display it as smooth instead of as separate surfaces.

Smooth the surfaces of the chimney

1. Load the view of the lounge named *Inside_Lounge.vw* and orbit until you can see the fireplace.
2. Turn off all the layers, then turn on the ROOM1_FIREPLACE layer.
3. Click the Enhanced button  to turn this feature on.

4. Select the surfaces that make up the fireplace chimney.



To do this quickly, use the Select Area Any Vertex feature and drag a rectangle across the top end of the chimney.

5. Right-click in the Graphic window, then choose Smoothing.

The Smoothing dialog appears.

6. Set the Smoothing Angle to 60 and click Make Smooth.

7. Click Close, then click the Deselect All button .

The surfaces of the chimney appear smoothed. Switch to the Solid display mode to see the smooth surface better.

8. Click the Enhanced button  to turn this feature off.

9. Turn on all the layers.

10. Since you have spent some time preparing the fireplace block, you can save it to the local library (*Gallery.blk*). If you need to import the DXF file again, the fixed block will be automatically swapped with the one in the DXF file.

Assigning and Aligning Textures

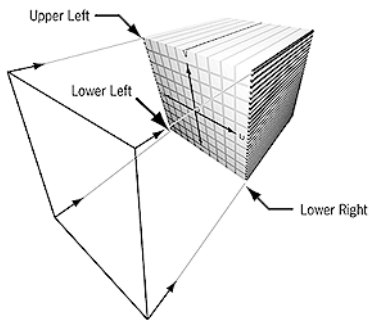
The texture assigned to a material can be positioned, rotated, and stretched during the alignment process. The alignment defines how the texture is applied to a particular surface.

There are five different ways of projecting a texture onto a surface:

- Orthographic: Projects the texture on a flat surface
- Cylindrical: Wraps the texture around a cylinder
- Spherical: Wraps the texture around a sphere
- UV: Uses coordinates that were imported with the model
- Reflection: Simulates reflection on objects

Aligning an Orthographic Texture

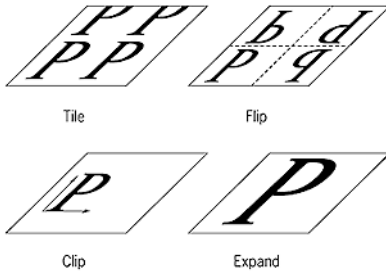
Most textures in a typical model are applied to flat surfaces and aligned using orthographic projection. An orthographic texture is aligned by choosing points for the lower-left corner, lower-right corner, and upper-left corner.



These points are used to create a plane that is projected onto the selected surfaces, applying the first copy of the texture map. Additional copies are placed in a grid on the surface based on the selected mapping mode.

Mapping Modes

Mapping modes determine where additional copies of the texture are placed. Each mapping mode can be applied separately in the horizontal and vertical directions. For example, it is possible to tile horizontally and clip vertically to create a single row of tiles along the base of a wall.



Assign a texture to the BRICK_ROMAN_TAN material

1. Click the Textures button .

There is no visible change in the model because none of the materials have textures assigned to them.

2. Double-click BRICK_ROMAN_TAN in the Materials table.

The Material Properties dialog appears.

3. Assign a texture to this material:

- a) Click the Texture tab.
- b) Click Browse.
- c) Double-click *Brick_tan.tga* in the File Name list to load this texture map.

4. In the Filter Methods group box, select Point MM from the Minimize list.

This option increases the quality of the sampling of the texture (in this case, making the block edges more clearly defined).

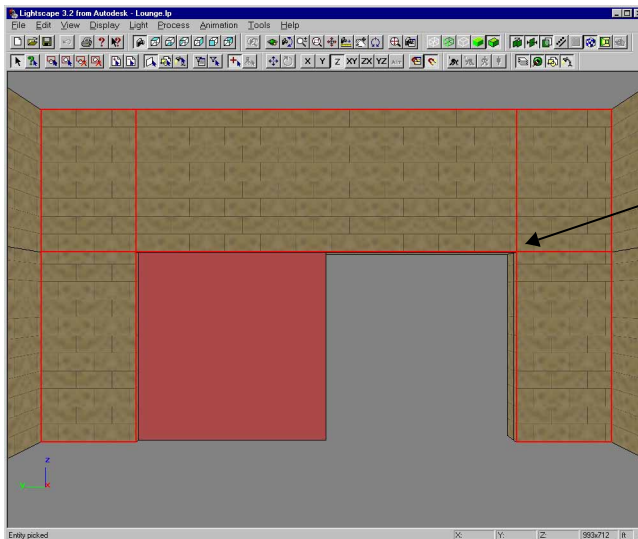
5. Enable Fixed Size, then set Width to 3 and Height to 1.5.

The concrete texture repeats every 3 feet in the horizontal direction and every 1.5 feet in the vertical direction. When you know the area represented by the image file, this option allows you to accurately scale the texture.

6. Click OK.

Align the texture to the wall surfaces

1. Turn off all layers except ROOM1_WALLS.
2. Click the Select button , then click the Surface button .
3. Change the view so that you can see all the surfaces for one wall, then select all the brick surfaces.



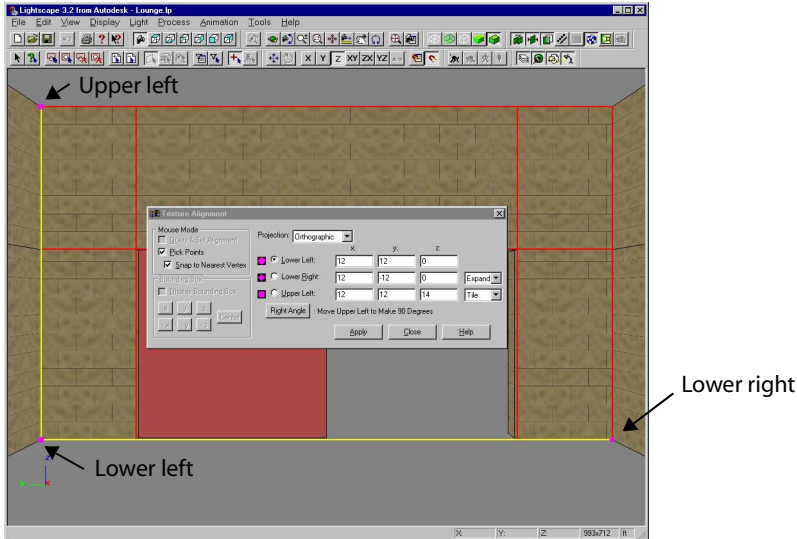
Although correctly aligned, the texture is not continuous over the whole wall; it begins from the lower-left corner of each surface.

4. Right-click in the Graphic window, then choose Texture Alignment.

The Texture Alignment dialog appears.

5. Enable Pick Points in the Mouse Selection Mode group box, and then enable Snap To Nearest Vertex (so that you do not have to be precise when picking the points).

6. Pick three points on the wall (lower left, lower right, and upper left).

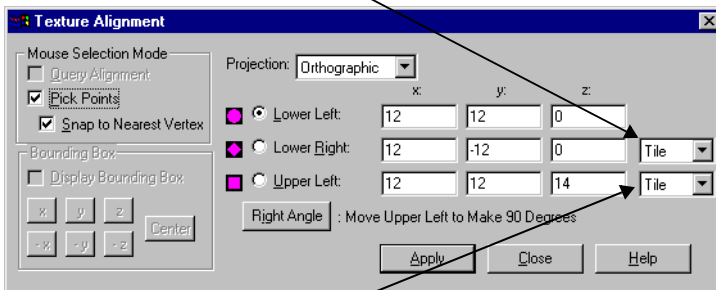


After you pick a point, the radio button for the next one is automatically selected.

Hint: The icons next to the radio buttons are used on the Graphic window to identify the three different points. For example, the round dot shows the lower-left point.

7. Set the mapping modes:

a) From the Lower Right list, select Tile.



b) From the Upper Left list, select Tile.

8. Click Apply, then click Close.

The brick texture now appears to be continuous.

9. Click the Deselect All button .

10. Repeat steps 3 to 9 for the other three walls.

Some of the brick surfaces on the other walls may already be aligned properly due to the default alignment.

Note: If you leave the Texture Alignment dialog open, you need to disable Pick Points or hold down the 1 key to select other surfaces.

11. Turn on all the layers.

Save your work

You have concluded the Preparation stage and are about to enter the Solution stage. The model you are currently working on is going to be changed during the Initiation process. It is a good idea to save your work at this time.

1. Choose File | Save As.
2. Change the path to *C:\Program Files\Lightscape\Projects\Lesson3*.
3. Type Lounge.lp in the File Name box.
4. Click Save.

Stage 3: Creating the Radiosity Solution

In this section, you learn about:

- Global Processing Parameters
- Determining Optimum Meshing Parameters
- Testing Direct Lights
- Testing Overall Light Levels
- Refining Global Meshing Parameters

You perform the following tasks:

- Preparing the Model for Initialization
- Setting the Processing Parameters
- Processing the Solution

Global Processing Parameters

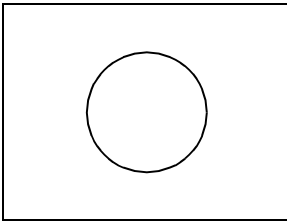
To represent variations of illumination across a surface, Lightscape automatically breaks down the surface into smaller pieces, called elements. The simulation then computes the illumination from a light source to each vertex of each element. The set of all the elements and vertices of a surface is a mesh.

To maintain as efficient a solution as possible, the system begins processing with a coarse mesh (that is, few elements) and automatically refines the mesh locally where high illumination gradients are detected. This process, called adaptive meshing, is controlled by the Receiver Group and Source Group parameters. By changing these parameters, you can provide the optimal balance between computation time, storage use, and simulation accuracy.

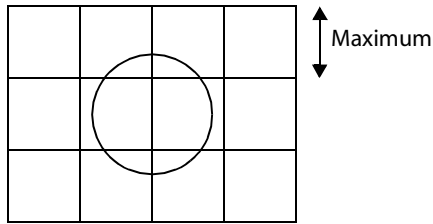
Adaptive Subdivision

Rather than meshing a surface using a uniform grid of mesh elements, the simulation process uses a more sophisticated adaptive subdivision scheme to create smaller elements in areas that contain smaller illumination details (such as shadow boundaries) and larger elements in areas where the illumination is fairly constant. This technique allocates processing resources to the areas of the model that require them.

The simulation starts by computing the contribution of the current light source to the vertices of the initial surface mesh. Then, for each mesh element, the system compares the values between the darkest and brightest of its vertices to compute an estimate of the illumination contrast over the element.



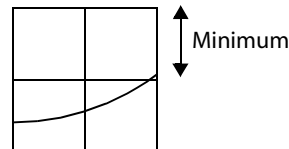
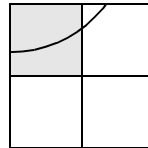
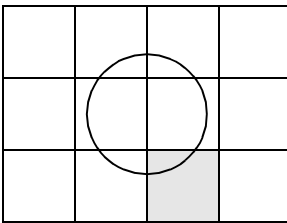
In this example, a spotlight is pointed directly at a surface.



First, the Maximum Mesh Spacing is used to create an initial mesh.

The illumination contrast is a measure of the variation in illumination across the given mesh element. A small contrast (close to 0) between two vertices of a mesh element indicates an approximately uniform illumination across the element. A larger contrast (close to 1) suggests that fine illumination details may cross the mesh element.

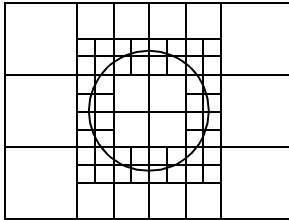
If the illumination contrast of an element is larger than the value of the Subdivision Contrast Threshold parameter, the system subdivides the element into four similar smaller elements and computes new illumination values for the new mesh vertices. It then computes the illumination contrast for the new elements and compares them against the threshold, possibly causing more subdivisions.



When the illumination contrast for an element is high enough, the element is subdivided. Here the gray element is divided into four smaller elements.

The subdivision ends when the illumination contrast is smaller than the Subdivision Contrast Threshold or the Minimum Mesh Spacing is reached. Here, one of the smaller elements (again shown in gray) is subdivided.

This process continues until the mesh elements are small enough to accurately reproduce the illumination of the surface of interest or until the Minimum Mesh Spacing is reached.



The final surface mesh has smaller elements where needed (on the edge of the light from the spotlight) and larger elements elsewhere.

Receiver Group

The parameters in the Receiver Group determine how surfaces are subdivided when they receive light, based on the following settings:

- **Maximum Mesh Spacing:** Sets the initial (coarse) size of the receiver mesh. A value of 36 inches creates an initial mesh where the longest side of each mesh element is less than or equal to 36 inches.
- **Minimum Mesh Spacing:** Sets the smallest size into which the mesh can be subdivided. A value of 4 inches means that the longest side of the smallest mesh element is greater than or equal to 4 inches.
- **Subdivision Contrast Threshold:** Sets the contrast level where subdivision occurs. The greater this value, the less likely that subdivision will take place. Subdivision continues until the contrast is no longer great enough or the Minimum Mesh Spacing value is reached.

Source Group

Just as surfaces are subdivided (sampled) when they receive light, they are also sampled when they reflect or emit light. When a surface is a light source, it is subdivided based on the Direct Source controls. When a surface reflects light, it is subdivided based on Indirect Source controls:

- **Source Subdivision Accuracy Parameters:** Control how many emitter sample points are used when shooting light energy from a source
- **Direct Source Minimum:** Sets the minimum size that an area or linear source can be subdivided
- **Indirect Source Minimum:** Sets the minimum size that a reflecting surface can be subdivided

Determining Optimum Meshing Parameters

Generally, when starting to work on a radiosity solution, you want to run several tests to find what the optimum meshing parameters should be. To test the radiosity mesh, you should start with fairly coarse settings (which give quick results but often produce a large number of mesh artifacts), then refine the solution by selecting less coarse global parameters and setting the local mesh parameters for selected individual surfaces.

Finding the right mesh parameters for a particular model may require several tests and a little practice. Once you do it a few times, you will develop a good intuition and the process gets easier. After you complete this lesson, you should return to experiment with additional settings to learn the effect they have.

You can set the initial test parameters using one or all the guidelines outlined in the next sections.

Receiver Group Guidelines

Use the model size ratio for the Receiver Group. First, calculate or estimate the average of the length, width, and height of the model. For example, a room 30 feet long x 15 feet wide x 10 feet high would have an average of $[30 + 15 + 10] / 3 = 18 \frac{1}{3}$ feet.

Use this ratio to determine the Maximum and Minimum Mesh Spacing:

- The Maximum Mesh Spacing should be 1/10 of the ratio (1.8 feet in the previous example)
- The Minimum Mesh Spacing should be 1/100 of the ratio (0.18 feet in the previous example)
- The Subdivision Contrast Threshold should be set to 0.6

Source Group Guidelines

To determine Source Group parameters:

- The Direct Source Minimum should be set to the Receiver Minimum Mesh Spacing value
- The Direct and Indirect Source Subdivision Accuracy should be set to 0.4
- The Indirect Source Minimum should be set to the Receiver Minimum Mesh Spacing value
- The Shadow Grid Size should be set to Two

Testing Direct Lights

To test the direct lights:

- Once the values are set, run the solution.
- Stop the processing after all the direct lights have shot their energy.

Examine the model to see if the lights are positioned and pointed as you want them, and check that their photometric properties are more or less correct. If not, return to the Preparation file and make any required position and aim adjustments.

- Display the model in Outline mode and note the mesh spacing.

In general, you are unlikely to adjust the mesh parameters during these early tests, but if the mesh spacing is so coarse that lighting effects do not appear at all, you should decrease the Maximum Mesh Spacing value.

Testing for Overall Light Levels and Artifacts

To test the overall light level:

- Continue to run the solution by allowing the secondary light sources to shoot.
The percentage of energy distributed is displayed on the status bar. Stop processing when the percentage is about 70%.
- Turn on Ambient (to distribute the remaining light energy evenly over all surfaces) and examine the model to see whether the overall light level is acceptable.
- If the light level is not acceptable, check the luminaire and material properties or simply adjust the brightness.
- Look for artifacts. If you see any mesh artifacts such as “jagged” shadows or shadow leaks, you can make local adjustments to the mesh on the surfaces where these artifacts occur.
- After making the appropriate changes, reset and rerun the solution.

You may need to repeat this process several times.

Preparing the Model for Initialization

Lightscape uses the Initialization Minimum Area to determine if original surfaces should be subdivided into smaller, more regular polygons before the actual radiosity calculations. Generally, you can set this value to half the Maximum Mesh Spacing value.

Set the Initialization Minimum Area before initializing the model

1. Choose Process | Parameters.

The Process Parameters dialog appears.

2. In the Tolerances group box, set Initialization Min Area to 1.3.
3. Click OK.

Begin the Solution stage by initiating the geometric data

1. Click the Solid button .
2. Click the Initiate button .
3. Click Yes to save changes.

After the *Lounge.lp* model is updated with the latest changes, the Solution model is created.

4. Click the Wireframe button .

Notice the effects of pre-meshing on the surfaces of the model. For example, adjacent coplanar surfaces that are on the same layer with the same material are grouped together. Click on the floor or a wall and most of it is selected.

Setting the Processing Parameters

As previously described, you can find values for the Receiver Group parameters using the size of the model. To calculate the model size ratio, you can measure the model with the Measure Distance tool.

When you imported the DXF file, however, a message listed the dimensions as 26 x 29 x 25 feet. The average length of a side is $26 \frac{2}{3}$ feet, so the Maximum Mesh Spacing is 2.6 feet and the Minimum Mesh Spacing is 0.26 feet.

Set the processing parameters

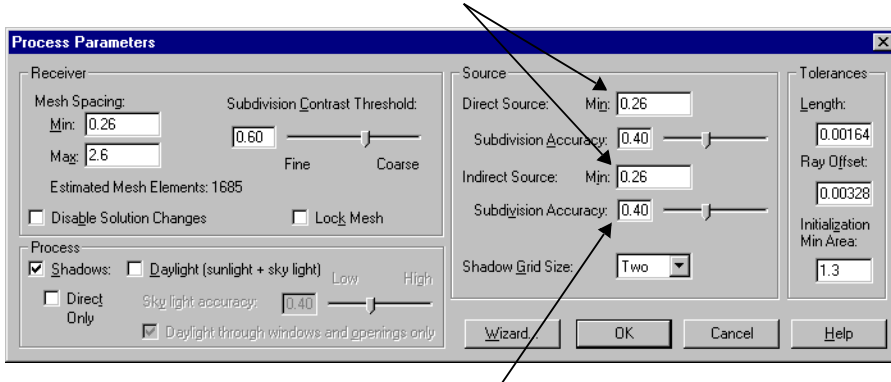
1. Choose Process | Parameters.

The Process Parameters dialog appears.

2. Set the Receiver Group parameters:
 - a) Set Mesh Spacing Min. to 0.26 and Mesh Spacing Max. to 2.6.
 - b) Set Subdivision Contrast Threshold to 0.6.

3. Set the Source Group parameters:

a) Set Direct Source Min. and Indirect Source Min. to 0.26.



b) Set Subdivision Accuracy for Direct Source and Indirect Source to 0.4.

c) Set Shadow Grid Size to Two.

4. Click OK.

Note: The settings for the processing parameters are saved with the model. If you set the values before initialization and then save the model, the Preparation file stores the settings. (When you initialize, the current settings are copied to the Solution model.) If you change the settings in the Solution model and then save, the new values are saved to the Solution file. You can choose File | Parameters | Save As to save the settings to a Parameters (.df) file, and then load the new parameters into the Preparation file.

Processing the Solution

There is a processing option that lets you easily test only the direct lights. After you process the direct light sources, you can continue with the reflected light.

Start processing the solution and test the direct lights

1. Click the Solid button .

2. Choose Process | Direct Only.

The solution runs for 15 iterations because there are 15 luminaires in the model. It stops after it has distributed the direct sources.

3. Click the Outlined button .

4. View the model to make sure that the luminaires are pointing at predictable places and that the mesh is acceptable.

A mesh is acceptable when there is enough adaptive subdivision in the areas that need it (for example, where light is cast on objects).

5. Click the Solid button .

Continue processing the solution and test the overall light level

1. Click the Go button .

Let the solution run about 35 more iterations or until about 65% of the energy has been distributed.

2. Click the Stop button .

3. Click the Ambient button .

4. View the model to make sure that the general light level and the mesh are acceptable.

5. Click the Ambient button  again to turn it off.

Save the Solution file

1. Choose File | Save As to save this file.

The Save As dialog appears.

2. Type Lounge.ls in the File Name box, then click Save.

Stage 4: Fine-Tuning the Solution

You can interrupt the processing of the radiosity solution at any time to alter or fine-tune the model's appearance. In this exercise, you change the global processing parameters, reset, and then create a better radiosity solution.

In addition to changing global processing parameters, you can change local surface meshing and other surface processing parameters such as Occluding, Receiving, and Reflecting. For more information, see “Setting the Local Processing Parameters” on page 167.

Improving the Radiosity Solution

Before resetting the solution, examine the shadow boundaries on the walls. Determine the areas where you want a finer mesh to more accurately reproduce the illumination.

Examine the model

1. Change the view so you are looking at a corner of the lounge.
2. Save the view as *Corner.vw* so you can load the same view for the next radiosity solution.
3. Look at the wall surfaces and note the quality of the shadows from the upright.
4. Click the Outlined button .
5. Choose Process | Statistics and write down the percentage of energy that is unshot and the number of mesh elements.

These statistics are found on the status bar.

Modify the global processing parameters

Earlier in this lesson, you learned about adaptive subdivision. Two ways to increase the meshing are by decreasing the Minimum Mesh Spacing and by decreasing the Subdivision Contrast Threshold. In this exercise, you try the first method.

1. Choose Process | Parameters.

The Process Parameters dialog appears.

2. Change the Mesh Spacing Min. to 0.16.

Now more adaptive subdivision will occur, so smaller elements will be created in areas that contain smaller illumination details (such as shadow boundaries).

3. Click OK.

Reset the solution and process again

1. Click the Solid button .
2. Click the Reset button .

When the warning appears, click Yes to continue resetting the radiosity solution. The model appears dark because no energy is distributed.

3. Click the Go button .

Let the solution run until about 65% of the energy has been distributed. This percentage should match the completion percentage for the first radiosity solution (not the percentage of energy that was unshot).

4. Click the Stop button .

Compare the new results with the first radiosity solution

1. Save the Solution file with a different name such as *Lounge_mesh.ls*.
2. Look at the wall surfaces and note the quality of the shadows from the upright.
3. Click the Outlined button .
4. Choose Process | Statistics and write down the percentage of energy that is unshot and the number of mesh elements.

These statistics are found on the status bar.

5. Load *Lounge.ls*.

If necessary, open the view named *Corner.vw*.

6. Switch between the two solutions and compare results.

Note: Although the shadow stepping is now reduced, you can eliminate it completely in your final images by ray tracing the direct illumination from the luminaire casting the shadow.

Stage 5: Outputting the Results

In this section, you learn about:

- Lightscape Animation

You perform the following tasks:

- Creating an Animation
- Creating Animation Images

Lightscape Animation

Lightscape provides a set of robust tools for defining and generating walk-through animations. Depending on the complexity of the model and the display hardware used, you may be able to run a real-time animation. In other situations, it may be more appropriate to save individual frames and display them using a movie-playing utility.

To generate a walk-through animation, you perform the following steps:

- Define a 3D camera path by setting a sequence of keyframes
- Define the speed of the camera as it travels along the path
- Play the animation and adjust the path and speed if necessary
- Generate the individual frames and store them on disk
- Transfer the frames to a movie file format on disk or export the images to video or film

It is easier and quicker to use Wireframe display mode rather than Solid display mode when setting up the animation. Also, because the Preparation file (.lp extension) is more compact, it is faster to define the animation using the Preparation model.

Animation

When you create the camera path, the Graphic window is split into four windows:

- Three orthographic views
- One perspective view that defaults to a view called the Director's view

You can toggle between the Director's view, which shows the current model view and a Keyframe view, which shows a view of the model as seen from the currently selected keyframe.

You create a camera path by:

- Adding keyframe points to a path. To do this, you choose the Add After or Add Before option on the Path panel and pick a point in any orthogonal view window.
- Using the Edit option to move the keyframes and adjust the camera path. To select a keyframe, click on it in the Graphic window and drag it to a position.
- Changing the shape of the camera path curve by adjusting the handles of a keyframe. The handles are dumbbell-shaped objects that are displayed tangent to the camera path at the keyframe location. To move a handle, select a single keyframe in Edit mode and then drag the handle.

Animation Motion

When setting the speed that the camera moves along the path, the display shows three views:

- The lower view shows a speed graph that you can edit. The vertical axis of the graph represents the distance along the camera path, and the horizontal axis represents time.
- The upper-left view shows the camera's view at the current time in the animation, indicated by a vertical green line on the speed graph. You can change the current time by clicking in the time legend of the graph.
- The upper-right view shows the Director's view of the model.
- By turning Grid on, you can display a light blue grid on the graph where the horizontal grid lines represent the location of camera path keyframes.

