



Getting Started with Maya Unlimited 6



© Copyright 2004 Alias Systems, a division of Silicon Graphics Limited ("Alias"). All images © Copyright Alias unless otherwise noted. All rights reserved.

Printed in the USA.

Alias is a registered trademark and the swirl logo, the Maya logo, Conductors, Trax, IPR, Maya Shockwave 3D Exporter and MEL are trademarks of Alias in the United States and/or other countries worldwide. Maya is a registered trademark of Silicon Graphics, Inc. in the United States and/or other countries worldwide, used exclusively by Alias. SGI, IRIX, Open GL and Silicon Graphics are registered trademarks of Silicon Graphics, Inc. in the United States and/or other countries worldwide. mental ray and mental images are registered trademarks of mental images GmbH & CO. KG. in the United States and/or other countries. Lingo, Macromedia, Director, Shockwave and Macromedia Flash are trademarks or registered trademarks of Macromedia, Inc. Wacom is a trademark of Wacom Co., Ltd. NVidia is a registered trademark and Gforce is a trademark of NVidia Corporation. Linux is a registered trademark of Linus Torvalds. Intel and Pentium are registered trademarks of Intel Corporation. Red Hat is a registered trademark of Red Hat, Inc. ActiveX, Microsoft and Windows are either registered trademarks or trademarks of Microsoft Corporation in the United States and/or other countries. Mac, Macintosh and QuickTime are trademarks of Apple Computer, Inc., registered in the United States and other countries. Adobe, Adobe Illustrator, Photoshop and Acrobat are either registered trademarks or trademarks of Adobe Systems Incorporated. UNIX is a registered trademark, licensed exclusively through X/Open Company, Ltd. AutoCAD, Discreet Logic, Inferno and Flame are either registered trademarks or trademarks of Autodesk, Inc. in the USA and/or other countries. OpenFlight is a registered trademark of MultiGen Inc. Java is a registered trademark of Sun Microsystems, Inc. RenderMan is a registered trademark of Pixar Corporation. Softimage is either a registered trademark or trademark of Avid Technology, Inc. in the United States and/or other countries. All other trademarks, trade names, service marks, or product names mentioned herein are property of their respective owners.

This document contains proprietary and confidential information of Alias, and is protected by Federal copyright law and international intellectual property conventions and treaties. The contents of this document may not be disclosed to third parties, translated, copied, or duplicated in any form, in whole or in part, or by any means, electronic, mechanical, photocopying, recording or otherwise, without the express prior written consent of Alias. The information contained in this document is subject to change without notice. Neither Alias, nor its affiliates, nor their respective directors, officers, employees, or agents are responsible for any damages of any kind arising out of or resulting from the use of this material, including, without limitation, any lost profits or any other direct, indirect, special, incidental, or consequential damages or for technical or editorial omissions made herein.



ALIAS ■ 210 KING STREET EAST ■ TORONTO, CANADA M5A 1J7

Table of Contents

1 Introduction9
About Getting Started with Maya Unlimited.....	10
Before you begin	11
Installing Maya Unlimited.....	11
Conventions used in this book	12
Using the lesson files	13
Restoring default user settings.....	14
2 Hair	17
About hair simulation	20
Preparing for the lessons	20
Lesson 1 Creating a basic hairstyle	22
Lesson setup	23
Creating hair on a surface.....	24
Styling the hair.....	29
Setting up hair collisions.....	35
Rendering the hair	39
Modifying hair attributes.....	40
Setting up shadowing on hair	43
Beyond the lesson	45
Lesson 2 Creating a dynamic non-hair simulation.....	48
Lesson setup	49
Setting up the curtain scene.....	49
Making the hair collide with another object	53

| Table of Contents

Assigning a Paint Effects brush to the hair	54
Setting up constraints	55
Rendering the curtain scene	60
Beyond the lesson	61
3 Fluid Effects	63
Preparing for the lessons	64
Lesson 1 Creating a dynamic 2D fluid effect.	65
Creating a 2-dimensional fluid container.	65
Adding a fluid emitter to a container	66
Changing the behavior of a fluid	68
Combining colors in a fluid.	72
Colliding a fluid with an object.	74
Beyond the lesson	75
Lesson 2 Creating a non-dynamic 3D fluid effect.	77
Creating a 3D fluid container	78
Adding fluid to a container	79
Defining shader attributes for a fluid	80
Texturing the contents of a fluid container	81
Adding self shadowing to texture density.	86
Fluids Lesson 2: Beyond the lesson	87
Lesson 3 Creating a dynamic 3D effect	89
Creating a 3D container.	89
Painting Fuel and Density into a container	90
Painting Temperature values into a container	96