You define the camera speed by adding control points on the camera motion spline. You can do this by selecting Insert and clicking locations in the speed graph.

- You use the Edit option to move the speed keyframes and adjust the camera motion spline.
- You can change the shape of the camera motion spline by adjusting the handles of a control point. Changing the curvature changes the acceleration: where the slope is steep, the camera moves quickly and where flat, the camera moves slowly. If the spline dips from left to right, the camera moves backward.

Creating an Animation

In this exercise, you create an animation and then adjust the speed during the middle of the path. The camera path starts near the piano and moves around the edge of the room.

Create the animation

1. Load the Preparation model:

- a) Choose File | Open.

- b) Double-click *Lounge.lp*.

2. Click the Wireframe button .

3. Choose Animation | New.

This command resets the animation settings.

4. Choose Animation | Edit.

The Animation dialog appears with a blank animation (no keyframes).

5. Click the Path tab.

6. Select Add After from the Mode list.

7. Click the Select button .

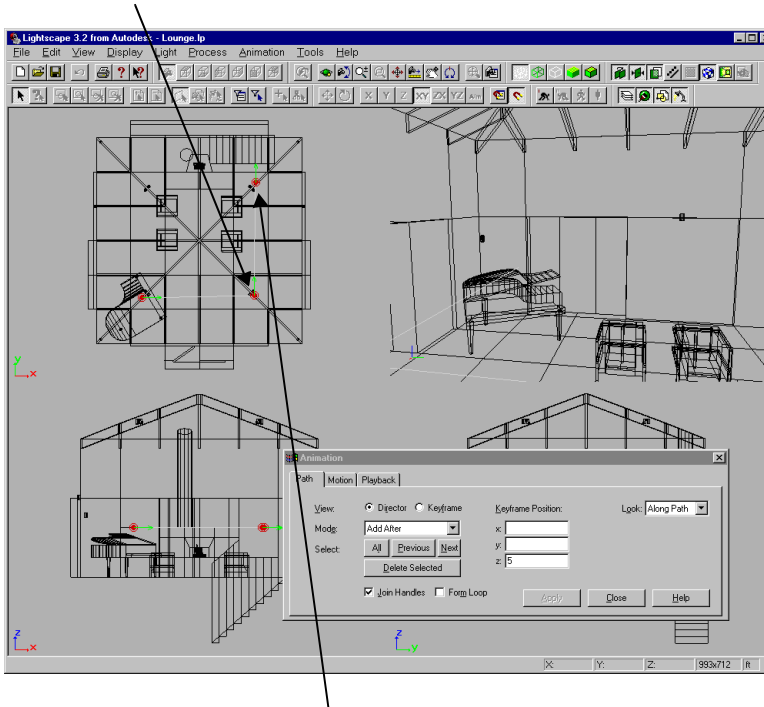
8. Add three points to the path:

a) In the top view window in the Graphic window, click a point in the lower-left corner (on the piano).

b) Type 5 in the Z box and click Apply.

This sets the height of this and all subsequent points to 5 feet above the floor.

c) Click a point near the lower-right corner to add a second keyframe.

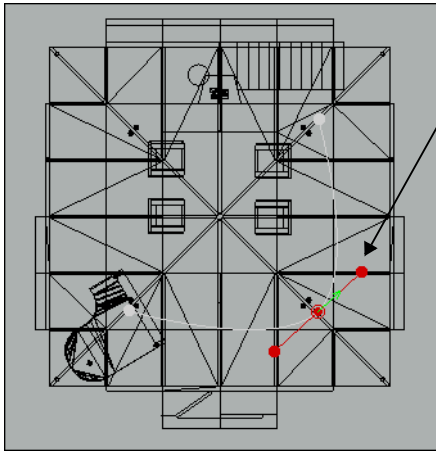


d) Click a point in the upper-right corner to add the last camera path keyframe.

Edit the handles of the three points so the path is C-shaped

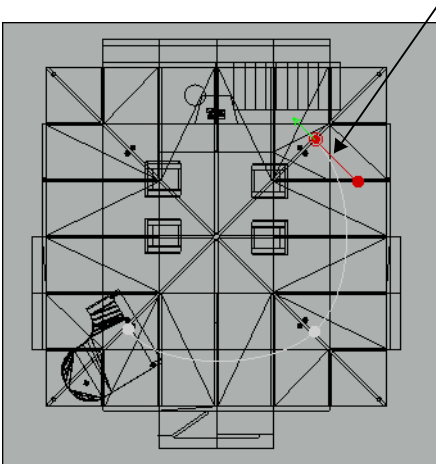
1. Select Edit from the Mode list.
2. Click on the middle keyframe in the top view window.

3. Drag the handles of the keyframe to a 45-degree position (lower left to upper right).



Hint: First click to select the keyframe, then click and drag to move the handles. If you click and drag an unselected keyframe (in the same motion), you will drag the keyframe instead of the handles.

4. Click on the first keyframe to select it.
5. Drag the handles of the keyframe to a 45-degree position (upper left to lower right).
6. Click on the last keyframe to select it.
7. Drag the handles of the keyframe to a 45-degree position (lower right to upper left) to finish forming the C-shaped curve.



Identify the Look path

Not only do you need to specify where the camera is during its motion but also the direction that the camera is looking. In this exercise, set the camera to look at the same point (on one of the chairs) throughout the length of the animation.

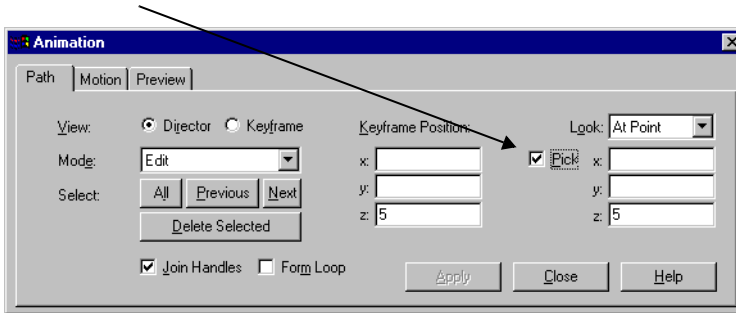
1. Click the All button in the Select area.

This selects all three keyframe points.

2. Select At Point from the Look list.

A green Look At line is added to each keyframe. By default, the lines point to the origin of the model, so you will need to move them.

3. Enable Pick.



This sets the function of the left mouse button to a special selection mode used for Pick.

4. In the top view window, select a point on the arm of a chair.

The green Look At lines point to the selected location.

5. Disable Pick to turn off the Pick mouse mode.

Create a motion spline

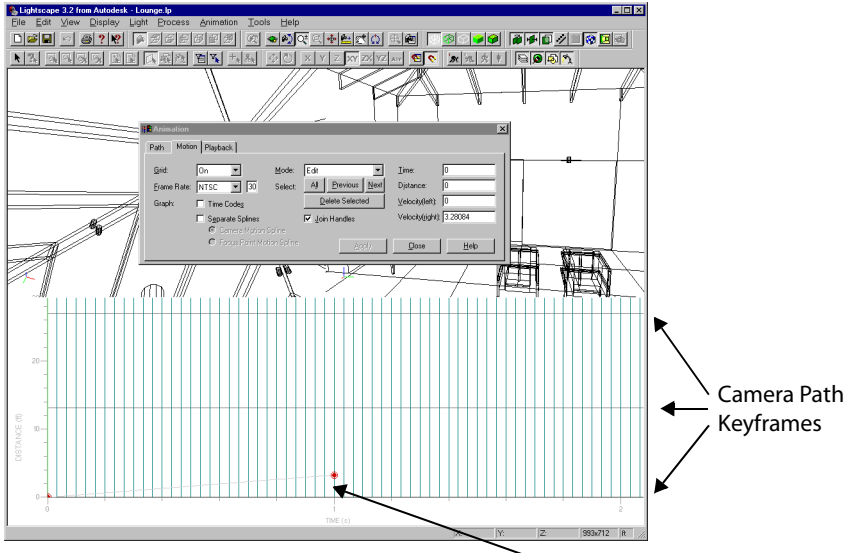
1. Click the Motion tab in the Animation dialog.

The Motion Editor appears.

2. Select On from the Grid list.

- Click the Zoom button , then drag downwards on the speed graph to show all keyframes. Then drag to the left on the graph until two seconds are showing.

The three camera path keyframes are represented by horizontal lines on the graph. One of these lines overlaps the 0 distance axis along the bottom of the graph. The other two show the real distance along the camera path.



One control point on the camera motion spline is at 0 feet and 0 seconds. By default, the other control point is set at a distance equal to 3.28 feet (1 meter) and a time equal to 1 second (30 frames in NTSC).

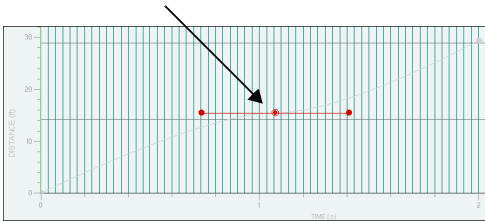
Hint: You can change the background and wireframe colors to see the graph better.

- Click the Select button , then drag the control point to the top horizontal keyframe line and to a time equal to 2 seconds.

Hint: You should click and drag the unselected control point (in the same motion) to drag the control point instead of the handle. If you accidentally drag the handle instead of the control point, drag the control point and then drag the handle back on top of the control point.

You can also select the control point, enter values in the Distance and Time boxes, and then click Apply. The distance depends on the total length of the camera path.

5. Add a control point:
 - a) Select Insert from the Mode list.
 - b) Click the Select button .
 - c) On the speed graph, click in the middle of the camera motion spline.
6. Select Edit in the Mode list.
7. Click on the new control point and drag its handles to a horizontal position to form an “S” curve.



The camera starts, decelerates to a stop in the middle of the path (at the horizontal position of the spline), and then accelerates as it continues around the room.

Play the animation

1. Click the Playback tab in the Animation dialog.

By default, the time in the Playback End Time box equals the duration of the animation (from the Motion panel).

Note: If the time equals 1, enter -1 in the Playback End Time box and click Apply. The correct value will appear in the Playback End Time box.

2. If necessary, click the Rewind button to rewind to the beginning of the animation.
3. Click the Play button to view the animation.

You may also want to view the animation in Solid mode.

4. Click Close to end the animation editing session.

Save the animation

Only one animation can be open. You should save the current animation so that you can use it again later.

1. Choose Animation | Save As.

The Save As dialog appears.

2. Change the path to *C:\Program Files\Lightscape\Projects\Lesson3*.
3. Type *Lounge.la* in the File Name box.
4. Click Save.

Creating Animation Images

In this exercise, you save animation frames of the model using OpenGL rendering. OpenGL uses hardware to display a rendered image. It is not as accurate or as full-featured as the Lightscape ray tracer, but it is much faster.

Save animation images of the model using OpenGL

1. Load the Solution model:

- a) Choose File | Open.
- b) Double-click *Lounge.ls*.

2. Load the animation file:

- a) Choose Animation | Open.
- b) Double-click *Lounge.la*.

3. Click the Solid button .

4. Click the Textures button  so that textures are displayed.

5. Choose File | Render.

The Rendering dialog appears.

6. Select Windows Bitmap (BMP) from the Format list.

7. Type Rad in the Name box of the Output File group box.

The animation files will be saved with the names *Rad0001.bmp*, *Rad0002.bmp*, and so on.

8. Select Test (300 x 300) from the Resolution list.

9. Make sure that Ray Tracing is not enabled.

10. Specify the animation frames that you want to process:

- a) In the Frame Generation group box, select Animation File from the Source list.

The current animation file (*Lounge.la*) is used.

- b) Type 0 in the From box.

- c) Type 14 in the To box.

For this exercise, you create 15 of the 60 frames.

11. Click OK.

15 images (½ second of animation) are processed and the files are saved to the current folder.

Note: You can use another program (such as *Makeavi.exe*) to make a movie clip from the animation images.

Save animation images of the model using the ray tracer

In this exercise, you render the same views but enable ray tracing to add the specular floor component and the procedural textures.

1. Choose File | Render.

The Rendering dialog appears.

2. Select Windows Bitmap (BMP) from the Format list.

3. Type Ray in the Name box of the Output File group box.

4. Select Test (300 x 300) from the Resolution list.

5. Enable Ray Tracing.

6. Specify the animation frames that you want to process:

a) In the Frame Generation group box, select Animation File from the Source list.

The current animation file (*Lounge.la*) is used.

b) Type 15 in the From box.

c) Type 29 in the To box.

For this exercise, you create 15 of the 60 frames. The files are saved with the names *Ray0015.bmp*, *Ray0016.bmp*, and so on.

7. Click OK.

Lightscape ray traces the model. Soon you will see the specular reflections and procedural textures that were not visible in the diffuse radiosity model or in the OpenGL rendered image. The processing takes longer than the OpenGL animation frames because of the ray tracing. For rendering animations, you would typically use the batch processing utilities. You can also use LSnet to distribute the rendering of frames over many processors on a network.

8. Choose File | Exit to finish your session in Lightscape.

You do not have to save the Solution model because no changes have been made to it.

Summary

Congratulations! You have reached the end of Lesson 3. At this point, you should be able to:

- Load DXF files using libraries and a material map
- Merge DXF Files
- Orient and smooth surfaces
- Assign and align textures
- Set global processing parameters without using the wizard
- Create an animation
- Render animation frames

4

The Exterior Model

In Lessons 2 and 3, you started the workflow by importing a DXF file. In this lesson, you import an AutoCAD (DWG) file into Lightscape. After preparing the materials and luminaires for the exterior model, you set local and global processing parameters.

Using local parameters, you change the meshing for selected surfaces, and make some surfaces non-occluding, non-receiving, or nonreflecting. To process the radiosity solution for the exterior model, you set the parameters that control daylight. At the end of the lesson, you play an animation for the exterior model.

In this lesson:

- Import an AutoCAD (DWG) file
- View the model
- Assign the correct textures to surfaces
- Assign procedural textures to the water, ground, and walls
- Add a luminaire from a library
- Replace placeholder surfaces with luminaires from a library
- Set local processing parameters for surfaces
- Process the radiosity solution for night, daytime, summer, and winter exteriors
- Play an existing animation for the model

At the end of this lesson, you create an image of the exterior model that resembles the following illustration.



Stage 1: Importing Data

In this section, you learn about:

- Using Good Modeling Practices

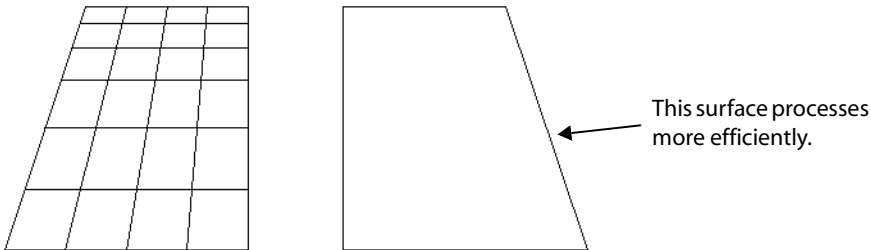
You perform the following task:

- Importing AutoCAD Files

Using Good Modeling Practices

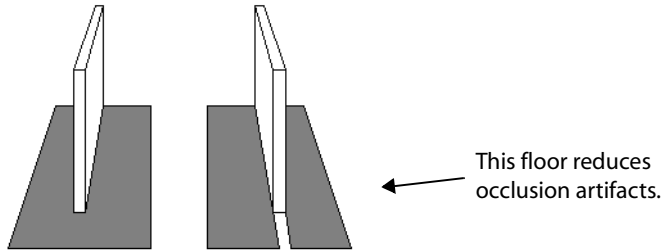
The exterior model for this lesson was created in AutoCAD. When you create geometry in your CAD or modeling application, there are several modeling habits you should practice. The techniques listed in this section are important because they result in a better model for Lightscape. Always that remember Lightscape calculates the relationship between light and surfaces.

1. Model only those surfaces that are visible. During processing, Lightscape calculates the effect of the current light source on every surface in the model. Extraneous surfaces (such as the inside surfaces of walls that never receive light) increase processing time.
2. Create large contiguous surfaces rather than many small discrete surfaces.



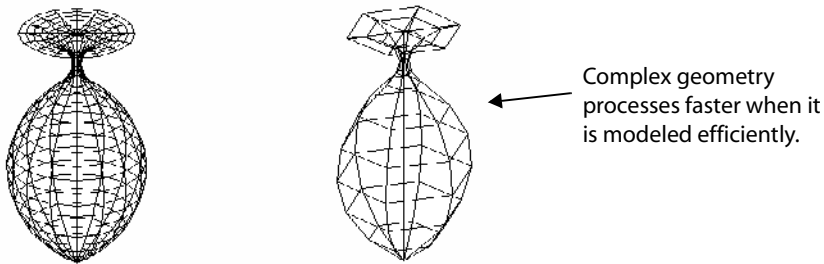
When several surfaces are used to represent a plane, each of the surfaces must be considered separately when reflecting energy into the environment. This increases processing time.

3. Model surfaces that are occluded by other surfaces as two or more separate surfaces.



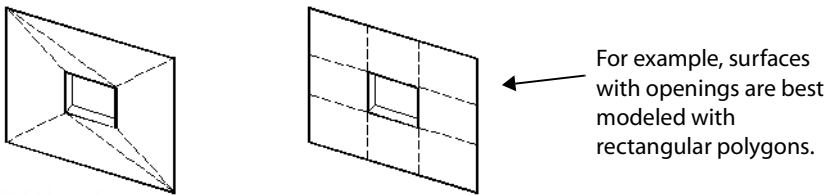
For example, where a wall intersects a floor plane or ceiling, the floor or ceiling surface should be made of two surfaces, not one.

4. Model surfaces in the most efficient way possible.



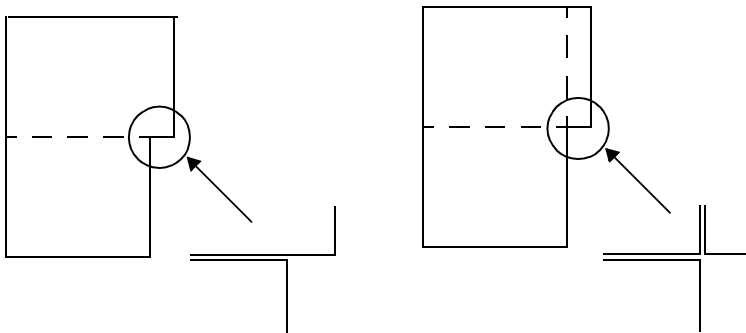
When creating a revolved surface, set the tessellation complexity of the object to a coarse value, and use smoothing within Lightscape.

5. Where possible, model surfaces as regular polygons.



Many shadow artifacts result from meshing strangely shaped surfaces (such as adjacent, long, thin, triangular surfaces). Rectangular polygons and equilateral triangles produce the best results.

6. You should try to avoid 'T' vertices. When you initiate a Preparation file containing 'T' vertices, Lightscape might recreate unwanted triangulation.



To avoid a 'T' vertex (as shown on the left), create the surface with three sections instead of two. The better configuration is shown on the right.

Importing AutoCAD Files

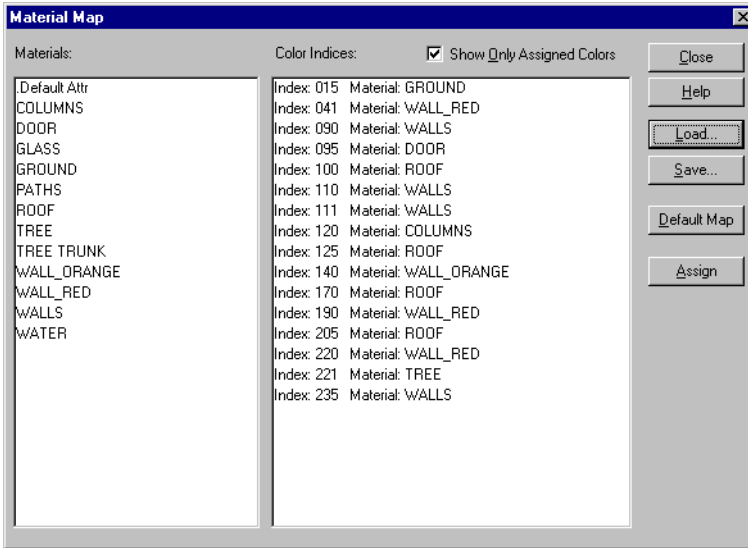
When you receive a model created by someone else, you may need to modify parts of the geometry to suit Lightscape. Sometimes, the geometry is good, but the surfaces are not assigned colors in a consistent manner. In this case, you can return to the modeling application or fix the materials in Lightscape.

The layers in DWG files are imported into Lightscape. The exterior model for this lesson is assigned colors according to its layers, but some of the layers contain geometry that needs to be assigned more than one material. For example, the DOOR1 layer includes two doors, a door frame, and two glass surfaces. During import, all surfaces in this layer are assigned the DOOR material, so you will assign materials to the frame and glass surfaces in Lightscape.

Material Maps

Objects in AutoCAD can have color numbers, but not materials. When you import a DWG file, you can specify a Material Map (.mm) file so the AutoCAD colors will be mapped to Lightscape materials. Thus, the colors defined in the DWG file are substituted with their associated materials.

You created a material map for the DXF file in Lesson 3, but the material map for this lesson is provided. If you want to create this file, load the materials in *Exterior.ctr*, choose Tools | Material Map, and assign the materials as listed below.



You may notice that a few of the materials listed on the left are not mapped to color indices. These materials will be assigned to various surfaces on layers that contain geometry that needs to be assigned more than one material.

Blocks and Layers

When you imported the DXF file in Lesson 3, you specified a block library so the blocks in the DXF file were replaced by the blocks of the same name stored in the block library. The objects in the DWG file for this lesson are not organized into blocks, but they are arranged onto many layers. This breaks the model into discrete parts, which can then be easily edited. (As an alternate organizing strategy, you can create blocks from the objects.)

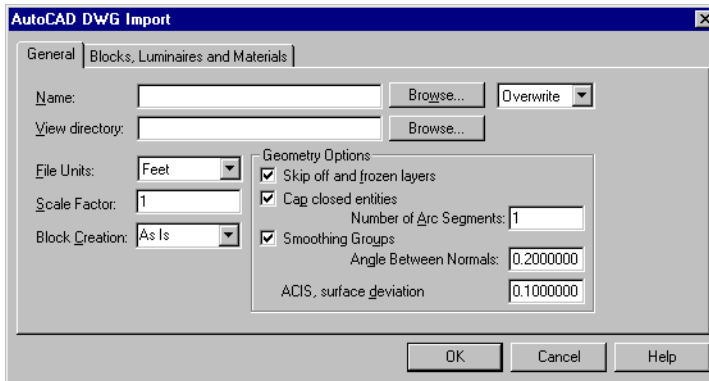
For more information on importing DWG files, see Chapter 5, “Importing Geometry” in the *Lightscape User's Guide*.

Import the AutoCAD DWG file

1. Start Lightscape if the application is not already running.

2. Choose File | Import | DWG.

The AutoCAD DWG Import dialog appears.



3. On the General panel, click the Browse button beside the Name box.

The Open dialog appears.

4. Change the path to *C:\Program Files\Lightscape\Projects\Lesson4*.

5. Double-click *Exterior.dwg*.

6. Select Feet from the File Units list.

This is the unit of measurement with which the model was created.

7. Make sure the mode (next to the Name box) is Overwrite, the Scale Factor is 1, the Block Creation mode is As Is, and Smoothing Groups is enabled.

8. Click the Blocks, Luminaires, and Materials tab.

9. Click the Browse button next to the Material Map box.

10. In the Open dialog, double-click *Exterior.mm*.

11. Click OK.

A dialog reports that the model is 165 x 137 x 60.9249 feet. Click Yes to open and display the file.

Change the background color to gray

While preparing the model, it sometimes helps to use a light background color.

1. Choose File | Properties.

The Document Properties dialog appears.

2. Click the Colors tab.

3. Set color values by typing numbers directly in the boxes (on the left of the dialog):

- a) Set Background H (Hue) to 0.
- b) Set Background S (Saturation) to 0.
- c) Set Background V (Value) to 0.70.

4. Click OK.

View the model

1. Click the Perspective button .2. Click the View Extents button .

3. Look at the exterior model:

- a) Adjust the view so that you are looking at the lower elevation.

For example, choose View | Setup and set the viewer position to [-95, -110, 50], the focus point to [-35, -50, 10], the near clip plane to about 0, and the field of view to 75 degrees.

- b) Save this view as Lower_Elevation.vw.

- c) Become familiar with the model as you practice setting up views.

For example, look at the courtyard, entrance, and upper elevation. Notice that the windows are assigned the same material as the walls and the tree trunks are assigned the same material as the trees.

Stage 2: Preparing Data

In this section, you perform the following tasks:

- Assigning the Correct Textures to Surfaces
- Assigning Procedural Textures
- Loading and Placing Light Fixtures From a Library
- Setting the Local Processing Parameters
- Setting the Global Processing Parameters

Assigning the Correct Textures to Surfaces

As explained earlier, you need to correct some of the material assignments in this model because some of the layers contain geometry that needs to be assigned more than one material.

Load the material library

1. Right-click in the Materials table, then choose Load.

The Open dialog appears.

2. Double-click *Exterior.atri*.

This file is located in the *Lesson4* folder.

3. In the Available Materials dialog, click Select All, then OK.

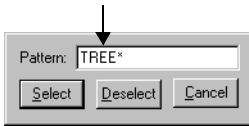
Note: Instead of loading all the materials, you can load only the materials that are not already loaded (GLASS, PATHS, TREE TRUNK, and WATER).

4. When the warning message appears, click Yes to overwrite the existing materials.

Assign a material to the tree trunks

1. Turn on all layers that start with the string "TREE":

- a) Click the Outlined button .
- b) Right-click in the Layers table, then choose All Off.
- c) Right-click in the Layers table, then choose Select Pattern.
- d) Type TREE* in the Pattern box and click Select.



Note: This is a case-sensitive text box. The asterisk is a wildcard character.

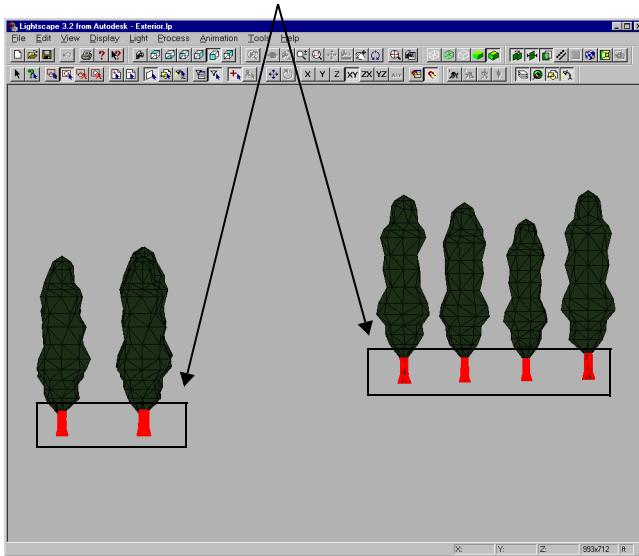
The thirteen TREE layers are selected.

- e) Right-click in the Layers table, then choose On.

2. Select the tree trunk surfaces:

- a) Click the Front button .
- b) Click the View Extents button .
- c) Click the Accumulate Pick button .
- d) Click the Area All button , then click the Surface button .

e) Draw two separate rectangular selection boxes around the tree trunks.



All the tree trunk surfaces are highlighted.

3. Assign the TREE TRUNK material:

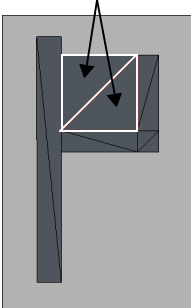
- a) Right-click in the Graphic window, then choose Assign Material.
- b) In the Assign Material dialog, double-click TREE TRUNK.
- c) Click the Deselect All button .

Assign the GROUND material to the inside of the courtyard

- 1. Turn on the COURT1 layer:
 - a) Right-click in the Layers table, then choose All Off.
 - b) Double-click COURT1 in the Layers table.

2. Select the two courtyard surfaces:

- a) Click the Top button .
- b) Click the Select button , then click the Surface button .
- c) Select the two center surfaces.



3. Assign the GROUND material:

- a) Right-click in the Graphic window, then choose Assign Material.
- b) In the Assign Material dialog, double-click GROUND.
- c) Click the Deselect All button .

Assign the correct materials to the door surfaces

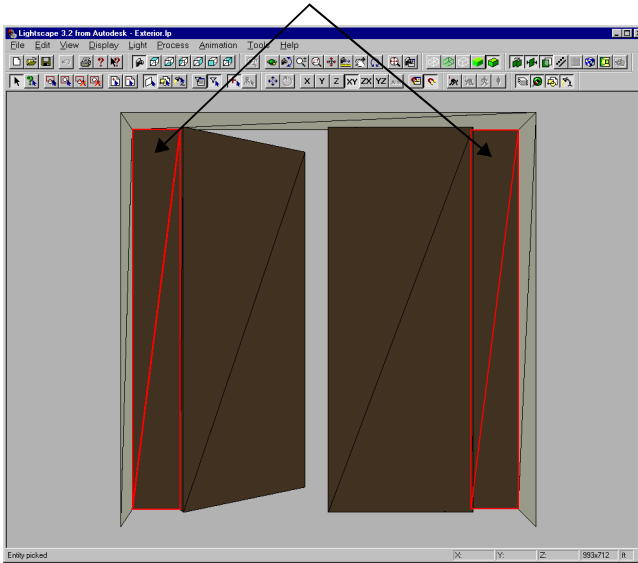
1. Turn on the DOOR1 layer:

- a) Right-click in the Layers table, then choose All Off.
- b) Double-click DOOR1 in the Layers table.

2. Select the door frame surfaces:

- a) Click the Perspective button .
- b) Click the View Extents button .
- c) Click the Select button , then click the Surface button .
- d) Select the six surfaces of the door frame (on the left, right, and top edges).

3. Assign the WALLS material:
 - a) Right-click in the Graphic window, then choose Assign Material.
 - b) In the Assign Material dialog, double-click WALLS.
 - c) Click the Deselect All button .
4. Select the four window surfaces.



5. Assign the GLASS material, then click the Deselect All button .
6. Turn on the DOOR2, DOOR3, and DOOR4 layers.
 - a) Right-click in the Layers table, then choose All Off.
 - b) Select DOOR2, DOOR3, and DOOR4 in the Layers table.
 - c) Right-click one of the layer names, then choose On.

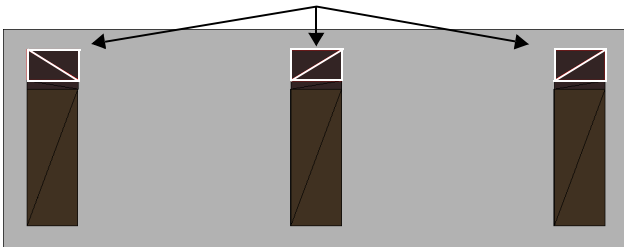
7. Select the outside door surfaces:

- a) Click the Right button .
- b) Click the View Extents button .
- c) Click the Select button , then click the Surface button .
- d) Select the six door surfaces (two per door).

8. Assign the DOOR material, then click the Deselect All button .

9. Select the window surfaces:

- a) Click the Select button , then click the Surface button .
- b) Select the six window surfaces (two per door).



10. Assign the GLASS material, then click the Deselect All button .

Assign the GLASS material to the windows

1. Turn on all layers that start with the string “WIND”:

- a) Right-click in the Layers table, then choose All Off.
- b) Right-click in the Layers table, then choose Select Pattern.
- c) Type WIND* in the Pattern box and click Select.

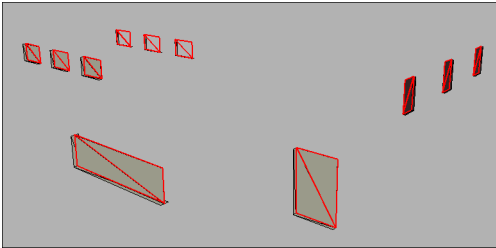
Note: This is a case-sensitive text box. The asterisk is a wildcard character.

The eleven window layers are selected.

- d) Right-click in the Layers table, then choose On.

2. Select the window (glass) surfaces:

- a) Click the Perspective button .
- b) Click the View Extents button .
- c) Click the Select button , then click the Surface button .
- d) Orbit in the Perspective projection to select the 22 window surfaces (two per window) of the WIND layers.



Hint: You can also select the surfaces in the Left (3 windows), Front (3 windows), and Back (5 windows) projections.

3. Assign the GLASS material, then click the Deselect All button .

Assign the correct materials to the ramp and pool

The courtyard includes a path that leads to a reflecting pool. The water surfaces and the path (including the bottom surfaces of the pool) need to be assigned the correct materials.

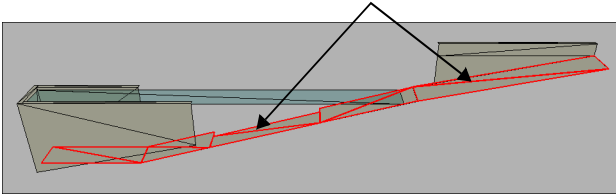
1. Turn on the RAMP1 layer:

- a) Right-click in the Layers table, then choose All Off.
- b) Double-click RAMP1 in the Layers table.

2. Select the two water surfaces:

- a) Click the Perspective button .
- b) Click the View Extents button .
- c) Orbit slightly to look down at the water surfaces and all the path surfaces.
- d) Click the Select button , then click the Surface button .
- e) Select the two long water surfaces.

3. Assign the WATER material, then click the Deselect All button .
4. Select the path surfaces:
 - a) Click the Select button , then click the Surface button .
 - b) Select the ten path surfaces that are above and below the water surfaces.



5. Assign the PATHS material, then click the Deselect All button .
6. Turn on all layers.
7. Click the View Extents button , then orbit the model to check the new material assignments.
8. Turn off Accumulate Pick.

Assigning Procedural Textures

A procedural texture modulates the color of a surface, making it appear more interesting. Since AutoCAD DWG files do not contain material information, you used a material map when you imported the file into Lightscape. The materials that are loaded do not have procedural textures set up. The two methods for generating procedural texture maps in Lightscape are Intensity Mapping and Bump Mapping.

Intensity Mapping

Intensity mapping varies the reflectance of the material, resulting in smooth variations in intensity over the surface. These variations can make a surface look dirty or slightly wavy. There are three settings for intensity mapping:

- The Width (specified in real units) controls the size of each intensity patch.
- The Contrast controls the variation in intensity.
- The Complexity controls the number of intensity layers of slightly different period to be added together. When several layers are added together, the intensity has fractal-like properties, sometimes called turbulence.

Bump Mapping

Bump mapping creates the appearance of bumps or dimples on the surface. There are three settings for bump mapping:

- The Width (specified in real units) sets the distance between bumps on the surface.
- The Height sets the height of the bumps relative to their width. A negative value turns every bump into a depression.
- The Baseline sets the amount of the surface to be bump mapped. A smaller value means less of the surface has bump mapping applied to it.

Assign a wavy procedural texture to the water

1. Double-click WATER in the Materials table.

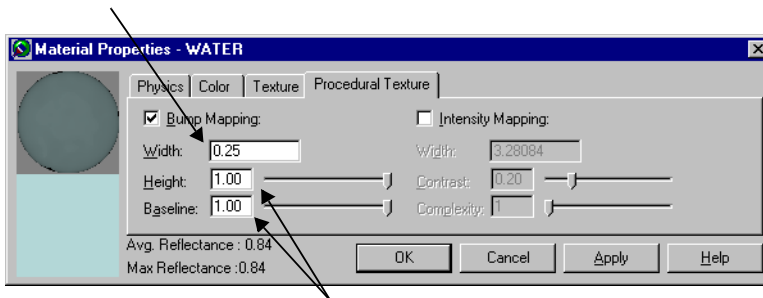
The Material Properties dialog appears with the WATER properties.

2. Click the Procedural Texture tab.

3. Set up the procedural texture:

a) Enable Bump Mapping.

b) Change the value in the Width box to 0.25.



c) Adjust the Height slider to 1.00 and the Baseline slider to 1.00.

4. Click OK to update the material and close the dialog.

Hint: If you click Apply (instead of OK), the dialog stays open and you can quickly assign a procedural texture to another material.

Procedural textures are only apparent when an image is ray traced. The water will have a wavy appearance.

Assign a procedural texture to the ground

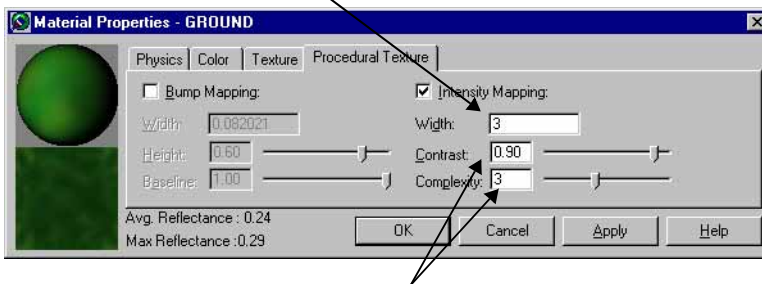
1. Double-click GROUND in the Materials table.

The Material Properties dialog appears with the GROUND properties.

2. Click the Procedural Texture tab.

3. Set up the procedural texture:

- a) Enable Intensity Mapping.
- b) Type 3.0 in the Width box.



- c) Adjust the Contrast slider to 0.90 and the Complexity slider to 3.

4. Change a few settings for the texture map:

- a) Click the Texture tab.
- b) Enable Fixed Size and type 10 in both the Width and Height boxes.
- c) Set the Brightness slider to 0.3.

5. Click OK to update the material and close the dialog.

The grass will have a color variation over its surface when an image is ray traced.

Assign stucco procedural textures to the walls

1. Select WALLS, WALL_RED and WALL_ORANGE from the Materials table.

To select more than one material, press the Ctrl key while selecting the material names.

2. Right-click in the Materials table, then choose Edit Properties.

The Material Properties dialog shows those properties that are common to the three materials. Notice there are no numeric settings for the color (on the Color panel), for example, but the Refractive Index slider (on the Physics panel) is set to 1.00, which is how all three materials are currently set.

3. Click the Procedural Texture tab.

4. Set up the procedural texture for the walls:

- a) Enable Bump Mapping.

- b) Change the value in the Width box to 0.05.

- c) Adjust the Height slider to 0.50 and the Baseline slider to 1.00.

5. Click OK to update the three materials and close the dialog.

Procedural textures are only apparent when an image is ray traced. The walls will have a stucco-like finish.

Loading and Placing Light Fixtures From a Library

Lightscape can store blocks and luminaires from any model in a block library. The library contains the complete definition of the object—its geometry, material, surface properties, and, in the case of luminaires, the photometric definition of the source. You can load blocks and luminaires into any Lightscape model once they are stored in a library.

If you load a block that has the same name as a block in the current model, the new definition of the block overwrites the existing one and all instances of the old block will instance the new one. This is a powerful feature that allows you to change from one version of a block to another—from a simple to a complex representation of the same object, for example.

Load two luminaires from a block library

1. Right-click in the Luminaires table, then choose Load.

The Open dialog appears.

2. Change the path to *C:\Program Files\Lightscape\Projects\Lesson4*.

3. Double-click *Exterior.blk*.

The Available Luminaires dialog appears.

4. Click Select All to select LITE_EXTERIOR1 and LITE_EXTERIOR2.

5. Click OK.

The two luminaires are added to the Luminaires table.

Add a luminaire to the front entrance

1. Turn off all layers except DOOR1 and make this the current layer.

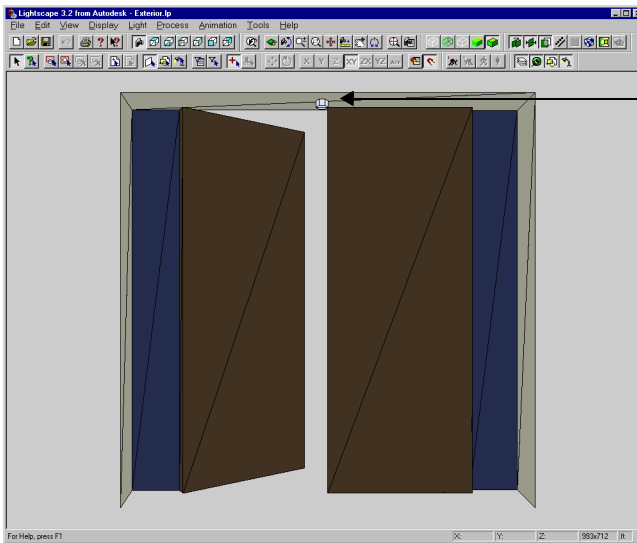
Note: To make it the current layer, select DOOR1 in the Layers table, right-click, then choose Make Current. A blue C appears to the left of the DOOR1 layer.

2. Make sure that the Perspective projection is still active.

3. Click the Outlined button .

4. Click the View Extents button  to show only the door.

5. Drag an instance of LITE_EXTERIOR1 from the Luminaires table to the top of the doorway.



Pick a point on the ceiling of the frame centered above the doors.

The luminaire is placed on the frame using the insertion point that is on the top of the fixture. It points down towards the ground.

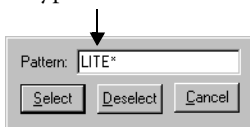
If necessary, use the interactive Move tool to fine-tune the position.

6. If necessary, click the Deselect All button .

Replace the surfaces on the LITE layers with the block LITE_EXTERIOR2

When a modeling program supports a block structure, the most efficient way to create luminaires is to create blocks, which are easily turned into luminaires in Lightscape. In modeling packages that do not support a block structure, however, the best technique is to create a small surface as a placeholder for each luminaire. The surface can be replaced by a luminaire within Lightscape.

1. Turn on all layers that start with the string "LITE":
 - a) Right-click in the Layers table, then choose All Off.
 - b) Right-click in the Layers table, then choose Select Pattern.
 - c) Type LITE* in the Pattern box and click Select.



Note: This is a case-sensitive text box. The asterisk is a wildcard character.

Thirteen layers beginning with the string “LITE” are selected. Each layer contains a triangular surface as a placeholder for a luminaire.

d) Right-click in the Layers table, then choose On.

2. Select the triangular placeholders:

a) Click the Select button , then click the Surface button .

b) Click the Select All button .

The thirteen surfaces are highlighted, though they may not be visible from the current view.

3. Create a view of the courtyard where you will be able to see the new luminaires:

a) Turn all layers on.

b) Adjust the view so you are inside the courtyard.

For example, set the viewer position to [37.5, -37.5, 5.2], the focus point to [15.4, -32.5, 5.2], the near clip plane to about 0, the far clip plane to about 19, and the field of view to 86 degrees.

c) Save this view as Courtyard.vw.

Later in this lesson, you use this view for an image of the courtyard at night.

4. Replace the selected placeholder surfaces with luminaires:

a) Right-click in the Graphic window, then choose Replace With Block/Luminaire.

b) Select LITE_EXTERIOR2 from the block/luminaire list.

c) Click OK.

The selected surfaces are replaced with luminaires.

5. Select the new luminaires:

a) Click the Select button , then click the Luminaire button .

b) Click the Select All button .

The luminaires are highlighted in red in the Graphic window.

c) Click the Deselect All button .

Setting the Local Processing Parameters

Local processing parameters take priority over the global settings for selected surfaces. For example, you can make the meshing more accurate in certain areas and less accurate in other areas of the model.

The exterior model has some large surfaces (like the roof) that reflect light up to the sky but do not reflect light on any surface in the model. Where these surfaces contribute negligibly to the scene, you can set them to be largely ignored in the solution. Modifying the properties of these surfaces can speed up the radiosity processing.

The processing parameters you can control locally are:

- **Occluding:** controls whether a surface casts a shadow or light goes straight through it unaffected.
- **Receiving:** controls whether light reaching a surface is recorded in its radiosity mesh. You can turn Receiving off to save computation time on a surface that is self-emitting (for example, an area light source).
- **Reflecting:** controls whether a surface reflects light incident on it back into the environment.
- **No Mesh:** controls whether a surface uses mesh subdivision. If you turn No Mesh on, the surface is not meshed at all. Only the original vertices are used in the radiosity calculation. This is useful for surfaces that are self-emitting or transparent (like glass windows).
- **Mesh Resolution:** scales the minimum and maximum mesh spacing for the selected surfaces. A value greater than 1 improves the quality of a radiosity mesh, and a value less than 1 decreases the number of mesh elements in the surface.

For example, a value of 2 decreases the size of the Receiver Mesh Spacing Min and Max values for this surface by half, thereby potentially quadrupling the number of mesh elements on this surface. A value of 0.5 increases the size of the Receiver Mesh Spacing Min and Max values for this surface by two, thereby potentially decreasing the number of mesh elements on this surface by 75%.

Remove meshing for all surfaces on the TREE layers

In this exercise, you prevent any adaptive subdivision meshing from taking place on the tree surfaces to decrease the total number of mesh elements in the model.

1. Turn on all layers that start with the string “TREE”:

- a) Right-click in the Layers table, then choose All Off.
- b) Right-click in the Layers table, then choose Select Pattern.
- c) Type TREE* in the Pattern box and click Select.

The thirteen TREE layers are selected.

- d) Right-click in the Layers table, then choose On.

2. View and select the tree surfaces:

- a) Click the View Extents button .
- b) Click the Select button , then click the Surface button .
- c) Click the Select All button .

All the surfaces are highlighted.

3. Remove meshing for the selected surfaces:

- a) Right-click in the Graphic window, then choose Surface Processing.

The Surface Processing dialog appears.

- b) Enable No Mesh.

Notice that the Mesh Resolution controls are disabled.

- c) Click OK.

Set all surfaces on the TREE and ROOF layers to be nonreflecting

In this exercise, you prevent Lightscape from computing the light energy that is reflected from the tree and roof surfaces. The light that would be reflected from these surfaces is not important for the rest of the model, so you can save some computation time.

1. Turn on the ROOF layers:

- a) Select EXTROOF1 and EXTROOF2 in the Layers table.
- b) Right-click in the Layers table, then choose On.

2. Add the roof surfaces to the selection set:

- a) Orbit the model to see the roof surfaces.
- b) Click the Select All button .

The roof and trees are selected.

3. Make the selected surfaces nonreflecting:

- a) Right-click in the Graphic window, then choose Surface Processing.

The Surface Processing dialog appears.

- b) Disable Reflecting.

- c) Click OK.

4. Click the Deselect All button .

Minimize the amount of meshing on the ROOF layers

Assuming that you normally view the building from eye level, the roof does not need to be meshed to the same level of detail as the rest of the building.

1. Turn off all layers except EXTROOF1 and EXTROOF2.

2. Click the Select All button .

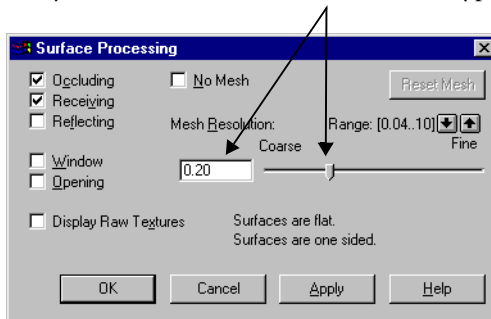
Only the roof surfaces are selected.

3. Decrease the meshing for the selected surfaces:

- a) Right-click in the Graphic window, then choose Surface Processing.

The Surface Processing dialog appears.

- b) Adjust the Mesh Resolution slider to 0.2 or type 0.2 in the adjacent box.



Hint: The Up and Down buttons adjust the range of the Mesh Resolution slider by a factor of 10.

- c) Click OK.

4. Click the Deselect All button .

Set the surfaces on the GROUND layer to be non-occluding

In this exercise, you prevent Lightscape from computing whether the ground surfaces create shadows on other surfaces. Because there are no surfaces below the ground onto which to cast shadows, there will be no shadows, so again you can save some computation time.

1. Turn on the GROUND layer:

- a) Turn off all layers.
- b) Double-click GROUND in the Layers table.

2. Click the Select All button .

Only the ground surfaces are selected.

3. Make the selected surfaces non-occluding:

- a) Right-click in the Graphic window, then choose Surface Processing.

The Surface Processing dialog appears.

- b) Disable Occluding.

- c) Click OK.

While the GROUND surfaces are still selected, you can align the texture map on them.

4. Click the Textures button to view the current texture alignment.

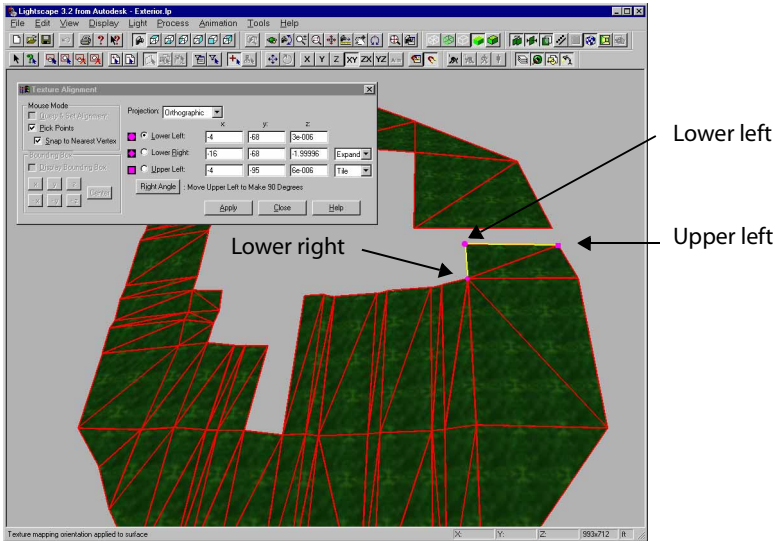
Notice that the texture starts again on every surface.

5. Right-click in the Graphic window, then choose Texture Alignment.

The Texture Alignment dialog appears.

6. Enable Pick Points in the Mouse Selection Mode group box, and then enable Snap To Nearest Vertex (so that you do not have to be precise when picking the points).