	Adding color to Density and Temperature	97
	Beyond the lesson	99
Lesson 4	Creating an ocean effect	101
	Creating an ocean plane and shader	102
	Adding a preview plane to an ocean	102
	Modifying ocean attributes	104
	Floating objects	108
	Beyond the lesson	110
4	Cloth	113
	About cloth animation	113
	Preparing for the lessons	116
Lesson 1	Making a Shirt	118
	Lesson setup	118
	Creating panels	119
	Creating a garment	122
	Seaming panels in a garment	123
	Creating and editing a property for a garment	126
	Making a 3D character a collision object	127
	Running a cloth simulation	128
	Increasing a garment’s resolution	130
	Changing the Collision Offset	131
	Beyond the lesson	132
Lesson 2	Making Pants	134
	Lesson setup	134
	Creating panel curves	135

| Table of Contents

Creating front and back panels.	136
Creating a garment	138
Seaming panels in a garment.	138
Increasing the resolution of panels	140
Creating a collision object.	140
Running and modifying a simulation.	140
Increasing the Base Resolution	142
Beyond the lesson	143
Lesson 3 Exporting & Importing Garments	144
Exporting garments	144
Positioning an imported garment	148
Creating a collision object for an imported garment	149
Importing solvers.	151
Transferring solvers.	152
Adding mesh constraints	153
Manually adjusting garments.	154
Running a simulation for an animation sequence	156
Saving the initial state for garments	158
Eliminating interpenetration.	159
Beyond the lesson	160
5 Fur	163
Preparing for the lesson	163
Lesson 1 Applying fur to a surface	164
Lesson setup	164
Creating fur on a surface.	166

	Modifying fur attributes	166
	Rendering fur	169
	Beyond the lesson	170
6 Live		173
	About Live	173
	Preparing for the lessons	176
	Lesson setup	178
Lesson 1 Track and solve		179
	Tracking preparation	179
	Tracking object points in a shot	182
	Evaluating the tracking of a point	185
	Tracking additional points	187
	Deleting tracking data	188
	Importing tracking data	191
	Preparing to solve	192
	Solving the shot	193
	Evaluating a solved solution	195
	Importing additional tracking data	198
	Live Lesson 1: Beyond the lesson	200
Lesson 2 Solving with survey data		203
	Creating a Distance constraint	203
	Creating a Plane constraint	204
	Registering a solution	206
	Creating additional Plane constraints	207

| Table of Contents

Evaluating the solution with imported geometry	209
Beyond the lesson	212
Index	215

1

Introduction

Maya Unlimited represents the total feature set of Maya software tools available from Alias. Maya Unlimited includes everything from Maya Complete plus additional tools.

Maya Unlimited includes Fluid Effects tools that allow for the simulation and rendering of viscous fluids, atmospheric, pyrotechnic, and ocean effects. Maya Cloth tools allow you to create simulations of fabric and clothing. Maya Fur lets you style and render short hair and fur effects using the Maya Artisan brush tools. Maya Live provides functionality to match original live-action footage with your 3D elements that were rendered in Maya.

The goal of this manual is to assist you in learning the features of Maya Unlimited.

If you have never used Maya Unlimited or Maya Complete, we recommend that you begin with *Getting Started with Maya* before attempting the lessons within *Getting Started with Maya Unlimited*.

This chapter provides the following information:

- About Getting Started with Maya Unlimited – Contents of the Getting Started with Maya Unlimited manual.
- Prerequisites – What you should know before using Getting Started with Maya Unlimited.
- Installing Maya Unlimited – Information on installing Maya Unlimited.
- Using the Getting Started with Maya Unlimited files – How to copy the files provided on CD-ROM for the Getting Started with Maya Unlimited lessons.
- Before you begin the lessons – Helpful instructions to aid your success with the lessons.