7. Pick three points on one of the surfaces (lower left, lower right, and upper left).



After you pick a point, the radio button for the next one is automatically selected.

Hint: The icons next to the radio buttons are used in the Graphic window to identify the three different points. For example, the round dot shows the lower-left point.

8. Set the mapping modes:

- From the Lower Right list, select Tile.
- From the Upper Left list, select Tile.

9. Click Apply, then click Close.

The ground texture now appears to be continuous.

10. Click the Deselect All button .

11. Turn on all the layers.

12. Show the view of the model named *Lower_Elevation.vw*.

13. Click the Textures button  to turn the textures off.

Setting the Global Processing Parameters

Processing parameters control the quality of the light distribution and the time it takes to process the model. In the previous exercises, you changed local parameters in the Surface Processing dialog. By default, surfaces are occluding, receiving, and reflecting, and the Mesh Resolution equals 1.

Global parameters are set in the Process Parameters dialog. When Mesh Resolution (in the Surface Processing dialog) equals 1, the meshing parameters for the selected surface are the same as the Mesh Spacing Min. and Max. parameters (in the Process Parameters dialog).

In this exercise, you change the global processing parameters.

Set the processing parameters

1. Choose Process | Parameters.

The Process Parameters dialog appears.

The numerical values in the various boxes of the dialog control the quality and speed of the Lightscape radiosity process. For this model, you will set the parameters individually instead of using the wizard.

2. Set the Receiver Group parameters:

- a) Set Mesh Spacing Min. to 0.3 and Mesh Spacing Max. to 3.0.

These values were calculated using the guidelines explained in Lesson 3. (The dimensions of the courtyard were used because the night solution does not show the entire model.)

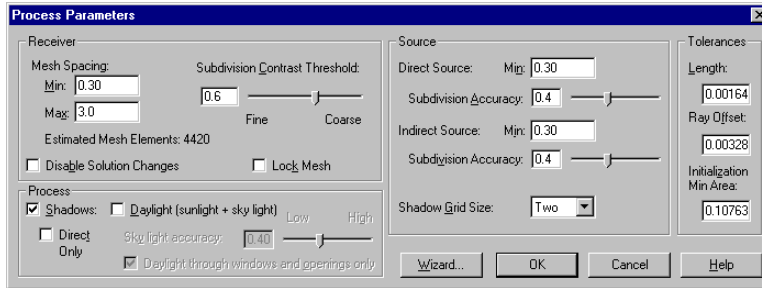
- b) Set Subdivision Contrast Threshold to 0.6.

3. Set the Source Group parameters:

- a) Set Direct Source Min. and Indirect Source Min. to 0.3.
- b) Set Subdivision Accuracy for Direct Source and Indirect Source to 0.4.
- c) Set Shadow Grid Size to Two.

4. In the Process group box, make sure Daylight (sunlight + skylight) is disabled.

You are creating a night solution first, so you do not consider daylighting.



As an example of the relationship between global and local parameters, consider the roof surfaces. Now Mesh Spacing Min. equals 0.3 and the Mesh Spacing Max. equals 3.0. Because you set the Mesh Resolution for the roof surfaces to 0.2 (in a previous exercise), the Mesh Spacing Min. used for the roof surfaces is 1.5 and the Mesh Spacing Max. is 15.

5. Click OK to close the Process Parameters dialog.

Save your work

1. Choose File | Save As.

The Save As dialog appears.

2. Change the path to *C:\Program Files\Lightscape\Projects\Lesson4*.
3. Type *Exterior.lp* in the File Name box.
4. Click Save.

Stage 3: Creating the Radiosity Solution

In this section, you perform the following tasks:

- Creating a Night Solution for the Courtyard
- Creating a Daytime Solution for the Full Model

Creating a Night Solution for the Courtyard

When you set the global processing parameters, you turned Daylight off. Another necessary setting for a night solution is to change the background color to black. You can also turn off some layers to save processing time.

Initiate the Preparation model, then turn off some layers

By leaving parts of the model out of a solution, you can create multiple solution options. You can also use this technique to quickly test illumination levels with only specific parts of the model displayed.

In this model, the only illumination at night is in the courtyard so you can turn off all the layers that cannot be seen from the courtyard: the trees, the ground, and the windows.

1. Click the Initiate button .

Note: It is important to initiate the model before you turn off layers because layers that are turned off are excluded from the Solution file. You still want these layers in the Solution file because you turn them back on to create the daytime solution later in this lesson.

2. To see the model, click the Hidden Line button .
3. Double-click GROUND in the Layers table to turn off this layer.
4. Turn off all layers that start with the string “TREE”:
 - a) Right-click in the Layers table, then choose Select Pattern.
 - b) Type TREE* in the Pattern box and click Select.

The thirteen TREE layers are selected.

 - c) Right-click in the Layers table, then choose Off.

5. Turn off all layers that start with the string “WIND”:
 - a) Right-click in the Layers table, then choose Select Pattern.
 - b) Type WIND* in the Pattern box and click Select.

The eleven window layers are selected.

- c) Right-click in the Layers table, then choose Off.

6. Show the view of the model named *Courtyard.vw*.

Save a Layer State file

When you are working with many layers that are being switched on and off, it can help to save the state of the layers in a separate file called a Layer State (.lay) file. It records all the layer names and their current state (whether on or off).

1. Right-click in the Layers table, then choose Save State.

The Save As dialog appears.

2. Change the path to *C:\Program Files\Lightscape\Projects\Lesson4*.
3. Type *Noland.lay* in the File Name box.
4. Click Save.

Note: To return to this layer state, simply right-click in the Layers table, then choose Load State. The name of the Layer State file also appears in the Recently Used list at the bottom of the menu.

Adjust the brightness, contrast, and background color

Brightness adjusts the color range of the displayed image for your monitor. It does not affect the actual lighting levels in the model. Some scenes, such as exterior models, contain extreme differences between the brightest and dimmest regions of the environment. You can use brightness and contrast to achieve a more appealing balance of light levels in such cases.

1. Choose File | Properties.

The Document Properties dialog appears.

2. On the Display panel, set Brightness to 75.

3. Set Contrast to 55.

Note: Brightness and contrast affect the display of Solution files only.

4. To change the background color, click the Colors tab.

You need a black background for the night solution.

5. Set color values by typing numbers directly in the boxes (on the left of the dialog):

a) Set Background H (Hue) to 0.

b) Set Background S (Saturation) to 0.

c) Set Background V (Value) to 0.

6. Click OK.

Process the radiosity solution

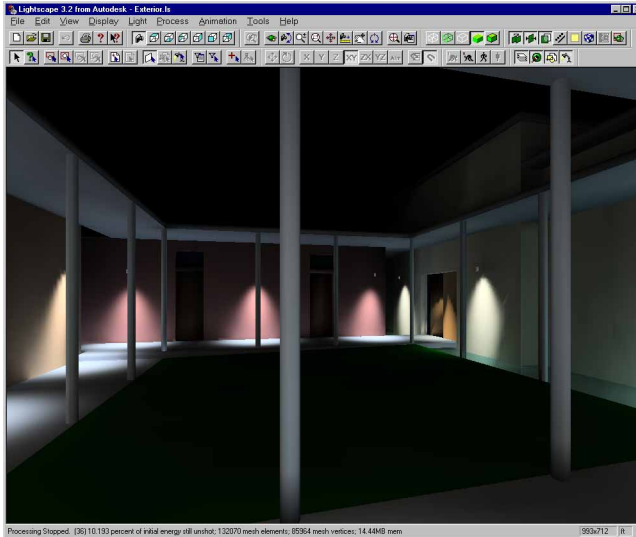
1. Click the Solid button .

2. Click the Go button .

Notice the various statistics shown on the status bar such as the number of iterations and percentage of the total energy distributed.

3. After about 35 iterations, click the Stop button .

The processing stops after the current iteration finishes. The result should look like this.



Save the night solution

1. Choose File | Save As.

The Save As dialog appears.

2. Change the path to *C:\Program Files\Lightscape\Projects\Lesson4*.
3. Type *Night.ls* in the File Name box.
4. Click Save.

Reset the solution and turn on all the layers

1. Click the Reset button .

When the warning appears, click Yes to continue resetting the radiosity solution. The model appears dark because no energy is distributed.

2. To see the model, click the Wireframe button .

3. To turn all the layers on, right-click in the Layers table, then choose All On.
4. Show the view of the model named *Lower_Elevation.vw*.

Creating a Daytime Solution for the Full Model

For the exterior model, you need to indicate the sun position and set the exterior daylight options. You can set the sun location by specifying latitude, longitude, date, and time of day, or you can set the sun location directly.

Lightscape calculates both the direct sunlight and diffuse skylight contributions. The diffuse skylight contribution considers the entire sky when performing this calculation (for example, the light on south-facing surfaces differs from the light on north-facing surfaces). This complex calculation takes more time, so controlling its accuracy is important.

To capture the sharp shadow boundaries from the sun on a large ground surface, you need an extremely fine mesh. Because this mesh contains a prohibitive number of mesh elements, a better solution is to use a crude mesh and then ray trace the sunlight when you render images. The crude mesh is sufficient for calculating the interreflection of light, and the ray tracer handles the fine shadow detail. To ray trace the daylight, you need to turn on the Ray Trace Direct Illumination option in the daylight processing options.

Set the processing parameters to consider daylight

1. Choose Process | Parameters.

The Process Parameters dialog appears.

2. Click the Wizard button to start the Process Parameters Wizard.
3. Select 1 for the lowest quality level, then click Next to show the next page of the wizard.
4. Select Yes (so daylight is considered in this model), then click the radio button for “The model is an exterior of a building or an object”.
5. Click Finish to update the processing parameters based on the selections made in the wizard.

Notice the change in the numerical values in the various boxes of the dialog. Also notice that “Daylight (sunlight + sky light)” is enabled and the Sky light accuracy is set at 0.2.

6. Click OK to close the Process Parameters dialog.

Set the daylight parameters

In this exercise, you set the sun location directly by using the Direct Control panel. Later in this lesson, you use the other method, which is specifying latitude, longitude, date, and time of day.

1. Choose Light | Daylight.

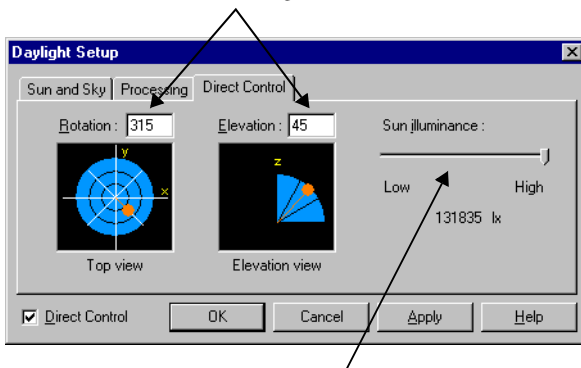
The Daylight Setup dialog appears.

2. Enable Direct Control (beside the OK button).

In the dialog, the Direct Control panel replaces the Place and Time panels.

3. Set the sun location:

- a) Click the Direct Control tab.
- b) Set the Rotation to 315 degrees and the Elevation to 45 degrees.



Hint: You can also set the Rotation and Elevation by clicking in the Top view and Elevation view, respectively.

- c) Adjust the Sun Illuminance slider to full brightness.

4. Check the Processing settings:

a) Click the Processing tab.

b) Make sure that all options except “Ray Trace Direct Illumination” in the Sky light group box are enabled.



With these settings, the sunlight and skylight are both calculated and displayed in the radiosity solution, and only the direct sunlight is ray traced when you render an image.

5. Click OK to update the sun parameters and close the dialog.

Save your work up to this point

1. Choose File | Save As.

2. Change the path to *C:\Program Files\Lightscape\Projects\Lesson4*.

3. Type *Day.ls* in the File Name box.

4. Click Save.

Process the solution

1. Before starting the processing, change the background color to blue.

For example, set Background H (Hue) to 211, Background S (Saturation) to 0.63, and Background V (Value) to 0.88. To review the two ways to set background color values, see “Change the background color” on page 56.

2. Click the Solid button .

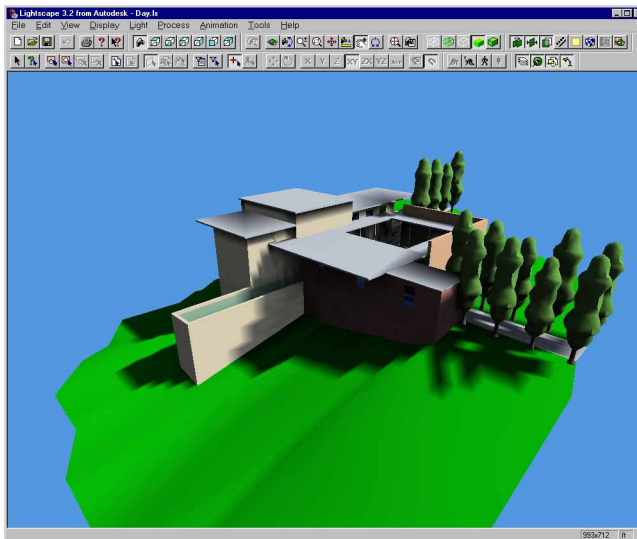
3. Click the Go button .

The first iteration, which calculates the direct sunlight, should be fairly quick. The next iteration, which calculates the skylight contribution, takes a bit longer.

4. After about 15 iterations, click the Stop button .

Note: Because the direct contribution from the sun is extremely large compared to the light bounced from the surfaces in the model, there is little visible change as the process continues to run.

The processing stops after the current iteration finishes. The result is as follows.



Notice that the shadows on the ground look very coarse; these will be cleaned up by using the ray tracer.

5. To save your work, choose File | Save.

Stage 4: Fine-Tuning the Solution

You can interrupt the processing of the radiosity solution at any time to alter or fine-tune the model's appearance. You can also create a series of radiosity solutions or ray-traced images.

Creating Sun Studies

A common task for an architect is to produce a sun study for a proposed building. Typically, the study includes images at different dates and times. Because sun studies do not typically require the subtle interreflections of radiosity, you can create these images more efficiently by using only the ray tracer. In this exercise, you create two images of the exterior model showing the shadows at the winter and summer solstices.

Set the daylight parameters for the summer solstice

The summer solstice is the time at which the sun is furthest north from the equator, about June 21 in the northern hemisphere. You set the sun location by specifying latitude, longitude, date, and time of day.

1. Save the Solution file as Summer.ls.
2. Switch to the Top projection.

This projection helps you set the North direction in the daylight settings.

3. Click the View Extents button .

4. Choose Light | Daylight.

The Daylight Setup dialog appears.

5. Disable Direct Control (beside the OK button).

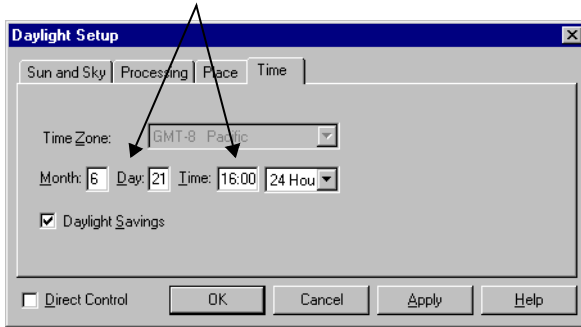
In the dialog, the Place and Time panels replace the Direct Control panel.

6. Click the Place tab.
7. Type 90 in the North box, and select San Francisco from the Location list.

8. Set the date and time:

a) Click the Time tab.

b) Set the date to June 21 and the time to 16:00.



c) Enable Daylight Savings.

9. Click the Processing tab.

10. Enable Ray Trace Direct Illumination in the Sky light group box.

All the options on this panel should now be enabled.

11. Click OK to update the sun parameters and close the dialog.

Render the summer solstice image

1. Click the Reset button .

When the warning appears, click Yes to continue resetting the radiosity solution. The model appears dark because no energy is distributed.

2. Choose Process | Parameters.

3. In the Process group box, adjust the Sky Light Accuracy to 0.1, then click OK.

4. Choose File | Render.

The Rendering dialog appears.

5. Set the Rendering options:

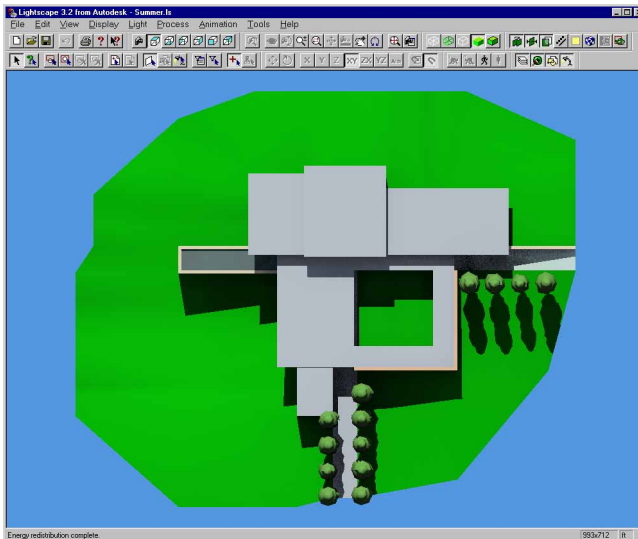
- a) Select Targa (TGA) from the Format list.
- b) Type Summer.tga in the Name box.
- c) Select Full Window from the Resolution list.
- d) Enable Ray Tracing.
- e) Enable Ray Trace Direct Illumination.

This will create better direct shadows by ray tracing the sunlight.

- f) Make sure the Ray Bounces box is set to 10.
- g) Click OK.

6. A warning appears because the lights have not been shot. Click Yes to continue.

Notice where the shadows are cast.



7. Choose File | Save.

You can render other images from this Solution file later.

Set the daylight parameters for the winter solstice

The winter solstice is the time at which the sun is furthest south from the equator, about December 22 in the northern hemisphere. In this exercise, you only change the date.

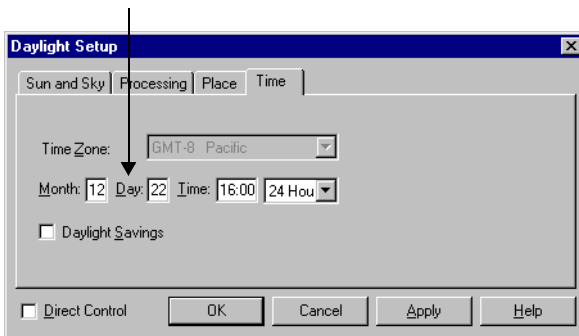
1. Choose Light | Daylight.

The Daylight Setup dialog appears.

2. Set the date:

a) Click the Time tab.

b) Set the date to December 22 and leave the time set to 16:00.



c) Disable Daylight Savings.

3. Click OK to update the sun parameters and close the dialog.

Render the winter solstice image

1. Choose Process | Parameters.
2. Check that the Sky Light Accuracy is set to 0.1, then click OK.
3. Choose File | Render.

The Rendering dialog appears.

4. Set the Rendering options:

- a)** Select Targa (TGA) from the Format list.
- b)** Type Winter.tga in the Name box.
- c)** Select Full Window from the Resolution list.
- d)** Enable Ray Tracing.
- e)** Enable Ray Trace Direct Illumination.

This will create better direct shadows by ray tracing the sunlight.

- f)** Make sure the Ray Bounces box is set to 10.
- g)** Click OK.

5. A warning appears because the lights have not been shot. Click Yes to continue.

Notice where the shadows are cast.

6. Save this file as Winter.ls.

You can render other images from this Solution file later.

Stage 5: Outputting the Results

In this section, you perform the following tasks:

- Ray Tracing a Region
- Playing an Existing Animation

Ray Tracing a Region

You can preview the effects of ray tracing for a selected region on the screen. When only part of an image contains interesting ray trace effects, you should check that the materials are displayed correctly when rendered.

Ray trace a part of the screen to preview a region of the final image

1. Open the *Day.ls* Solution file.
2. Load the view named *Lower_Elevation.vw*.

Hint: To see the intensity mapping effect, you should ray trace a large area of the ground. To clearly see the bump mapping effects (on the walls and water), the view should be relatively close to the building.

3. Click the Textures button .

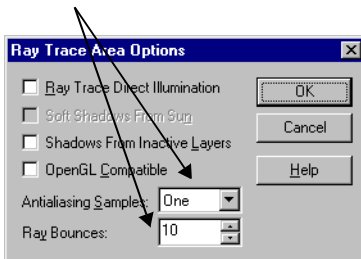
The textures maps (for the ground and trees) load into memory.

4. Check the Ray Trace Area options:

a) Choose Display | Ray Trace Area Options.

The Ray Trace Area Options dialog appears.

b) Make sure that Antialiasing Samples is set to One, Ray Bounces is set to 10, and all other options are disabled.



c) Click OK to close the Ray Trace Area Options dialog.

5. Click the Ray Trace Area button .

This sets the function of the left mouse button to a special selection mode used for Ray Trace Area.

6. Drag in the Graphic window to select the area to be ray traced.

For example, draw a rectangle that includes some water and a large area of the ground to show the bump and intensity mapping effects.

7. Change the Ray Trace Area options:

a) Choose Display | Ray Trace Area Options.

The Ray Trace Area Options dialog appears.

b) Enable Ray Trace Direct Illumination.

The direct sunlight is ray traced when you render an image because this option and the Ray Trace Direct Illumination option in the Daylight Setup dialog are turned on. You ray trace the daylight to obtain better quality shadows.

c) Click OK.

8. Drag in the Graphic window to select the area to be ray traced.

For example, draw a rectangle that includes some of the ground near the trees.

9. After you release the mouse button, a warning appears. Click OK to confirm that some additional processing can occur before the ray tracing is started.

Note: This warning only appears when Ray Trace Direct Illumination is enabled.

The selected region is ray traced with the bump and intensity effects as well as ray-traced sun shadows.

10. To clear the current ray-traced image, choose Display | Refresh, press F5, or change the view of the model.

Create a ray-traced image

1. Create another view for an exterior image:

a) Adjust the view so that you are looking at the upper elevation.

For example, choose View | Setup and set the viewer position to [93.7, -61, 66], the focus point to [14, -27, 18], the near clip plane to about 0, and the field of view to 67 degrees.

Hint: For an exterior image, you should establish a horizon. For example, pick a view with objects (like land or part of the model) along the entire horizon.

b) Save this view as Upper_Elevation.vw.

2. Choose File | Render.

The Rendering dialog appears.

3. Set the Rendering options:

a) Select Targa (TGA) from the Format list.

b) Type RayExterior.tga in the Name box.

c) Select Two from the Antialiasing Samples list.

Antialiasing over-samples pixels to remove “jaggies” along the edges of objects.

d) Select Full Window from the Resolution list.

e) Enable Ray Tracing.

f) Enable Ray Trace Direct Illumination.

This will create better direct shadows by ray tracing the sunlight.

g) Make sure the Ray Bounces box is set to 10.

h) Click OK and confirm the additional processing.

The ray-traced image shown in the Graphic window should resemble the image at the beginning of this lesson. You can use the LVu utility to view the *RayExterior.tga* file.

Hint: To get a good ray-traced image, you may want to adjust the Brightness before rendering.

Playing an Existing Animation

A Lightscape animation file named *Exterior.la* was created for the exterior model. You can load the file to play the animation.

Load an animation and play it

1. Choose Animation | Open.

The Open dialog appears.

2. Change the path to *C:\Program Files\Lightscape\Projects\Lesson4*.

3. Double-click *Exterior.la*.

The animation file loads into memory.

4. Click the Wireframe button .

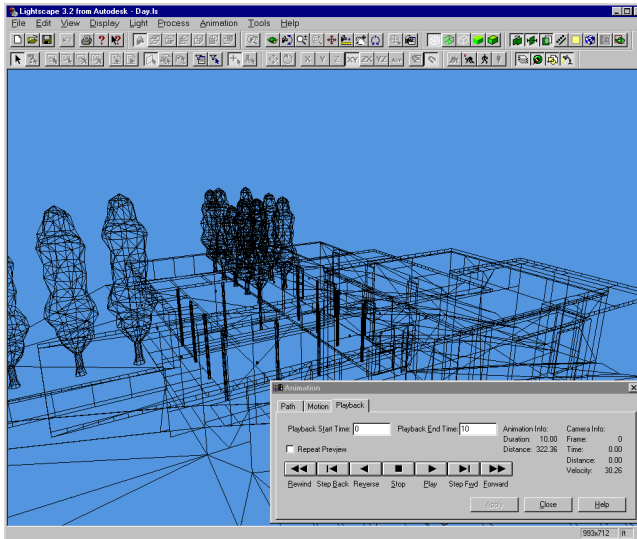
5. Play the animation in Wireframe mode:

- a) Choose Animation | Edit.

The Animation dialog appears. When the Path panel is showing, the Graphic window is split into four views. When the Motion panel is showing, the Graphic window is split into two views and a Camera Speed graph. For more information on these views, see “Lightscape Animation” on page 134.

b) Click the Playback tab.

The Graphic window shows the camera's view at the current time in the animation.



c) If necessary, move the Animation dialog out of the way.

d) Click the Play button.

The animation plays in the Graphic window.

6. Play the animation in Solid mode:

a) Click the Solid button .

By default, the time in the Playback End Time box equals the duration of the animation (from the Motion panel).

Note: If the time equals 1, enter -1 in the Playback End Time box and click Apply. The correct value will appear in the Playback End Time box.

b) Click the Rewind button.

c) Click the Play button.

Note that the animation is smoother in Wireframe mode. This is because more frames per second can be displayed in non-solid modes.

d) Click Close when you are done viewing the animation.

Finish your session in Lightscape

1. Choose File | Save to save this file.
2. Choose File | Exit.

Summary

Congratulations! You have reached the end of Lesson 4. At this point, you should be able to:

- Import AutoCAD (DWG) files
- Assign materials to groups of surfaces
- Assign bump mapping and intensity mapping to materials
- Select layers by pattern and work with layer states
- Set parameters for lighting an exterior model
- Set local processing parameters to control the quality (and speed) of a radiosity solution
- Create radiosity solutions of selected portions of a model

5

Working with Solution Files

A good approach for working with a large project is to break it into several smaller models. After you create radiosity solutions for the models, you can join the Solution files together. In this lesson, you merge the Solution files created in the first four lessons. The new model includes the gallery, sunroom, exterior model, and lounge.

To increase the interactivity of models, you can convert surfaces to textures. In this lesson, you optimize the Gallery model by converting selected radiosity mesh elements into texture maps. Lightscape can export to panoramic and VRML formats. You will export the completed sunroom model in both of these formats.

In this lesson:

- Join the Solution files from the first four lessons
- Convert mesh to texture for the Gallery model
- Export a Solution file in panoramic and VRML formats

Merging Files

In Lesson 3, you merged a DXF file with a Preparation model. You can merge Preparation models together or Solution models together. In this exercise, you merge the Solution files for the gallery, sunroom, lounge, and exterior model.

Note: Solution files for the first four lessons are included in the Lesson5 folder. You can use these files or the files that you created. For example, you can open the *Lesson1.ls* file or your *Gallery.ls* file.

Load a Solution file

You start by opening one of the files that you want to join together. It does not matter which one you pick first.

1. Start Lightscape if the application is not already running.

2. Choose File | Open.

The Open dialog appears.

3. Select Solution Files (*.ls) from the Files of Type list.

4. Change the path to *C:\Program Files\Lightscape\Projects\Lesson5*.

5. Double-click *Lesson1.ls*.

This is a similar Solution file to the one you created during Lesson 1, “The Gallery.”

6. Click the Solid button .

The model appears in Solid mode.

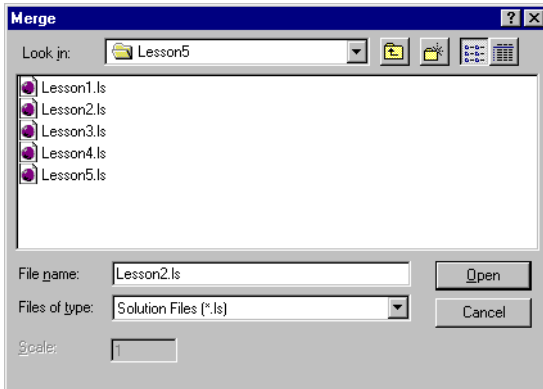
7. Choose File | Save As, type Lesson5.ls in the File Name box, and click Save.

Merge the Lesson 2 Solution file

When you merge a file, the new geometry is added to the existing geometry in the model.

1. Choose File | Merge.

The Merge dialog appears.



2. Change the path to *C:\Program Files\Lightscape\Projects\Lesson5*.
3. Double-click *Lesson2.ls*.

This is a similar Solution file to the one you created during Lesson 2, “The Sunroom.”

4. Look at the model:

a) Click the View Extents button .

b) Orbit around the model.

The gallery and sunroom both appear in the Graphic window.

Merge the Lesson 3 Solution file

1. Choose File | Merge.

The Merge dialog appears.

2. Change the path to *C:\Program Files\Lightscape\Projects\Lesson5*.

3. Double-click *Lesson3.ls*.

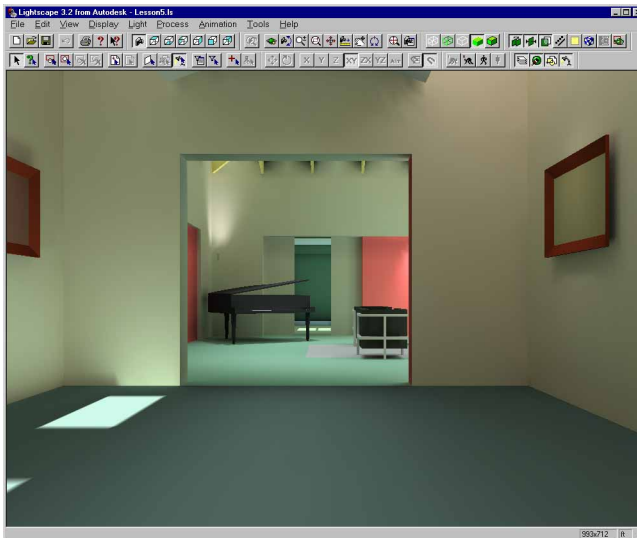
This is a similar Solution file to the one you created during Lesson 3, “The Lounge.” A message appears asking if you want to save changes to the model.

4. Click Yes to confirm saving the file *Lesson5.ls*.

The lounge appears between the gallery and sunroom.

5. Look at the inside of the model:

- a) Adjust the view so that you are inside the building.



For example, set the viewer position to $[32, -3, 3]$, the focus point to $[0, -3, 3]$, and the near clip plane to about 0.

Note: These values are in feet. If you need to change the units of the model to feet, choose File | Properties and change the Length unit on the Units panel.

- b) Save this view as *Inside_Building.vw*.

Merge the Lesson 4 Solution file

The last file to add to the model is the exterior.

1. Choose File | Merge.

The Merge dialog appears.

2. Change the path to *C:\Program Files\Lightscape\Projects\Lesson5*.
3. Double-click *Lesson4.ls*.

This is a similar Solution file to the one you created during Lesson 4, “The Exterior Model.” A message appears asking if you want to save changes to the model.

4. Click Yes to confirm saving the file *Lesson5.ls*.

5. Look at the entire model:

- a) Click the View Extents button .

- b) Orbit around the model.

The exterior model appears extremely bright because the display settings are currently adjusted for the lower lighting levels of an interior model. This is like the situation of having a real camera adjusted for an interior exposure and then using the same settings outside. You would need to adjust your camera aperture to accommodate the brighter light; in Lightscape, you would need to adjust the brightness setting to accommodate the various lighting levels.

6. Adjust the brightness:

- a) Choose File | Properties.

The Document Properties dialog appears.

- b) On the Display panel, change the Brightness to 15 and click Apply.

This displays the exterior more appropriately.

- c) Open the view named *Inside_Building.vw*.

The interior portions of the model are displayed darker than usual.

- d) Set the Brightness back to 55 to correct the problem.

- e) Click OK to close the Document Properties dialog.

7. Choose File | Save to save the Solution file.

Using the Mesh to Texture Tool

To obtain real-time display of Lightscape models, it is important to minimize the number of polygons in your scene. For example, if your CPU can display 100,000 rendered polygons a second, and you want to show a smooth animation (at 15 frames per second, for example), the model should contain no more than 6,666 polygons ($100,000 / 15$).

By carefully setting global and local processing parameters, you can achieve this in some models. A second method, converting polygonal radiosity meshes to texture maps, is also available in Lightscape. This technique can be particularly useful for creating an interactive 3D environment for use in applications such as games, web sites (VRML), and virtual sets.

With this technique, the lighting information stored in the radiosity mesh is converted into a texture map. The texture map is then applied to the surface and the underlying mesh is removed from the model.

Note: The texture maps can also be exported to display radiosity lighting data in modeling applications and rendering engines that cannot display vertex-colored mesh data.

There are three methods available for converting selected geometry to textures:

- Creating a separate texture for each selected surface (Texture per Surface)
- Taking existing texture maps and remaking them with the radiosity lighting data included as part of the texture (Relight Textures)
- Creating a single texture from a group of selected surfaces (Project all Textures)

With this last option, you can further optimize your model by deleting some of the original geometry from it. For example, if a wall surface has a few windows in it and several pictures hanging on it, you can project all this geometry as decals on the single wall surface.

As you use the Mesh to Texture tool, you create two selection sets:

- The first selection set is only used for the third conversion method. It contains geometry that will be projected onto other surfaces. This geometry can be deleted after it is projected.
- The second selection set contains the geometry where the textures will be placed. It generally contains all the geometry you want to map into the texture, less the geometry in the first selection set, if any.

Load a Solution file

You perform this exercise with a file that has already been created.

1. Choose File | Open.

The Open dialog appears.

2. Select Solution Files (*.ls) from the Files of Type list.

3. Change the path to *C:\Program Files\Lightscape\Projects\Lesson5*.

4. Double-click *Lesson1.ls*.

This is a similar Solution file to the one you created during Lesson 1, “The Gallery.”

Note: You should use the *Lesson1.ls* file instead of your *Gallery.ls* file because a few textures have been set up for this exercise.

Note the number of polygonal mesh elements in the model

1. Click the Outlined button .

The mesh appears on the surfaces. Notice how simple surfaces, such as the wall in the alcove, have been subdivided to capture the lighting in the model.

2. Choose Process | Statistics.

The model statistics appear on the status bar. Write down the number of mesh elements for future reference.

3. Click the Textures button .

4. Click the Solid button .

Create two texture maps to replace the floor and alcove wall

In this exercise, you create a separate texture map for each surface in a group of surfaces.

1. Choose View | Display Original View to reset the view.

2. Choose Tools | Mesh to Texture.

The Mesh To Texture wizard appears.

3. Select the “Convert each surface to a single texture per surface” radio button, then click Next.

This conversion method makes a single texture for each surface that is selected.

4. Make sure that the first selection set is empty:

Note: The Select Projected Geometry page is where you select the surfaces to be projected (as decals) onto the surfaces in the second selection set. Typically, you delete these surfaces from the model after the Mesh to Texture conversion process. For this exercise, leave the selection set blank.

- a) If necessary, click the Deselect All button .

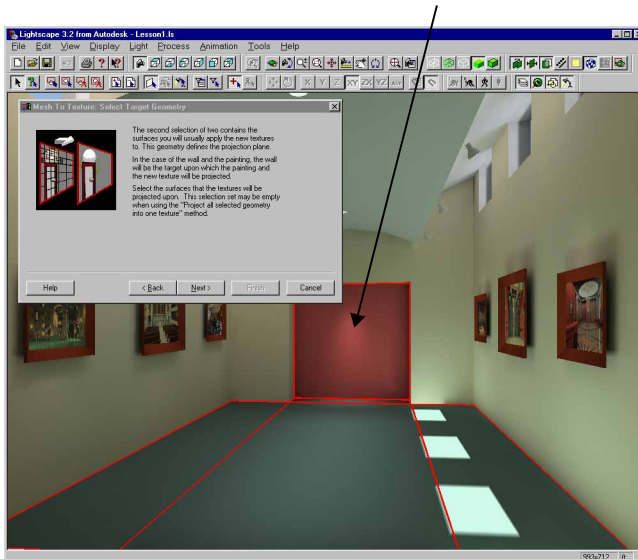
Hint: When the selection set is empty, both the Deselect All menu item and the toolbar button are disabled.

- b) Click Next.

5. Select the geometry for the second selection set:

Note: The Select Target Geometry page is where you select the surfaces on which textures will be placed.

- a) Turn on the Accumulate Pick feature.
- b) Click the Select button , then click the Surface button .
- c) Pick a point on the alcove wall to select this surface.



- d) Pick a point on the floor surface.
- e) Click Next.

6. On the Use Existing Texture Filenames page, make sure that “Use existing texture filenames” is disabled, then click Next.

Note: You can only use existing filenames when the floor already has a texture map applied to it.

7. Set the options on the Texture Output Information page:

- a) Select JPEG (JPG) from the Format list.

- b) Type Floor_Alcove.jpg in the New Textures Base Name box.

When you click Finish in step 9, two textures named *Floor_Alcove000.jpg* and *Floor_Alcove001.jpg* will be created.

- c) Make sure that the Manually Size radio button is selected, then type 256 in the Width box and 128 in the Height box. (The floor and alcove wall are about twice as wide as they are long.)

- d) Click Next.

8. Set the options on the Rendering Options page:

- a) Type 0 in the Ray Bounces box.

Typically, you want to avoid rendering specular reflections because they are view dependent and not suitable for 3D interactive applications.

- b) Enable Pad Texture Edge.

This option eliminates potential artifacts around the edges of textures by filling in all the pixels in the texture that do not lie on the target geometry with pixels of a similar color.

- c) Click Next.

9. Set the options on the Replace/Delete page:

- a) Enable “Replace textures on target geometry”.

This places the texture maps on the surfaces.

- b) Enable “Reset the mesh on the target geometry”.

This removes the radiosity mesh from the floor and alcove wall.

- c) If necessary, disable “Delete the projected geometry from the model”.

- d) Click Finish to create the texture maps.

Two new materials are added to the Materials table and the surfaces are replaced with texture maps. Before you used the Mesh to Texture tool, the floor surface used the FLOOR material and the alcove used the WOODPANEL material. The names of the new materials are:

- M2T WOODPANEL->Floor_Alcove000.jpg
- M2T FLOOR->Floor_Alcove001.jpg

Note: M2T means “Mesh to Texture”; and the numbers at the end of the material names are determined by the order in which you select the surfaces.

View the new textures and note the changed number of mesh elements

1. If necessary, click the Textures button to display the new textures. When Textures is off, the floor and alcove wall appear black.
2. Select M2T WOODPANEL->Floor_Alcove000.jpg in the Materials table and look at the texture in the material preview.

You can also double-click the material name to open the Material Properties dialog.

3. Select M2T FLOOR->Floor_Alcove001.jpg in the Materials table and look at the texture in the material preview.

Hint: Each material has a different texture map. You can quickly look at several texture maps by opening the image files in the LVu utility.

4. Click the Outlined button .

The mesh on the surfaces appears. Notice how the floor and alcove wall surfaces are no longer meshed.

5. Choose Process | Statistics.

The model statistics appear on the status bar. Write down the number of mesh elements and compare with the previous number. It should be much less.

6. Click the Solid button .

Create one texture map for all surfaces of the south wall

In this exercise, you map many surfaces into a single texture map, apply the single texture back onto the surfaces, and delete some of the geometry.

1. Turn off all layers except ROOM2_WALLS and ROOM2_PICTURES.

2. Orbit the view so that you can see all surfaces of the south wall.

You are viewing the south wall when the east wall (the one with the alcove) is to your left.

3. Choose Tools | Mesh to Texture.

The Mesh To Texture wizard appears.

4. Select the “Project all selected geometry into one texture” radio button, then click Next.

This conversion method makes a single texture from all the surfaces you select.

5. Select the geometry for the first selection set:

Note: The Select Projected Geometry page is where you select the surfaces to be projected (as decals) onto the surfaces in the second selection set. Typically, you delete these surfaces from the model after the Mesh to Texture conversion process. For this exercise, select the pictures and picture frames. These surfaces will be deleted from the model and an image of them pasted onto the wall surface.

a) If necessary, click the Deselect All button  to make sure that the selection set is empty.

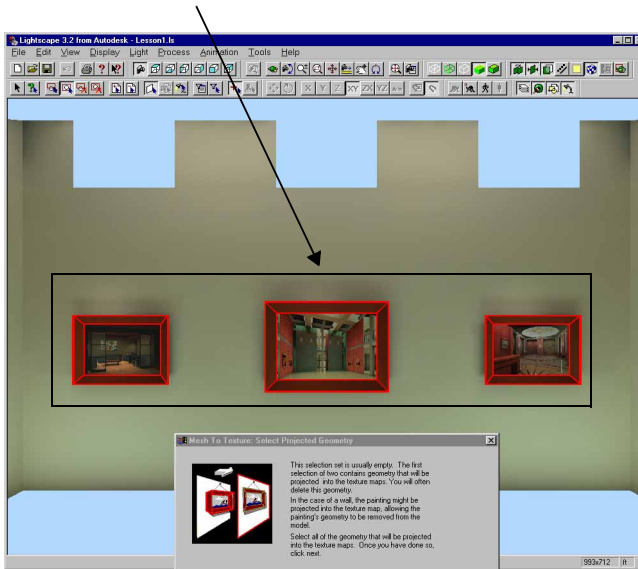
b) Click the Area All button .

c) Open the view named *Southwall.vw*.

The viewer position for this view is closer to the south wall to make selecting easier.

Note: When you select the surfaces using the Area All tool, be careful not to include surfaces from the pictures on the north wall in the selection set.

d) To select the surfaces of the pictures and the picture frames, drag a selection rectangle around the three pictures.



e) Click Next.

6. Select the geometry for the second selection set:

Note: The Select Target Geometry page is where you select the surfaces on which textures will be placed.

a) Click the Select button , then click the Surface button .

b) Select the five surfaces that make up the south wall.

Do not include the pictures and picture frames.

Hint: You can turn on the Accumulate Pick feature (or press and hold the Ctrl key) and select the surfaces one by one.

c) Click Next.

7. Set the options on the Select Projection Method page:

a) Click the Select button  if it is not still selected.

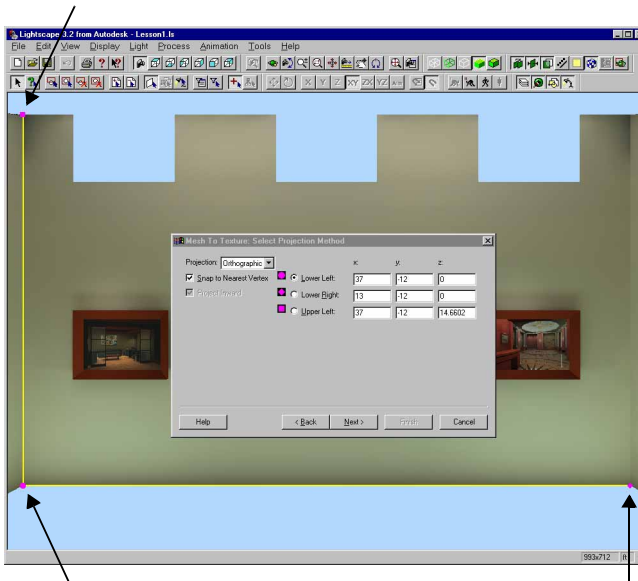
To pick the three points, you must be in Select mode.

b) Select Orthographic from the Projection list.

c) Enable Snap To Nearest Vertex so you do not have to be precise when picking the points.

d) Make sure that the Lower Left radio button is selected, then pick three points on the wall (lower left, lower right, and upper left).

Upper-left point



Lower-left point

Lower-right point

After you pick a point, the radio button for the next one is automatically selected.

Hint: The icons next to the radio buttons are used on the Graphic window to identify the three different points. For example, the round dot shows the lower-left point.

e) Click Next.

8. Set the options on the Texture Output Information page:

- a)** Select JPEG (JPG) from the Format list.
- b)** Type South.jpg in the New Textures Base Name box.

When you click Finish in step 10, a texture named *South000.jpg* will be created.

- c)** Make sure that the Manually Size radio button is selected, then type 512 in the Width box and 256 in the Height box.
- d)** Click Next.

9. Set the options on the Rendering Options page:

- a)** Type 0 in the Ray Bounces box.

Typically, you want to avoid rendering specular reflections because they are view dependent and not suitable for 3D interactive applications.

- b)** Enable Pad Texture Edge.

This option eliminates potential artifacts around the edges of textures by filling in all the pixels in the texture that do not lie on the target geometry with pixels of a similar color.

- c)** Click Next.

10. Set the options on the Replace/Delete page:

- a)** Enable “Replace textures on target geometry”.

This places the texture maps on the wall surfaces.

- b)** Enable “Reset the mesh on the target geometry”.

This removes the radiosity mesh from the walls.

- c)** Enable “Delete the projected geometry from the model”.

This removes the pictures and their frames from the model once the texture has been created.

- d)** Click Finish to create the texture map.

A new material named M2T WALLS->South000.jpg is added to the Materials table and the south wall surface is replaced with a texture map.

View the south wall texture and note the changed number of mesh elements

1. Select M2T WALLS->South000.jpg in the Materials table and look at the texture in the material preview.

You can also double-click the material name to open the Material Properties dialog.

2. Click the Outlined button .

The mesh on the surfaces appears. Notice how the wall surfaces are no longer meshed and the geometry of the picture frames is gone.

3. Choose Process | Statistics.

The model statistics appear on the status bar. Write down the number of mesh elements and compare with the previous number. Again, it should be much smaller.

4. Click the Solid button .

Create one texture map for all surfaces of the north wall

In this exercise, you map many surfaces into a single texture map and apply the single texture back onto the surfaces. This time, the first selection set is left empty, so you do not delete any of the geometry.

1. Turn off all layers except ROOM2_WALLS and ROOM2_PICTURES.
2. Orbit the view (or click the View Extents button ) so that you can see all surfaces of the north wall.

You are viewing the north wall when the east wall (the one with the alcove) is to your right.

3. To make it easier to select the entire wall in step 7, change the view so the viewer position is closer to the north wall. One way to do this is by dollying in.
4. Choose Tools | Mesh to Texture.

The Mesh To Texture wizard appears.

5. Select the “Project all selected geometry into one texture” radio button, then click Next.

This conversion method makes a single texture from all the surfaces you select.

6. Make sure that the first selection set is empty:

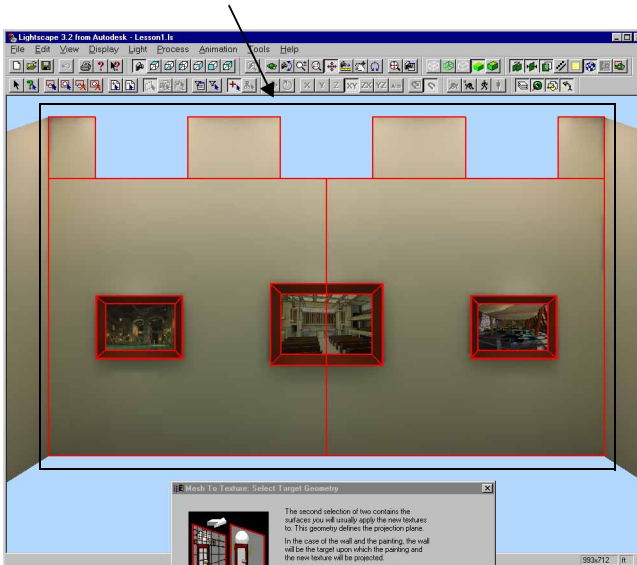
Note: The Select Projected Geometry page is where you select the surfaces to be projected onto the surfaces in the second selection set. This is not required for the method you are using in this exercise.

- a) If necessary, click the Deselect All button .
- b) Click Next.

7. Select the geometry for the second selection set:

Note: The Select Target Geometry page is where you select the surfaces on which textures will be placed.

- a) If necessary, click the Deselect All button  to make sure that the selection set is empty.
- b) Click the Area All button .
- c) To select the entire north wall (including the pictures and picture frames), drag a selection rectangle around the wall.



Note: When you select the surfaces using the Area All tool, be careful not to include surfaces from the pictures on the south wall in the selection set.

- d) Click Next.

8. Set the options on the Select Projection Method page:

- a) Click the Select button .

To pick the three points, you must be in Select mode.

- b) Select Orthographic from the Projection list.

- c) Enable Snap To Nearest Vertex so you do not have to be precise when picking the points.

- d) Make sure that the Lower Left radio button is selected, then pick three points on the wall (lower left, lower right, and upper left).

Pick the points the same way you did for the south wall. After you pick a point, the radio button for the next one is automatically selected.

- e) Click Next.

9. Set the options on the Texture Output Information page:

- a) Select JPEG (JPG) from the Format list.

- b) Type North.jpg in the New Textures Base Name box.

When you click Finish in step 11, a texture named *North000.jpg* will be created.

- c) Make sure that the Manually Size radio button is selected, then type 512 in the Width box and 256 in the Height box.

- d) Click Next.

10. Set the options on the Rendering Options page:

- a) Type 0 in the Ray Bounces box.

Typically, you want to avoid rendering specular reflections because they are view dependent and not suitable for 3D interactive applications.

- b) Enable Pad Texture Edge.

This option eliminates potential artifacts around the edges of textures by filling in all the pixels in the texture that do not lie on the target geometry with pixels of a similar color.

- c) Click Next.

11. Set the options on the Replace/Delete page:

a) Enable “Replace textures on target geometry”.

This places the texture maps on the wall surfaces.

b) Enable “Reset the mesh on the target geometry”.

This removes the radiosity mesh from the walls.

c) Make sure that “Delete the projected geometry from the model” is disabled.

d) Click Finish to create the texture map.

Five new materials are added to the Materials table and the surfaces are replaced with texture maps. Because you only created one new texture, all the materials use a part of the same texture. Before you used the Mesh to Texture tool, the north wall included surfaces that used the WALLS, FRAME, PICTURE1, PICTURE2, and PICTURE3 materials. The names of the new materials are:

- M2T WALLS->North000.jpg
- M2T FRAME->North000.jpg
- M2T PICTURE1->North000.jpg
- M2T PICTURE2->North000.jpg
- M2T PICTURE3->North000.jpg

View the north wall texture and note the changed number of mesh elements

1. Select M2T WALLS->North000.jpg in the Materials table and look at the texture in the material preview. This texture is used for all the new materials.

You can also double-click the material name to open the Material Properties dialog.

2. Click the Outlined button .

The mesh on the surfaces appears. Notice how the wall surfaces are no longer meshed but the geometry of the picture frames is still there.

3. Choose Process | Statistics.

The model statistics appear on the status bar. Write down the number of mesh elements and compare with the previous number. Again, it should be much smaller.

4. Click the Solid button .

5. Orbit around the model and compare the Mesh to Texture results for the south and north walls.

6. Turn on all layers.

Save your work

1. Choose File | Save As.

The Save As dialog appears.

2. Change the path to *C:\Program Files\Lightscape\Projects\Lesson5*.

3. In the File Name box, type MeshToTexture.ls.

4. Click Save.

Exporting Solution Files

In the first four lessons, you rendered images from Solution files. Lightscape can also export Solution files to panoramic and VRML file formats.

There are two exercises in this section:

- Exporting to Panoramic Format
- Exporting to VRML Format

Exporting to Panoramic Format

In this exercise, you export the sunroom model as a panoramic file using the RealSpace (*.ivr) format. An IVR file is a text file that describes the positions and other characteristics of images in a panoramic environment. When you create an IVR file, the corresponding JPEG image is also created. You must keep these two files in the same folder.

Live Picture, Inc. develops and sells applications for creating and viewing panoramic images. Live Picture® Zoom Viewer 3.2—Plug-in Version can be downloaded for no charge from www.livepicture.com. This plug-in requires Netscape® Navigator 3.01 or 4.0, or Microsoft® Internet Explorer® 3.0 or 4.0.

After you download and install this browser plug-in, you can view IVR images that are embedded in a web page. A sample HTML file called *Lightscape_IVR.html* is included in the *Lesson5* folder.

Note: Other applications can display IVR panoramic, IBM PanoramIX and VRML files. If you do not have a viewer application, you can still view the exported image in JPEG format.

Load a Solution file

You perform this exercise with a file that has already been created.

1. Choose File | Open.

The Open dialog appears.

2. Select Solution Files (*.ls) from the Files of Type list.
3. If necessary, change the path to *C:\Program Files\Lightscape\Projects\Lesson5*.
4. Double-click *Lesson2.ls*.

This is a similar Solution file to the one you created during Lesson 2, “The Sunroom.”

Create a panoramic image of the model

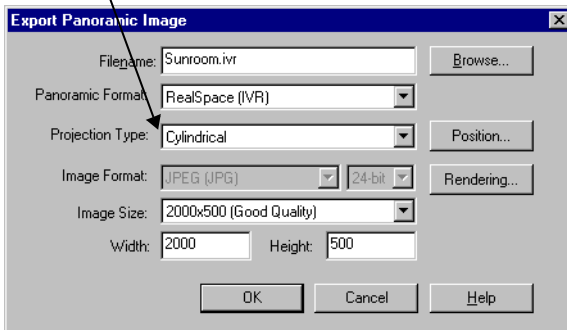
1. Adjust the view so you are in the center of the room.

For example, set the viewer position to $[-21.5, -4, 4]$ and the focus point to $[-21.5, 3, 4]$.

2. Rotate to view the model.
3. Choose File | Export | Panoramic.

The Export Panoramic Image dialog appears.

4. Type Sunroom in the Filename box.
5. Select RealSpace (IVR) from the Panoramic Format list.
6. Select Cylindrical from the Projection Type list.



Hint: The viewer position and focus point from the current view are used. Click Position if you want to change them. To change any of the rendering options, you can click Rendering.

7. Select 2000x500 (Good Quality) from the Image Size list.
8. Click OK.

View the panoramic image

1. If you installed Live Picture® Zoom Viewer 3.2, start your browser application (Navigator or Internet Explorer).

When you created the panoramic image, a file named *Sunroom.jpg* was also created. If you do not have an application to view the IVR file, you can display this file in LVu or another viewer.

2. Open the file named *Lightscape_IVR.html*.

The HTML code in this page includes an `<EMBED>` tag which loads *Sunroom.ivr*.

3. Examine the model using the Zoom In, Zoom Out, and Pan features.

Notice how the floor and ceiling are included in the panoramic image.

4. Close the browser application.

Exporting to VRML Format

VRML is a 3D file format designed to view 3D models over the World Wide Web (WWW). Several browsers can display VRML files. In this exercise, you export the gallery model as a VRML file.

Load a Solution file

You can perform this exercise with the original *Lesson1.ls* file or the optimized version named *MeshToTexture.ls* created earlier in this lesson. You may want to repeat the steps for both files to compare the results.

1. Choose File | Open.

The Open dialog appears.

2. Select Solution Files (*.ls) from the Files of Type list.
3. Change the path to *C:\Program Files\Lightscape\Projects\Lesson5*.
4. Double-click the file *Lesson1.ls*.

A message may appear if the *Lesson2.ls* file is still open. Click No so that changes are not saved.

Create a VRML version of the model

1. Click the Textures button  so that textures are displayed.

2. Choose File | Export | Vrmf.

The Export VRML dialog appears.

3. Type Gallery in the Name box, then click OK.

4. A warning explains that referenced textures may not be accessible from the VRML file. Click OK.

The *Gallery.vrmf* file is created in the *Lesson5* folder.

5. Choose File | Exit.

Summary

Congratulations! You have reached the end of Lesson 5. At this point you should be able to:

- Join Solution models
- Simplify models using the Mesh to Texture tool
- Export to panoramic and VRML formats

More Techniques with Blocks and Luminaires

Organizing surfaces and blocks into layers is important for managing the data in a complex model. All the surfaces of a model can be assigned to one layer, but it is easier to view, select, and modify objects if the model has good organization using layers. You can control how layers are created and how surfaces and blocks are assigned to the layers when you import a model. You can also change the organization in Lightscape.

Luminaires are blocks with photometric properties. To create a luminaire, you add photometric properties to an existing block. If you do not have a block to define as a luminaire, you can create one from surfaces or import a geometry file that includes a block.

In this lesson:

- Replace existing blocks by renaming and reloading the block
- Create a layer containing blocks for an alternate furniture layout
- Change the layer that block surfaces are on without changing where the block instances are
- Create a nested luminaire from geometry
- Load a luminaire from a library
- Make the light-emitting surface of a luminaire self-illuminating

Working with Blocks and Layers

If you want to modify objects in Lightscape (instead of your modeling application), use the various tools for transforming and duplicating surfaces and blocks. You can also add new blocks and layers.

Replacing Existing Blocks

In Lesson 4, you replaced several placeholder surfaces with instances of a luminaire. The Replace Surface with Block/Luminaire feature is useful for inserting blocks and luminaires into models imported from formats that do not support the block structure. However, you cannot use this feature to replace a block with another block.

Instead, you can rename the original block with the name of the replacement block, and then load the replacement block from a library to overwrite the original block. You avoid a lot of work because instances of the original block are automatically replaced by instances of the new block.

About the Lightscape Libraries

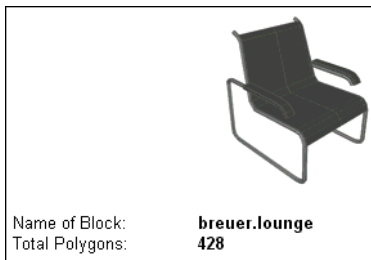
A set of Lightscape libraries is included with the Lightscape application. The libraries include hundreds of blocks, luminaires, and materials that you can easily use in your models.

Note: Depending on your Lightscape installation, the libraries may already be installed in the *C:\Program Files\Lightscape\Lib* folder. Otherwise, perform a Custom installation and select only Lightscape Libraries from the list of items to install.

You can find a pictorial index of all the library contents in the Online Help for the Lightscape libraries. To load the block, you need to know the name of the block and the library where it is stored.

Replace several instances of a block with another block

In Lesson 3, you replaced the CHAIR block as you imported the model from a DXF file. In this exercise, you replace the same block with a Breuer lounge chair. The name of the block is *chair.breuer.lounge* and it is stored in the library *Classic Chairs.blk*.



You can find this picture of the Breuer lounge chair by browsing through the contents of the block libraries in the Online Help.

1. Start Lightscape if the application is not already running.

2. Open the Lounge model from Lesson 3:

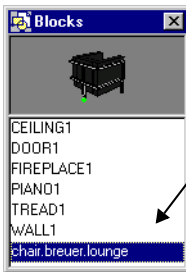
- a) Choose File | Open.
- b) Change the path to *C:\Program Files\Lightscape\Projects\Lesson3*.
- c) Double-click *Lounge.lp*.

3. Adjust the view so you can see the chairs and the piano.

For example, set the viewer position to [10, -11, 10], the focus point to [2, -3, 5], the near clip plane to about 0, and the field of view to 80.

4. Rename the CHAIR block:

- a) Right-click CHAIR in the Blocks table, then choose Rename.
- b) Type *chair.breuer.lounge* and press Enter.



Note: Block names are case-sensitive.

5. Load the replacement block:

- a) Right-click in the Blocks table, then choose Load.
- b) Change the path to *C:\Program Files\Lightscape\Lib\Blocks\Classic Furniture*.
- c) Double-click *Classic Chairs.blk*.
- d) In the Available Blocks dialog, select *chair.breuer.lounge* and click OK.
- e) When the warning message appears, click Yes to overwrite the blocks that already exist in the database.

The four chairs are replaced with Breuer chairs.

Note: You can also use this replacement technique to replace luminaires in Lightscape.

Rotate the Breuer chairs to face each other

In this exercise, you rotate the new chairs because the block definition had a different rotation. When you make your own block libraries, you should try to make similar blocks use the same rotation so they can be easily swapped. For example, all chairs could face the front side.

1. Select the four chairs:

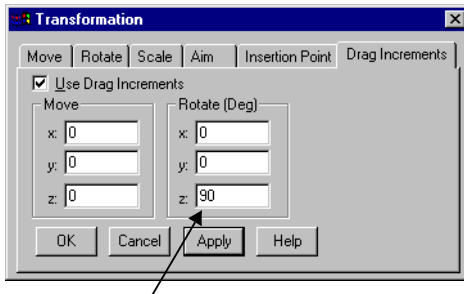
- a) Click the Select button , then click the Block button .
- b) Click the Accumulate Pick button  to turn on this feature.
- c) Select the four chairs.

2. Set the drag increments to help you use the interactive Rotate feature:

With the interactive Move and Rotate features, you can transform a block without using the Transformation dialog.

- a) Click the Edit Drag Increments button .

The Transformation dialog appears with the Drag Increments panel selected.

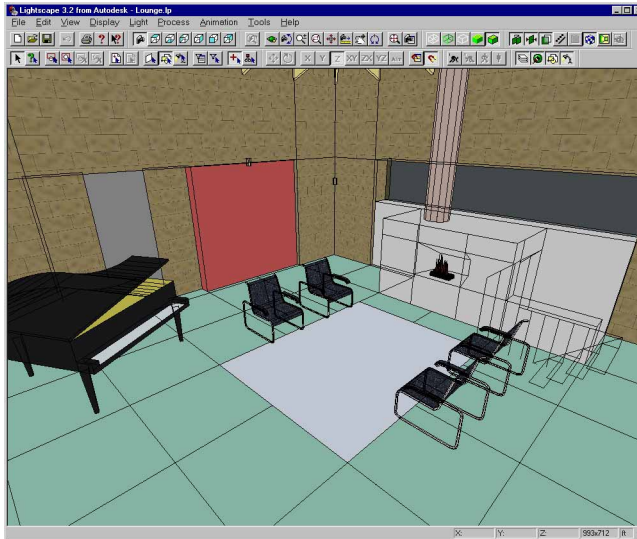


- b) Type 90 in the Rotate Z box.
- c) Enable Use Drag Increments.

You can also toggle this option by clicking the Use Drag Increments button .

- d) Click OK.

3. Rotate the selected blocks around the Z axis:
 - a) Click the Rotate button  to switch to interactive Rotate mode.
 - b) Click the Z Constraint button  to rotate around the Z axis.
 - c) Drag in the Graphic window until the block instances are rotated 90 degrees.



4. Click the Deselect All button  to clear the selection set.
5. Click the Accumulate Pick button  to turn off this feature.
6. Save the Preparation file as *NewLounge.lp* (in the *Lesson6* folder).

Using Layers to Organize Blocks

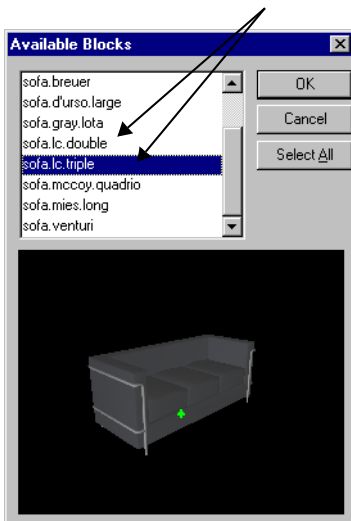
Using layers can be a valuable way to test design alternatives. For example, one layer can have the same blocks as another, but with different positions.

Create a second furniture layout

In this exercise, you create a layer with different blocks on it.

1. Double-click ROOM1_FURNITURE in the Layers table to turn off the layer.

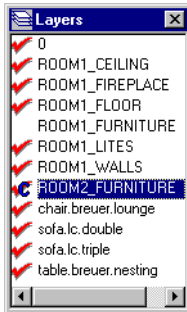
2. Create a new layer for the furniture layout:
 - a) Right-click in the Layers table, then choose Create.
 - b) Type ROOM2_FURNITURE and press Enter.
 - c) To make this layer current, right-click the layer name in the Layers table and choose Make Current.
3. Load the blocks for two Breuer sofas from the Lightscape libraries:
 - a) Right-click in the Blocks table, then choose Load.
 - b) Change the path to *C:\Program Files\Lightscape\Lib\Blocks\Classic Furniture*.
 - c) Double-click *Classic Sofas.blk*.
 - d) In the Available Blocks dialog, select sofa.lc.double and sofa.lc.triple.



Hint: You can press Ctrl or Shift while clicking to select more than one block.

- e) Click OK.

4. To load a Breuer table, repeat step 2 and select the table.breuer.nesting block from *Classic Tables.blk*.



A new layer with the same name as the block is created for each of the loaded blocks. This layer is used for the block surfaces, while the current layer is used for block instances.

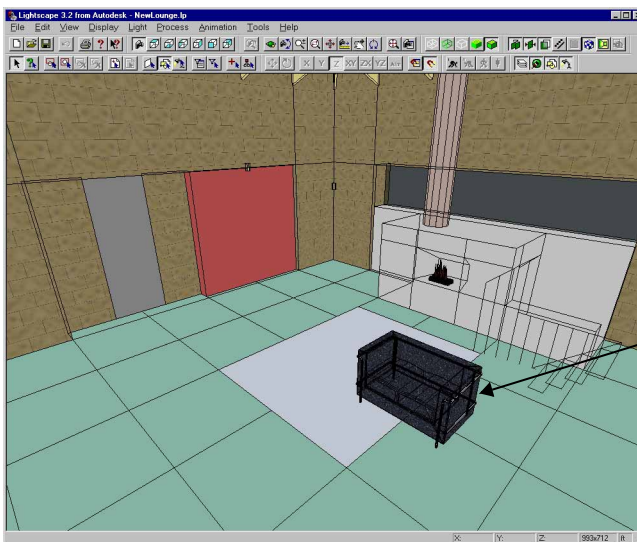
Note: A block stored in a library may have layers associated with the surfaces of the block. When you load the block into Lightscape, any layers associated with the block surfaces are added to the Layers table for the model.

The following steps show the difference between block instances and surfaces.

5. Place an instance of the sofa.lc.double block on the current layer:

- a) Select sofa.lc.double in the Blocks table.
- b) Drag an instance of the sofa to the Graphic window.

As you drag, the mouse cursor is a blue arrow with a block symbol.



Place the sofa in this position.

6. Hide the block instance:

- a) Double-click ROOM2_FURNITURE in the Layers table to turn off the layer.

The sofa disappears because the block instance is assigned to this layer. (The ROOM2_FURNITURE layer was the current layer when you created the instance.)

- b) Turn on the ROOM2_FURNITURE layer.

- c) To verify the layer that the block instance is on, click the Query Select button , click the Block button , and then select the sofa.

Information about the block instance (including the layer name) appears on the status bar.

7. Hide the block surfaces:

- a) Double-click sofa.lc.double in the Layers table to turn off the layer.

The sofa disappears because all the sofa surfaces are assigned to this layer.

- b) Turn on the sofa.lc.double layer.

- c) To verify the layer that the block surfaces are on, click the Query Select button , click the Surface button , and then click on the sofa.

Information about the selected surface (including the layer name) appears on the status bar.

8. Rotate the sofa around the Z axis:

- a) Click the Select button , then click the Block button .

- b) Select the sofa.

- c) Check that the Use Drag Increments feature is still on.

Earlier in this lesson, you set the drag increments to help you use the interactive Rotate feature.

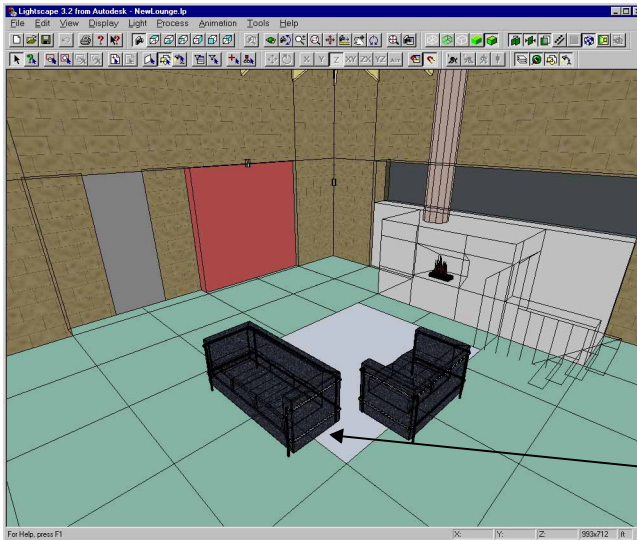
- d) Click the Rotate button  to switch to interactive Rotate mode.

- e) Click the Z Constraint button  to rotate around the Z axis.

- f) Drag in the Graphic window until the block instance is rotated 270 (or -90) degrees.

9. Place an instance of the sofa.ls.triple block on the current layer:

- a) Select sofa.lc.triple in the Blocks table.
- b) Drag an instance of the sofa to the Graphic window.



Place the second sofa
in this position.

c) Rotate the second sofa 180 degrees. See step 8 for specific instructions.

10. Place an instance of the table.breuer.nesting block on the current layer:

- a) Select table.breuer.nesting in the Blocks table.
- b) Drag an instance of the table to the Graphic window between the two sofas.
- c) Rotate the table 90 degrees. See step 8 for specific instructions.

Alternate between the two furniture layouts

Although the surfaces of the furniture are on three different layers, you can easily hide the new furniture layout because the block instances are all on the ROOM2_FURNITURE layer. When you turn off the layer with the block instances, the block instances disappear.

1. Double-click ROOM2_FURNITURE in the Layers table to turn off the layer.
2. Double-click ROOM1_FURNITURE in the Layers table to turn on the layer.

The block instances for this furniture layout are all on the ROOM1_FURNITURE layer.

3. In this case, it is easy to switch between the two layouts. You can save the various layer states, especially if a model has more than a few layers that differ.

To save a layer state, right-click in the Layers table and choose Save State. In the Save As dialog, enter a filename and click Save.

Note: The Israd, Isray, and Isrender batch processing utilities have an option where you specify a Layer State file. These batch utilities are explained in Lesson 7.

4. Switch layouts to show the ROOM2_FURNITURE layer again.

Moving Block Surfaces between Layers

As described in the previous exercises, block surfaces can be assigned to layers other than the layer that the block instance is assigned to. This feature is useful for when you only want to show some parts of a block.

When you create a block instance in Lightscape, it is assigned to the current layer. The block surfaces themselves continue to be assigned to their respective layers. For example, a table includes a metal frame (on the FRAME layer) and a glass top (on the GLASS layer). The current layer is named FURNITURE. You select all the table surfaces and create a block named TABLE. The block instance is on the FURNITURE layer, the frame surfaces are on the FRAME layer, and the glass surface is on the GLASS layer.

Note: The one exception when block surfaces do not continue to belong to their respective layers is when they are assigned to layer 0 when the block is created. In this case, the surfaces on layer 0 inherit the layer where the block instance is inserted. If the glass top in the previous example was originally on layer 0, it would be on the FURNITURE layer after the block instance was created.

Move the chrome surfaces to their own layer

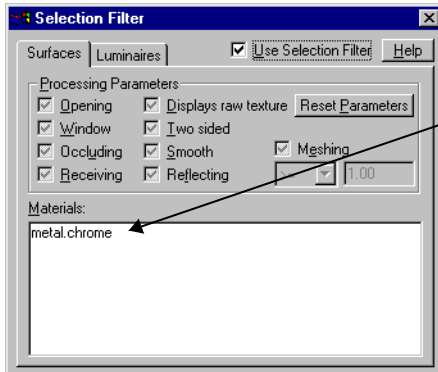
In this exercise, you create a separate layer for all the chrome surfaces of the new furniture.

1. Create a new layer and make it current:
 - a) Right-click in the Layers table, then choose Create.
 - b) Type Chrome and press Enter.
 - c) Right-click Chrome in the Layers table, then choose Make Current.

2. Select all the surfaces that are assigned the metal.chrome material:

- a) Right-click metal.chrome in the Materials table, then choose Add to Selection Filter.
- b) Choose Edit | Selection | Filter or click the Toggle Selection Filter Dialog button .

The Selection Filter dialog appears.



The material assigned to the chrome surfaces appears in the Materials list.

You can also use other characteristics of surfaces to select surfaces. For more information, see Chapter 4, "The Interface" in the *Lightscape User's Guide*.

c) Enable Use Selection Filter.

You can also toggle this option by clicking the Use Selection Filter button .

d) Click the Select button , then click the Surface button .

e) Click the Select All button to select all the chrome surfaces.

3. Change the selected surfaces to the Chrome layer:

- a) Right-click in the Graphic window, then choose Change to Current Layer.
- b) Click the Deselect All button .
- c) Right-click in the Materials list of the Selection Filter dialog, then choose Remove All.
- d) Click the Close box in the upper-right corner of the dialog or click the Toggle Selection Filter Dialog button .

4. Check that the block surfaces are on the new layer:

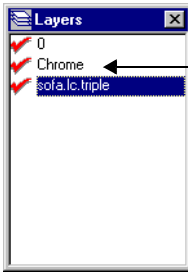
- a) To hide all the chrome surfaces, double-click Chrome in the Layers table.
- b) Double-click Chrome again to show the surfaces.

5. Another way to check which layers the block surfaces are on is to isolate a block:

a) Double-click sofa.lc.triple in the Blocks table.

The block appears by itself in the Graphic window.

b) Look at the layers in the Layers table.



The only layers displayed are layer 0 and the layers that include surfaces in this block. Layer 0 is the default layer, so it is always present.

c) Double-click sofa.lc.triple in the Layers table to turn this layer off.

All the sofa surfaces except the frame disappears.

d) Double-click sofa.lc.triple again to turn the layer on.

e) Right-click in the Blocks table, then choose Return to Full Model.

6. Save the changes made to the *NewLounge.lp* file.

Working with Luminaires

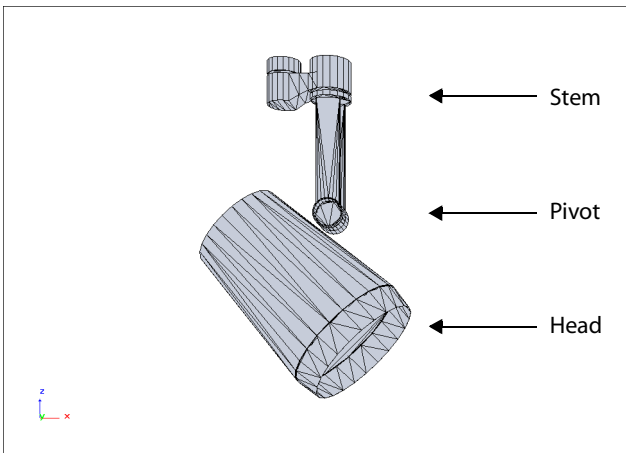
In Lesson 2, you converted an existing block into a luminaire and loaded a luminaire from a block library. This exercise shows how one of the luminaires in the ERCO library was created and it teaches the entire process of creating a complex luminaire from a geometry that is not a block.

Manufacturers often provide their luminaires in Lightscape-ready format, so you can simply load the luminaire from the block library and then use it in your model. In other cases, you may obtain only the photometric file from the manufacturer, so you must model and create the luminaire.

Note: This is an advanced exercise describing how you can create complex nested luminaires in Lightscape that consist of multiple moving components.

Creating a Luminaire from Geometry

The ERCO Unipar 77546 spotlight shown below has a stem that attaches to the ceiling and a head that rotates about a pivot.

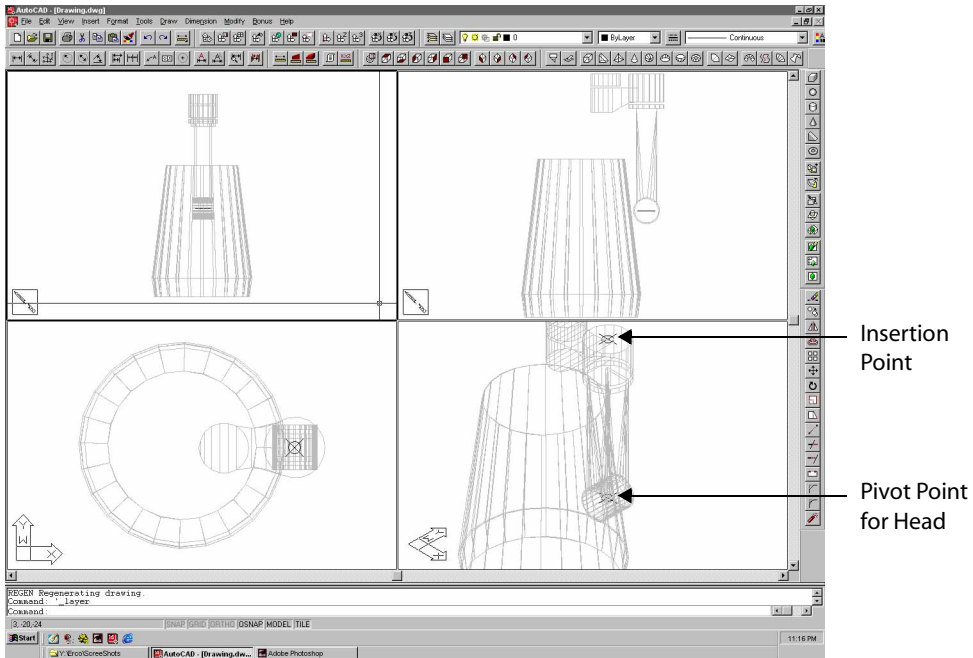


The insertion point represents the local coordinate system origin for the block. It acts as the point of contact and rotation when you insert a block instance into your model. It is important to strategically locate the insertion point to make it easier to insert and rotate luminaire instances.

If you created the Unipar 77546 luminaire as one block with a single insertion point (on the top of the stem) and one LID position (at the bottom of the head), you could not rotate the head without also rotating the stem. One solution is to create a nested luminaire. You have one insertion point for the stem to position the luminaire and a nested insertion point for the head so you can rotate it without affecting the stem.

Preparing the original geometry

The original model for the ERCO Unipar 77546 luminaire was created in AutoCAD. The head of the luminaire was defined as a block with its insertion point located at the pivot point of the head. Then the head block was nested with the stem surfaces into another block with its insertion point located at the top of the stem. This position is where the luminaire would typically be attached to a ceiling surface.



The geometry for the spotlight is given as a DXF file named *ERCO-Unipar-77546.dxf*. The head block in this file is named ERCO-UNIPAR-77546-HEAD, and the fixture block (which includes the nested head block) is named ERCO-UNIPAR-77546.

Note: Whenever a luminaire has a rotating head, make sure that the head in the geometry file points in the -Z direction. Rotations of the head in Lightscape will be relative to the original geometry. If the head in the geometry file does not point in the -Z direction, you will have to compensate for the offset every time you rotate or aim it.

Create the luminaire in Lightscape

In this exercise, you import the DXF file and then convert the blocks into luminaires.

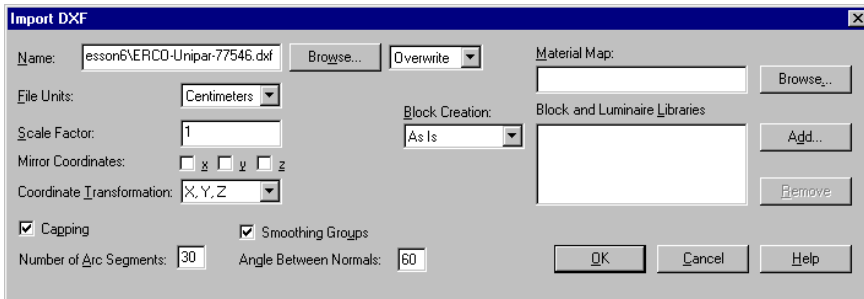
1. In Lightscape, choose File | New to create a new file.

2. Import the DXF file:

a) Choose File | Import | DXF.

The Import DXF dialog appears.

b) Click the Browse button beside the Name box, select *ERCO-Unipar-77546.dxf* in the *Lesson6* folder, and click Open.



c) Select Centimeters from the File Units list.

d) Enable Capping

e) Enable Smoothing Groups, then click OK.

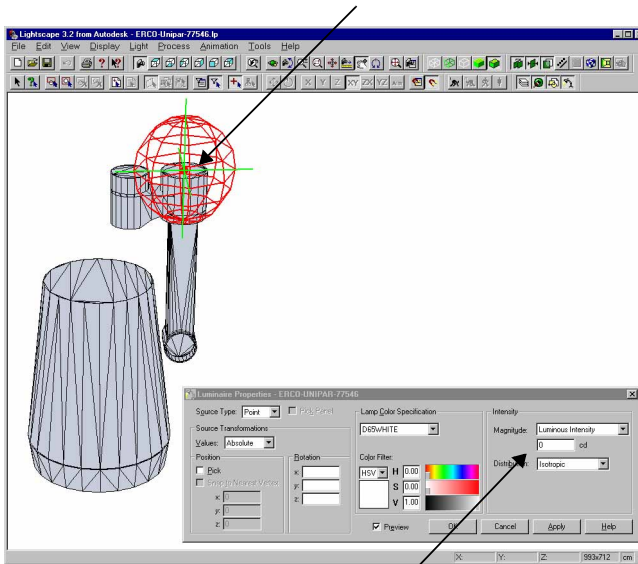
A dialog reports that the dimensions are 13.7166 x 11.5 x 23.4 centimeters. Click Yes.

f) Click the View Extents button .

3. Define the fixture block as a luminaire:

- a) Right-click ERCO-UNIPAR-77546 in the Blocks table, then choose Define as Luminaire.
- b) Click Yes to confirm that you want to continue this action.

The Luminaire Properties dialog appears and the view shows the isolated luminaire. Notice that the insertion point (green crosshairs) is correctly at the top of the stem.



- c) Type 0 in the Magnitude (Luminous Intensity) box.

When you create nested luminaires in Lightscape, all components (hierarchy levels) of the luminaire must be defined as luminaires even though some components may not emit light. For these components, you simply set the intensity to 0.

- d) Click OK, then click Yes to overwrite the luminaire.

4. Define the head block as a luminaire:

a) Right-click ERCO-UNIPAR-77546-HEAD in the Blocks table, then choose Define as Luminaire.

b) Click Yes to confirm that you want to continue this action.

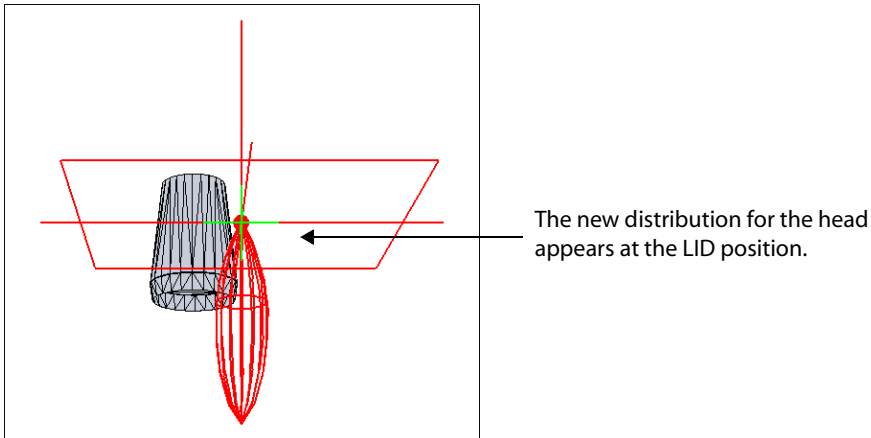
Notice that the insertion point (green crosshairs) is correctly at the pivot point. By default, the LID position is set to the original insertion point of the block (as set in the DXF file). You will move the LID to the bottom-center of the head in step 5.

c) Select Photometric Web from the Distribution list.

The distribution on the luminaire is replaced with a blank distribution.

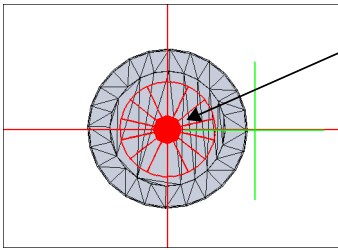
d) Click the Browse button, select *ERCO-77546-PAR30FL-75.ies*, and click Open.

The Magnitude (Luminous Intensity) box is set to 2200, which is the value in the IES file.



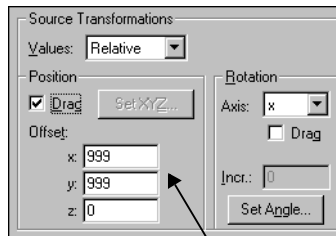
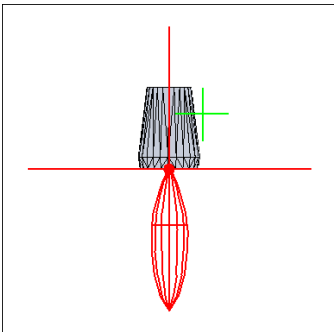
5. Move the LID to the bottom-center of the head:

- a) Select Relative from the Values list.
- b) Click the Bottom button  to show the Bottom projection.
- c) Enable Drag (in the Position group box) and drag the photometric web to the center.



- d) Click the Front button  to show the Front projection.

- e) Enable Drag (in the Position group box) and drag the photometric web to the bottom of the fixture.



Hint: You can limit the position change to only one axis by making the other Offset values very large. For example, set the Offset X and Y values to 999 to change only the Z position.

6. Click OK, then click Yes to overwrite the luminaire.

The Graphic window shows the full model.

7. View the new nested luminaire:

- a) Click the Select button , then click the Luminaire button .

Now the selection filter only selects luminaires.

- b) Click on the stem to select the entire spotlight.
- c) Make sure Accumulate Pick is turned off.
- d) Click near the bottom of the luminaire to select only the head.

If you cannot do this, check that the Pick Top Block feature is turned off. Choose Edit | Selection | Pick Top Block or click the Pick Top Block button .

- e) Click the Deselect All button .

8. Rename the material used for the luminaire surfaces:

- a) Right-click 9 in the Materials table, then choose Rename.
- b) Type ERCO-UNIPAR-77546, then press Enter.

Note: If the IES LID intersects with the geometry of a luminaire, you should make all the surfaces non-occluding and non-reflecting. In this example, the LID does not intersect the geometry, so this modification is not necessary.

9. Save both luminaires to a block library:

- a) Right-click in the Luminaires table, then choose Save All.

The Save As dialog appears.

- b) Type ERCO in the File Name box, then click Save.

Because you saved the luminaires to a block library and there is nothing else in this model, you do not have to save the current Preparation file.

Note: The Lightscape libraries include a selection of impressive ERCO luminaires in the *C:\Program Files\Lightscape\Lib\Lights\ERCO* folder. You can find additional luminaires at www.professional.erco.com.

Add and aim instances of the luminaire

The rotation of the real ERCO Unipar 77546 spotlight is constrained by a number of pivot points. Lightscape provides a convenient feature for aiming luminaires which aligns the -Z axis of the selected block towards a point on a selected surface. This Aim feature rotates the luminaire without constraints. In other words, the luminaire in Lightscape can only approximate the movement of the real luminaire. To control the rotation, you must place an insertion point at the pivot point when you model the luminaire.

1. Open the sunroom model:

- a) Choose File | Open.
- b) Change the path to *C:\Program Files\Lightscape\Projects\Lesson6*.
- c) Double-click *NewSunroom.lp*.

This copy of the sunroom model includes a framed picture. You will add two spotlights and aim them at the picture.

2. Load the ERCO luminaire from the block library:

- a) Right-click in the Luminaires table, then choose Load.

The Open dialog appears.

- b) Change the path to *C:\Program Files\Lightscape\Projects\Lesson6*.
- c) Double-click *ERCO.blk*.

The Available Luminaires dialog appears.

- d) Click Select All, then click OK.

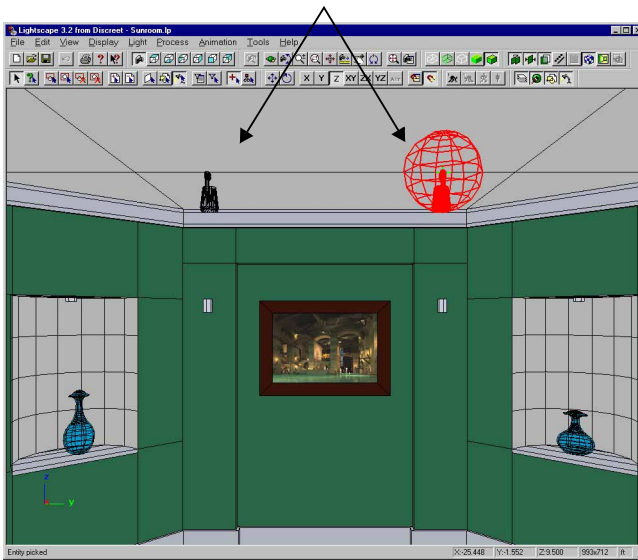
Lightscape can find the IES file for the ERCO luminaire because it is stored in the same folder as the current Preparation file.

Note: When you load luminaires from block libraries in another folder, make sure you add the path to the IES file to the paths for Luminaire Distributions. You can view and change these paths on the Paths panel of the Document Properties dialog.

3. Open a view of the picture:

- a) Choose View | Open.
- b) In the Open dialog, double-click *PictureView.vw*.

4. Drag two instances of the ERCO-UNIPAR-77546 luminaire from the Luminaires table to the ceiling of the sunroom.



If necessary, move the luminaires using the interactive Move feature.

5. Orbit to see the luminaires from the side.

Before you can aim the head luminaires at the picture, you must rotate the fixture luminaires 180 degrees. Otherwise, the head will intersect the stem of the luminaire.

6. Rotate the two luminaires:

- a) Select both luminaires.

You can use Accumulate Pick or press Ctrl as you select the second luminaire.

- b) Right-click in the Graphic window, then choose Transformation.

- c) Click the Rotate tab on the Transformation dialog.

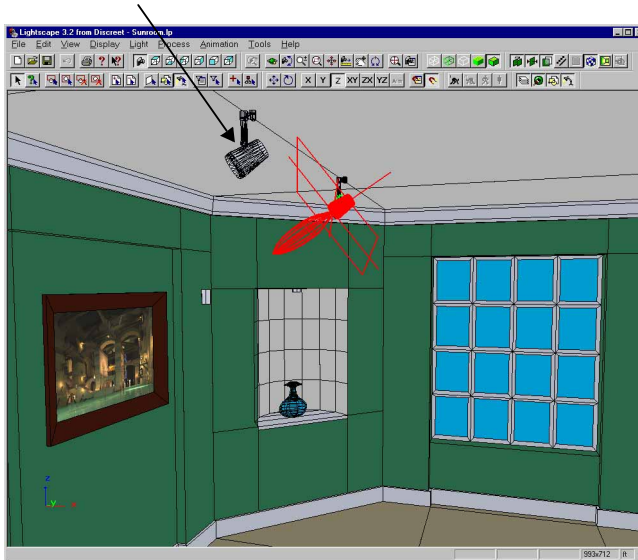
- d) Enable Relative.

- e) Enter 180 in the Offset Z box, then click OK.

7. Aim one of the luminaires:

- a) Click the Deselect All button .
- b) Make sure that the Pick Top Block feature is turned off.
- c) Select the head luminaire that is nested in one of the ERCO-UNIPAR-77546 luminaires.
- d) Right-click in the Graphic window, then choose Transformation.
- e) Click the Aim tab on the Transformation dialog.
- f) Enable Pick.
- g) Select a point on the picture.

Notice that the rotation of the head luminaire that is nested in the second spotlight changes to the same amount of rotation. If necessary, orbit the model to see this.



This demonstrates that when you are using nested luminaires, you should create a copy of the luminaire instead of an instance. The copies of the nested luminaire can then have different rotations.

Note: If you accidentally move the head luminaire away from the stem, you should reload the luminaires from the block library.

- h) Click OK.

8. Duplicate the luminaire definition:

- a) Click the Deselect All button .

One way to duplicate a luminaire definition is to select it in the Luminaires table, right-click, and choose Duplicate. Since you already placed the second luminaire, you can select it and make a duplicate definition using the Make Block feature. Both techniques produce the same result.

- b) Select the other fixture luminaire.

Make sure you pick the top luminaire of the hierarchy.

- c) Right-click in the Graphic window and choose New Block.

- d) In the New Block dialog, type ERCO-UNIPAR-77546 Copy 1.

- e) Click OK.

9. Aim the second luminaire:

- a) Click the Deselect All button .

- b) Make sure that the Pick Top Block feature is turned off.

- c) Select the head luminaire that is nested in the second ERCO-UNIPAR-77546 luminaire.

- d) Right-click in the Graphic window, then choose Transformation.

- e) Click the Aim tab on the Transformation dialog.

- f) Enable Pick.

- g) Select a point on the picture.

- h) Click OK.

Assign a self-emitted luminance to the luminaire

You can still modify the surfaces of the block after it is defined as a luminaire. For example, you can assign a different material to the surfaces.

In this exercise, you make the luminaire appear to be on by assigning a material that has a self-emitted luminance value. Although light appears to be emanating from the source, no light energy is ever contributed to the scene by this material. (In Lightscape, the actual light source is not visible.)

1. Create a new material:

- a) Right-click in the Materials table, then choose Create.
- b) Type ERCO-UNIPAR-77546-GLOW as the name of the new material, then press Enter.

2. Change the properties of the new material:

- a) Double-click the ERCO-UNIPAR-77546-GLOW material.

The Material Properties dialog appears.

- b) On the Color panel, set V (value) to 1.00.
- c) On the Physics panel, enable Pick Light.
- d) Select a luminaire in the Graphic window, then click OK.

The intensity value for the luminaire is copied to the material's Luminance (Glow) box. Using the Pick Light option produces accurate results when you perform lighting analysis.

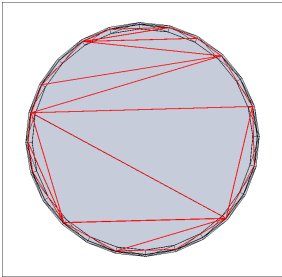
Hint: You can also set Luminance (Glow) to a reasonable value such as 1000.

3. Isolate the ERCO-UNIPAR-77546-HEAD fixture to assign the material:

- a) Right-click ERCO-UNIPAR-77546-HEAD in the Luminaires table, then choose Isolate.
- b) Click the Bottom button  to show the Bottom projection.

4. Select the bottom surfaces:

- a) Click the Select button , then click the Surface button .
- b) Click the Accumulate Pick button  to turn on this feature.
- c) Click the 13 bottom surfaces one at a time.



← You can safely omit the small surfaces near the edge.

- d) Right-click in the Graphic window and choose Assign Material.
- e) In the Assign Material dialog, double-click ERCO-UNIPAR-77546-GLOW.
- f) Right-click in the Luminaires table and choose Return to Full Model.

After you initiate the Solution model and before the first iteration of energy distribution is performed, the luminaires appear to be on. You can orbit the model to check the glowing luminaire surfaces.

Process the radiosity solution

1. Choose View | PictureView.vw to load the view of the framed picture.
2. Initiate the model:
 - a) Save the Preparation file.
 - b) Click the Solid button .
 - c) Click the Initiate button .

3. To see the light from the luminaires you created, process the solution:

- a) Click the Go button .
- b) Click the Stop button  after about 15 iterations.

Look at the light cast on the framed picture.

- c) Save the Solution file as *NewSunroom.ls*.

4. Choose File | Exit.

Summary

Congratulations! You have reached the end of Lesson 6. At this point, you should understand:

- How to replace existing block instances
- Basics of how layers can be created for blocks and luminaires
- How to place blocks and luminaires in the model
- How to move surfaces and blocks between layers
- How to convert a nested block into a nested luminaire
- How to assign a photometric web to a luminaire

Using the Batch Processing Utilities

You are now familiar with the main Lightscape application. Many of the features in the Lightscape interface can also be accessed through batch processing utilities that you can run at the command line or in batch files. When you enter the utility commands in a batch file, they can be run sequentially over an extended period of time.

The main advantage of batch processing is efficiency. First, a utility command runs faster than the equivalent feature in the Lightscape application because there is no graphical user interface. For example, you can process a radiosity solution faster because the model is not redrawn after each iteration. Second, you do not need to be present when a batch file is running. You can do something else and then return to find several different solutions and images.

LSnet is a network rendering system that offers a handy interface to the three main batch processing utilities. You can use it for radiosity processing, ray tracing, and OpenGL rendering.

In this lesson:

- Run a batch utility from the command line
- Create and run a simple batch file
- Edit and run an existing batch file
- Use LSnet to process a radiosity solution and render an image
- Use LSnet to create a sequence of animation images

Using the Command Line

It is easy to run a batch utility from the command line of an MS-DOS window. The most common batch utilities are:

- **lsrad**: Creates the radiosity solution
- **lsray**: Creates a ray-traced image of a Solution file
- **lsrender**: Creates a non-ray-traced image of a Preparation or Solution file

Other utilities are used for importing models, merging Lightscape files, and converting files between different formats. The batch utilities generally use this syntax:

COMMAND [options] Lightscape_file

The option flags are sometimes followed by an extension. For example, the output (-o) option is followed by a filename but the verbose (-v) option has no extension. For a complete list of the options for each batch utility, see Appendix B, “Batch Processing Utilities” in the *Lightscape User’s Guide*.

Run a batch utility

In this exercise, you run the **lsrad** batch utility by typing the following line:

```
lsrad -term 20 -v -o Gallery2.ls Gallery.lp
```

The **lsrad** batch utility is called with three options and a filename:

- The terminate iterations (-term) option stops the command after 20 iterations
- The verbose (-v) option prints status information as the command runs
- The output (-o) option outputs the solution to the file *Gallery2.ls* instead of *Gallery.ls* (the default based on the name of the Preparation file)
- The file *Gallery.lp* is the input for the command

1. Choose Start | Programs | MS-DOS Prompt.

Note: In Windows NT, MS-DOS Prompt is named Command Prompt.

2. At the prompt, type the following line to change the directory:

```
cd "C:\Program Files\Lightscape\Projects\Lesson1"
```

The quotation marks are required because there is a space between the words Program and Files. If you have installed Lightscape in a place other than the default location, you need to alter the path accordingly.

3. Type the following line to create a radiosity solution for the Gallery model:

```
..\..\bin\lsrad -term 20 -v -o Gallery2.ls Gallery.lp
```

A file named *Gallery2.ls* is created in the current folder.

Note: You type `..\..\bin` to indicate the relative location of the `lsrad` batch utility. For more information on file locations, see “Location of Batch Utilities and Project Files” on page 246.

4. Type `exit` and press `Enter` to close the MS-DOS window.
5. Start Lightscape if the application is not already running.
6. Open the Solution file that you created with the `lsrad` batch utility:
 - a) Choose `File | Open`.
 - b) Select `Solution Files (*.ls)` from the `Files of Type` list.
 - c) Change the path to `C:\Program Files\Lightscape\Projects\Lesson1`.
 - d) Double-click the file *Gallery2.ls*.
7. View the model.

The batch utility used the radiosity processing parameters that are saved with the Preparation file. Choose `Process | Parameters` to see the current settings, then click `Cancel`.

Using Batch Files

A batch file is an ASCII text file saved with the .BAT extension. The file contains a series of executable commands with command options. When you run a batch file, the commands are executed in the order they are listed in the file.

Because the commands are executed without your supervision, you can run the batch file when you are not using the computer. For example, you can process a radiosity solution and then render several ray-traced images overnight.

Note: While you run batch processing utilities, system performance decreases significantly if you use screen savers other than the one that shows a blank screen. This warning is especially true for OpenGL screen savers.

You can create (and edit) a batch file in a text editor such as Notepad. To run the batch file, you type the name of the file (without the extension) on the command line in MS-DOS Prompt.

Hint: You can also double-click a batch file in Windows Explorer to run it. When you do this, an MS-DOS window automatically opens to display the status information of the commands. After the last command finishes, the window closes.

Location of Batch Utilities and Project Files

In a batch file, you must consider the locations of the project files and the batch utilities. If you save the batch file in the *bin* folder, you do not need to supply the paths for the batch utilities. You only have to give the location of the project files.

Likewise, if you save the batch file in the folder of the current project, you do not need to supply the paths for the project files. You only have to give the location of the batch utilities. You can do this easily with the Path command:

path C:\Program Files\Lightscape\bin

The Path command is used to find executable files in the specified folder. (By default, the search path is the current folder only.)

Create and run a simple batch file

In the previous exercise, you ran a batch utility from the command line. Now, you create a simple batch file that contains the following line:

```
lsrad -termp 75 -cp 10 -v -o Sunroom2.ls Sunroom.lp
```

The `lsrad` batch utility is called with three options and a filename:

- The terminate percentage (`-termp`) option stops the command after 75 percent of the energy is distributed
- The checkpoint (`-cp`) option outputs a Solution file every 10 iterations in case of power failure or other problems
- The verbose (`-v`) option prints status information as the command runs
- The output (`-o`) option outputs the solution to the file *Sunroom2.ls* instead of *Sunroom.ls* (the default based on the name of the Preparation file)
- The file *Sunroom.lp* is the input for the command

1. Choose Start | Programs | Accessories | Notepad.

You can use Notepad to create or edit text files that do not require formatting and are smaller than 64K. Notepad opens and saves text in ASCII (text-only) format only.

2. Type the following line to indicate the location of the batch utilities:

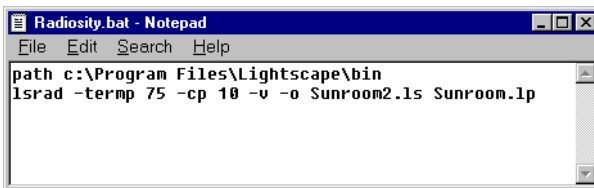
```
path C:\Program Files\Lightscape\bin
```

Note: You can omit this line if you add the path to the system environment on your computer.

3. Press Enter, then type the following line to create a radiosity solution for the Sunroom model:

```
lsrad -termp 75 -cp 10 -v -o Sunroom2.ls Sunroom.lp
```

This is the last line of your batch file.



You do not need to indicate the location of the Sunroom files because you will save the batch file in the folder where the project files are.

4. Save the batch file:

- a) Choose File | Save As.
- b) Change the path to *C:\Program Files\Lightscape\Projects\Lesson2*.
- c) Type *Radiosity.bat* in the File Name box, then click Save.

Note: You must type the file extension (.bat) to make the file a batch file. For example, you cannot run the batch file if the file extension is .txt.

- d) Choose File | Exit to close Notepad.

5. Choose Start | Programs | Windows Explorer.

6. Change the path to *C:\Program Files\Lightscape\Projects\Lesson2*.

Hint: To quickly open a folder and display its subfolders, double-click the folder on the left side of the window.

7. Double-click *Radiosity.bat* to run the batch file.

An MS-DOS window automatically opens to display status information of the commands.

Notice that *Sunroom2.ls* is saved before the end of processing due to the checkpoint option. The window closes after 75% of the energy is distributed.

8. If you want to view the radiosity solution, open *Sunroom2.ls* in Lightscape.

Edit and run an existing batch file

Instead of creating a new batch file, you can edit one to suit your current project. For example, you can simply change the filenames and a few command options.

In this exercise, the *lsrad* batch utility is called with the same options as the previous exercise except the terminate iterations (-term) option is used instead of the terminate percentage (-termp) option. The following options are used with the *lsrender* and *lsray* utilities:

- The antialiasing (-aa) option improves the image quality (using a factor of 2 in this example)
- The verbose (-v) option prints status information as the command runs
- The image width (-x) and image height (-y) options set the size of the output image

1. Choose Start | Programs | Accessories | Notepad.

2. Open an existing batch file:

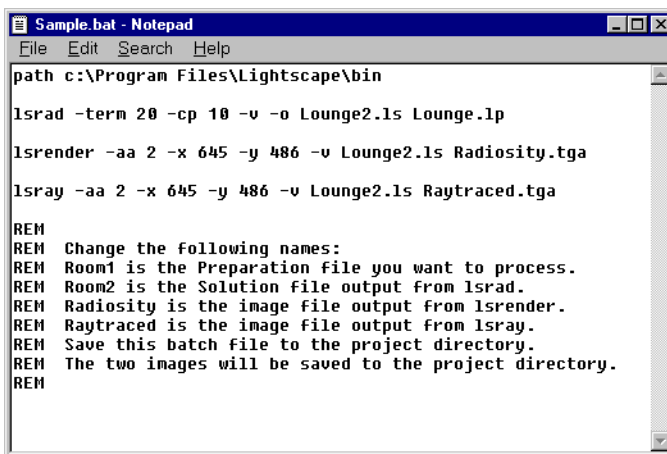
- a) Choose File | Open.
- b) Change the path to *C:\Program Files\Lightscape\Projects\Lesson7*.
- c) Select All Files from the Files of Type list.
- d) Double-click *Sample.bat*.

Notice that several lines are commented out. To comment out a line that you do not want to execute, insert “REM” at the beginning of the line.

3. Edit the batch file so that the *Lounge.lp* file from Lesson 3 is processed, rendered, and ray traced:

- a) On the *lsrad* line, change “Room1” to “Lounge”.
- b) On the *lsrad*, *lsrender*, and *lsray* lines, change “Room2” to “Lounge2”.

The resulting batch file should look like this.



```

path c:\Program Files\Lightscape\bin

lsrad -term 20 -cp 10 -v -o Lounge2.1s Lounge.lp

lsrender -aa 2 -x 645 -y 486 -v Lounge2.1s Radiosity.tga

lsray -aa 2 -x 645 -y 486 -v Lounge2.1s Raytraced.tga

REM
REM  Change the following names:
REM  Room1 is the Preparation file you want to process.
REM  Room2 is the Solution file output from lsrad.
REM  Radiosity is the image file output from lsrender.
REM  Raytraced is the image file output from lsray.
REM  Save this batch file to the project directory.
REM  The two images will be saved to the project directory.
REM

```

4. Save the batch file as *Images.bat*, then exit Notepad.

5. In Windows Explorer, copy *Images.bat* to the *Lesson3* folder.

You need to save the batch file in the folder where the project files are because the location of these files is not given in the batch file.

6. Run the batch file from Windows Explorer:

- a) Change the path to *C:\Program Files\Lightscape\Projects\Lesson3*.
- b) Double-click the *Images.bat* file.

An MS-DOS window automatically opens to display status information of the commands. After the images are created, the MS-DOS Prompt window closes.

7. Start the LVu utility so you can view the results of the batch file. Compare the two images created by the batch processing utilities.

The diffuse surfaces look good in *Radiosity.tga*, but the image is not ray traced. The *Raytraced.tga* image shows procedural textures (on the red wall) and reflections on the shiny surfaces (such as the chrome on the chairs).

8. To exit LVu, choose File | Exit.

Using LSnet

LSnet is a network rendering system for Lightscape. With LSnet, you can use your network to perform radiosity processing, ray tracing, and OpenGL rendering.

LSnet is included with the Lightscape application, but it is not installed automatically. For information on how to install LSnet, refer to Appendix C, “LSnet” in the *Lightscape User's Guide*.

You can perform radiosity processing of several Lightscape files on several computers on the network simultaneously. (A radiosity solution cannot be split over multiple nodes, though.) You can do simultaneous ray tracing and OpenGL rendering of multiple views or animation frames to increase rendering productivity.

LSnet can also be used on a single computer. The user interface makes it easy to create a list of jobs, allowing the unattended processing of a series of Lightscape files. If you perform a common sequence of jobs on your files, then you can save and load LSnet project files.

With the LSnet JobQ utility, all members of your team can easily create rendering jobs and submit them to LSnet for processing. Finally, the scheduling feature allows you to automatically run jobs when your network is underutilized.

Note: The following exercises assume that LSnet is already installed and your computer has one processor (node). Extra information is given in the instructions for systems with more than one node.

Set up and process two jobs in LSnet

Although LSnet does not create batch files, it offers a handy interface to all the options of the `lsrad`, `lsray`, and `lsrender` batch processing utilities. In this exercise, you set up two LSnet jobs.

1. Choose Start | Programs | Lightscape | LSnet JumpStarter.

Note: If the shortcut is not there, find the application in the shared JumpStarter folder. (There is only one Jumpstarter folder for the whole network.)



2. When Online appears in the CPU 1 Status box, minimize the LSnet JumpStarter window.

This program must be running on each LSnet node.

Note: If a computer has two processors, it can have two nodes but only one LSnet JumpStarter needs to be running on it.

3. Choose Start | Programs | Lightscape | LSnet.

Note: If the shortcut is not there, find the application in the shared LSnet folder. (There is only one LSnet folder for the whole network.)

4. Choose Network | Initialize or click the Initialize button  to initialize the node(s).

Note: The LSnet application must be configured correctly. If you have problems with this exercise, refer to Appendix C, "LSnet" in the *Lightscape User's Guide*.

LSnet communicates with the nodes. The status of the nodes appears in the Node List at the bottom of the LSnet window.

5. Load a Lightscape file for a new job:

- a) Choose Project | Load File.

- b) Change the path to *C:\Program Files\Lightscape\Projects\Lesson1*.

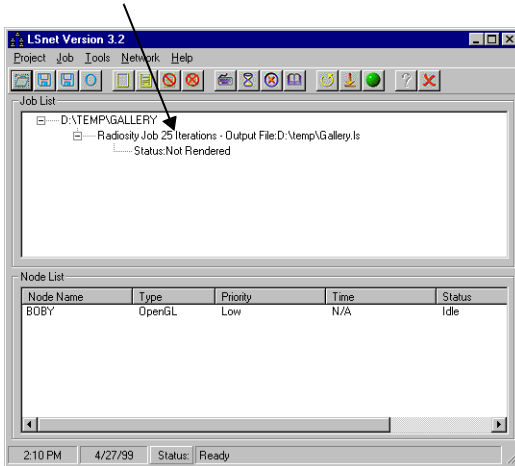
If any nodes are on networked computers, you should first copy all the files in the *Lesson1* folder to a shared folder, and then change the path to the shared folder in this step.

Note: Before you can make a shared folder, display the Network properties, click File and Print Sharing, and enable "I want to be able to give others access to my files". Then, select the folder that you want to share, show its properties, and set up full access on the Sharing panel.

- c) Double-click *Gallery.lp*.

6. Edit the job:

a) Select the new job.



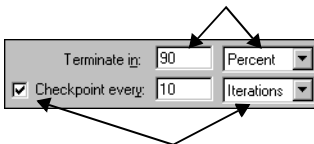
b) Right-click and choose Edit Job, or click the Edit Job button .

The Job Setup window appears. The three types of job are LSrad, LSray, and LSrender. By default, the Job Type for a Preparation file is set to LSrad.

c) Change the Output File Name from Gallery.ls to Gallery3.ls.

By using a different output file, you avoid overwriting the existing Solution file.

d) Change the Terminate In box to 90 percent.



e) Enable Checkpoint and set the unit to Iterations.

f) Click OK.

7. Create a second job:

a) Select the first job.

b) Right-click and choose New Job | Clone Job, or click the Clone Job button .

The Job Setup window for the new job appears. Notice how the options are set the same as the first job.

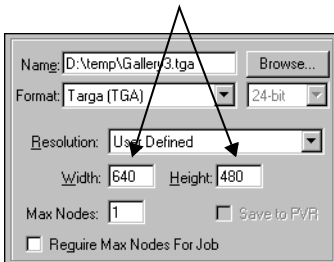
c) Select LSrender from the Job Type list.

d) Change the Input File box from Gallery.lp to Gallery3.ls.

Note: Make sure you change the extension as well as the name because the first job creates a Solution file, not a Preparation file.

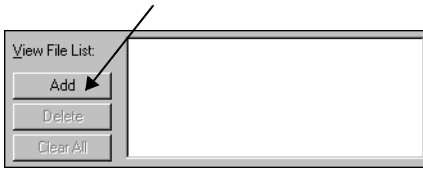
This creates a dependency between the two jobs because the output of the first job is the input required for the second job.

e) Change Width to 640 and Height to 480.



The default output file for the LSrender job is named *Gallery3.tga*.

- f) For the View File List, click Add, select *Rm1.vw*, and click Open.



This view is used to render the image.

- g) Select Two from the Antialiasing Level list.

- h) Click OK.

A dialog appears with the warning “The Input File you have chosen does not exist yet. Continue?” Click Yes.

8. Choose Network | Render or click the Render button .

The first job starts and the second job waits.

Note: Even if you have two or more nodes, the second job will wait because of the job dependency.

9. To view the rendered image, open *Gallery3.tga* in the LVu utility.

This file is saved in the *Lesson1* folder.

Create a sequence of animation images

In this exercise, you set up an lsray job for the exterior model from Lesson 4. Using an animation file, LSnet creates a sequence of images.

Note: It is very important that the Preparation and Solution files that you use with LSnet have textures and IES distributions that are not referenced with full paths. LSnet will not be able to find the textures and the jobs will not continue. For example, a texture map on the Material Properties dialog could be named *Picture.tga*, but not *C:\Images\Picture.tga*. For this exercise, the texture files are in the same folder as the Solution file and the filenames are set up correctly in the Solution file.

1. To clear all the existing jobs, right-click in the Job List and choose Clear All.

When the warning message appears, click Yes to continue.

2. Load a Lightscape file for a new job:

- a) Choose Project | Load File.
- b) Change the path to *C:\Program Files\Lightscape\Projects\Lesson4*.

If any nodes are on networked computers, you should first copy all the files in the *Lesson4* folder to a shared folder, and then change the path to the shared folder in this step.

- c) Select Solution File (*.ls) from the Files of Type list.
- d) Double-click *Day.ls*.

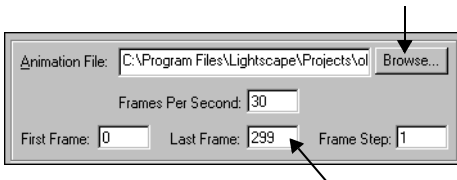
You use this Solution file to create ray-traced images.

3. Edit the job:

- a) Select the new job.
- b) Right-click and choose Edit Job, or click the Edit Job button .

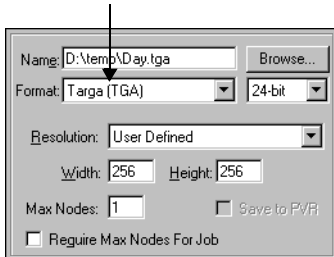
The Job Setup window appears. By default, the Job Type for a Solution file is set to LSray.

- c) To select an animation file, click the Browse button beside the Animation File box (at the bottom of the window), select *Exterior.la*, and click Open.



If you want to decrease the processing time, you can shorten the animation by decreasing the Last Frame value.

4. Select Targa (TGA) from the Format list.



The default name for the image files is Day.tga so, for example, the sequence starts with *Day0000.tga*.

5. Click OK.
6. Choose Network | Render or click the Render button .

If you have one node, the frames are processed one at a time. If you have two or more nodes, the frames are distributed over the nodes.

7. Choose Project | Quit.

When you quit without saving the project, a warning message appears. Click Yes to quit without saving the setup information.

Note: The files created when the job was processed are not affected.

8. To view the rendered images, start the LVu utility and open the *Lesson4* folder.

Note: You can use another program (such as *Makeavi.exe*) to make a movie clip from animation images.

9. Exit LSnet JumpStarter.

Summary

Congratulations! You have reached the end of Lesson 7. At this point, you should be able to:

- Use a few of the batch processing utilities
- Re-use an existing batch file for another project
- Use LSnet for radiosity processing, ray tracing, and OpenGL rendering

Going Further with Sample Files

Sample Preparation and Solution files are included with the Lightscape application. You can experiment with these files to learn more about Lightscape.

Note: Depending on your Lightscape installation, these files may already be installed in the *C:\Program Files\Lightscape\Projects\Samples* folder. Otherwise, perform a Custom installation and select only Sample Files from the list of items to install.

Here are some other things to try:

- Look at how the Preparation models are constructed.
- Look at the radiosity meshing, materials, and luminaires in the Solution files.
- Compare your system performance for a large Lightscape model to the smaller models used in these lessons.
- Practice using view controls in the *2 rooms.lp* model.
- Try different materials with the various lamp color source attributes in the *cl_temp.ls* model.
- Try different materials with the luminaires in the *lum_rev.ls* model. These luminaires are included in the prototype luminaire library named *lvs_lum.blk*.
- Reset radiosity solutions and experiment with different processing parameters.
- Learn more about the Lightscape libraries by completing the Library lesson. This extra lesson is available in PDF format on the Lightscape CD.
- Read the *Lightscape User's Guide*. This guide contains a reference to all the software features as well as descriptions of the underlying lighting concepts. The more you understand about these concepts, the faster you can master working with Lightscape.

index

Index

Numerics

- 0 layer 226
- 3D Studio MAX 2
- 3DS file format 2, 45

A

- A/E/C 45
- Accumulate Pick 59
 - usage tip 63
- adaptive subdivision 29, 81, 124
- aligning textures 119
- ambient approximation 85
- Ambient display control 128
- animation
 - creating 134
 - frames 142, 255
 - playing 141, 190
- animation files
 - loading 190
 - saving 142
- antialiasing 37
- Antialiasing display control 55
- artifacts 128
- ASCII text file 246
- Auto-Orient tool 113
- average reflectance 18

B

- backface culling 15, 113
- background color 56
- batch file 246

- batch processing utility 29
 - in batch file 246
 - running 244
- beam angle 69
- Blending display control 55
- blocks
 - creating 60
 - defining as luminaire 68
 - duplicating 62
 - isolating 108
 - moving 63
 - nested 230
 - previewing 12
 - replacing during import 101
 - replacing using names 218
 - rotating 64
 - scaling 64
 - surfaces on layers 226
- Blocks table 12
- brightness 83
- bump mapping 161

C

- camera motion spline 135
- camera path 135
- camera position 50
- cloning jobs 254
- color
 - background 56
 - of material 18
 - picker 56
- constraints 63
- context menu 11

- contrast 83
- control points 135
- Create Single Instance 63
- culling 15
- Culling display control 15
- current layer 58

D

- day solutions 178
- daylight
 - using Direct Control 179
 - using Place and Time 90, 182
- daylight calculation 87
- diffuse material 73
- Direct Control 179
- direct lights, testing 127
- Direct Source Minimum 126
- display controls 55
- display modes 14, 55
- Display Original View 15
- distance
 - measuring 48
 - units 47
- Dolly view control 53
- Double Buffer display control 55
- double-sided surfaces 1
- drag and drop
 - a block 63
 - a luminaire 164
 - a material 32
 - a texture map 19
- drag increments 65
- Draw Every Nth Face 85
- duplicating
 - blocks 62
 - luminaires 239
- DWG file format 149
- DXF file format 45, 101
- Dynamic View mode 11

E

- element 81
- EMBED tag 214
- Enhanced display control 15
 - for smoothing 117
- ERCO 70, 229

F

- far clipping plane 50
- field angle 69
- field of view 50
- Filter Methods 120
- Fixed Size 120
- focus point 50
- frustrum 50

G

- Glow 240
- Graphic window 10

H

- handles 135
 - for control points 135
 - for keyframes 135
 - moving 138
- Hidden Line display mode 55
- hot keys, for view controls 16, 53
- Hue 73

I

- importing
 - DWG files 149
 - DXF files 45, 101
 - setting units 46
 - using Merge option 111
 - using replacement libraries 109
 - with material map 109
- Indirect Source Minimum 126
- Initialization Minimum Area 128
- initiation 81
- insertion point 61, 229
- intensity mapping 160
- interactive
 - Move 63
 - Rotate 66
- isolating blocks 108
- IVR file format 212

J

- job 252
- job dependency 254

JobQ 251
JumpStarter 251

K

keyframes
 adding 135
 handles 135

L

layer 0 226
Layer State file 175, 226
layers
 for alternate layouts 221
 for block surfaces 223
 making current 58
 selecting patterns 154
 turning on and off 57

Layers table 11

Level of Detail 30

libraries

 Lightscape 69
 of blocks 69
 of luminaires 164
 of materials 20

lighting analysis 33, 93

lights, turning off 92

Look paths 139

LSnet application 251

luminaires

 aiming 22, 236
 changing properties 35
 duplicating 239
 nested 238
 photometric properties 68
 previewing 12
 self-emitting 240

Luminaires table 12

Luminance (Glow) 240

LVu utility 41

M

mapping modes 120
material maps 101, 150
 creating 105
 using 109

materials

 creating 17
 library 20
 previewing 11
 properties 73
 renaming 74

Materials table 11

Maximum Mesh Spacing 126

maximum reflectance 18

measuring distance 48

merging

 Solution files 194

mesh 29

Mesh Resolution 167

Mesh to Texture 198

meshing

 adaptive 124

Minimum Mesh Spacing 126

modeling practices 147

Most Recently Used List 55

mouse

 buttons 11

 modes 11

moving blocks 63

MS-DOS Prompt 246

N

near clipping plane 50

nested luminaires 238

night solutions 174

No Mesh 167

node 251

Notepad 246

O

Occluding 167

OpenGL 39

opening 87

Orbit view control 53

orbiting 14

orienting surfaces 113

origin 61

orthographic projections 49

Outlined display mode 14

P

- padding textures 206
- Pan view control 15, 53
- panning 15
- panoramic files 212
- paths
 - for luminaire distributions 236
 - for textures 18
- Perspective projection 49
- photometric properties 36
- Pick Light 240
- Pick mouse mode 23
- Pick Top Block 235
- pivot point 230
- plan projection 49
- Preparation files 3
 - opening 9
 - saving 80
- procedural textures 160
- processing 81
- processing parameters
 - guidelines 127
 - saving and loading 130
 - setting global 26, 124, 172
 - setting local 167
 - wizard 27
- progressive refinement 28, 82
- projected geometry 200
- projection methods 205
- projections 14, 49

Q

- Query mode 11, 224

R

- radiosity 1
 - image 37
- Ray Trace Area 40
- Ray Trace Direct Illumination
 - for luminaires 38
 - for skylight 180, 183
- ray tracing 38
 - for procedural textures 187
- Receiving 167

- reflectance 18
- Reflecting 167
- Refractive Index 73
- renaming
 - blocks 218
 - materials 74
- rendering 94
- Rotate view control 53
- rotating
 - blocks 64

S

- Saturation 73
- scaling blocks 64
- Select Area All Vertices 60
- Select Area Any Vertex 60
- Select mode 11, 24
- Select Pattern 154
- selecting
 - blocks 61
 - luminaires 22
 - surfaces 59
- shadows
 - from the sun 178
- Shininess 73
- Show Axis option 13
- Smoothing 118
- Solid display mode 14
- solstices 182
- Solution files 3
 - merging 194
 - saving 97
- Source Subdivision Accuracy 126
- Special Selection mode 11
- specular material 73
- speed graph 135
- stages of workflow 2
- statistics
 - lighting 33
 - processing 84, 132
- Subdivision Contrast Threshold 126
- summer solstice 182
- sun studies 182

- surfaces
 - orientation 113
 - replacing with luminaires 165
 - selecting 59
 - smoothing 117
- swapping blocks 218

T

- target geometry 200
- technical support 6
- textures
 - aligning 119, 170
 - assigning to material 18, 32
 - displaying 21
 - using Mesh to Texture 198
- Textures display control 21
- Tilt view control 53
- toolbars 13
- Transparency 73

U

- Undelete 25
- Undo Zoom Window 16
- units of measurement
 - measuring 47
 - specifying 46
- Use Drag Increments 63
- user interface 10
- utilities 244

V

- Value 18, 73
- vertex 81
- view controls 14
 - concepts 53
 - hot keys 16
- View Extents 13
- view frustrum 50
- View Setup 49
- viewer position 50
- views
 - creating 51
 - opening 54
 - saving 54
- VRML file
 - exporting 214

W

- walk-throughs 134
- window 87
- winter solstice 185
- Wireframe display mode 14
- wizard
 - for Mesh to Texture 198
 - for processing parameters 27

Z

- Zero Energy State 26
- Zoom view control 53
- Zoom Window 16

Acknowledgements

We are pleased to acknowledge the following manufacturers which have licensed digital representations of their products for the Lightscape libraries:

Luminaires

Bega, 1005 Mark Ave., Carpinteria, CA 93013 (www.bega-us.com)

Erco Leuchten GmbH, Brockhauser Weg 80-82, D-58507 Ludenscheid, Germany (www.erco.com)

Kurt Versen Company, 10 Charles St., P.O. Box 677, Westwood, New Jersey 07675

Lithonia Lighting, 1400 Lester Rd., Conyers, GA 30207 (www.lithonia.com),

including the Peerless Lighting line (www.peerless-lighting.com)

Additional luminaires are available at www.professional.erco.com

Materials

Appalachian Millwork & Lumber Co., 8230 Expansion Way, Huber Heights, OH 45424

Mannington Carpets, Inc., P.O. Box 12281, Calhoun, GA 30703 (www.mannington.com)

Marble and Granite, Inc., 29 Tower Road, Newton MA 02464 (www.marbleandgranite.com)

National Terrazzo and Mosaic Association, 110 East Market St., Leesburg VA 20176 (www.ntma.com)

Cover Image Credit

Louis I. Kahn's unbuilt Palazzo dei Congressi, Venice, Italy

Lightscape image by Kent Larson

From the book *Louis I. Kahn: Unbuilt Masterworks*, by Kent Larson

Monacelli Press

<http://www.monacellipress.com>