1 | Introduction

> About Getting Started with Maya Unlimited

About Getting Started with Maya Unlimited

Getting Started with Maya Unlimited introduces the modular areas of Maya Unlimited in a set of brief tutorials. The tutorials are designed to let you learn these modules at your own pace.

Getting Started with Maya Unlimited is not meant to replace the documentation that comes with the Maya software. Only the commands and options used in the lessons are explained in this manual. You will find the Maya Help provides an excellent companion reference to the lessons and much more.

Many of the tutorials in *Getting Started with Maya Unlimited* contain one or more separate lessons that provide step-by-step instructions for creating or accomplishing specific tasks within Maya. You can follow the tutorials in this book from start to finish or complete only the tutorials that correspond to your interests and needs.

Getting Started with Maya Unlimited is also available for use within the Maya Help. This version of *Getting Started with Maya Unlimited* contains mpeg movies for some of the lessons.

To use the tutorials from the Maya Help

- 1 In Maya, select Help > Tutorials.
The Maya Help window displays the *Getting Started with Maya Unlimited* tutorials.
- 2 Click the tutorial you want to work through.
The Maya Help displays the associated lessons for that tutorial.

Before you begin

We recommend that any new Maya user begin by completing the following:

- Viewing the Essential Skills Movies that are available when you first start Maya.
- Completing the Maya Basics tutorial (Chapter 2) in *Getting Started with Maya* which introduces many fundamental concepts and skills related to the Maya user interface.
- Completing the tutorials within *Getting Started with Maya* before attempting the lessons within *Getting Started with Maya Unlimited*.

Beyond that, you can attempt the tutorials in any order you want.

If you require an overview or review of basic mouse, keyboarding, and related usage techniques, we recommend that you refer to the documentation that came with your particular computer and operating system.

If you are new to 3D computer graphics and animation, you may want to obtain *The Art of Maya* (ISBN: 1-8948931-3-1) available from the Alias online store. It explains many concepts and techniques that are unique to the world of 3D computer graphics as they relate to Maya.

Installing Maya Unlimited

You must have Maya Unlimited installed and licensed on your computer system to successfully complete the lessons in this manual. To operate Maya Unlimited on your computer you must be running a qualified Windows, IRIX, or Linux operating system with the recommended minimum memory and storage requirements. Maya requires a three button mouse to access its full functionality for menus, commands, and 3D viewing.

1 | Introduction

> Conventions used in this book

For complete instructions on qualified hardware and operating systems, as well as installation and licensing of the Maya software, please refer to the *Installation and Licensing* manual that came with your Maya software or see the Support section of our Web site at www.alias.com.

Conventions used in this book

Some important conventions used throughout Getting Started with Maya Unlimited are explained here.

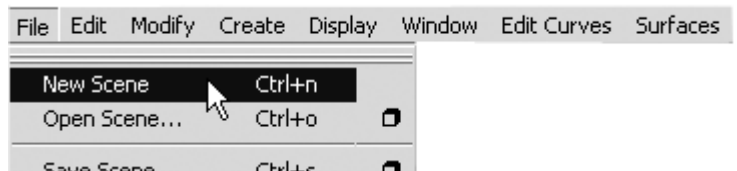
Maya Unlimited is available for use on a wide range of operating systems. Any differences between operating systems when operating Maya are identified in the following ways:

(Windows), (IRIX), & (Linux)

The screen illustrations and examples within Getting Started with Maya Unlimited vary among the Windows, IRIX, and Linux operating systems. Maya's interface is generally consistent across these systems.

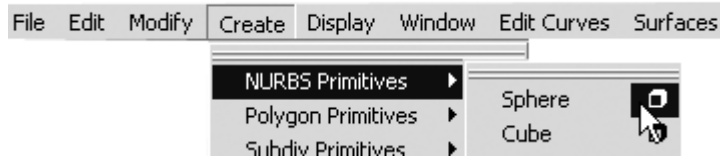
When instructed to select a menu within Maya, we use the following convention:

- Menu > Command (for example, File > New Scene)



When you are instructed to select the option box for a particular menu item within Maya, we use the following convention:

- Menu > Command > Option (for example, Create > NURBS > Primitives > Sphere >)



Using the lesson files

Some of the Getting Started with Maya Unlimited lessons have accompanying Maya files that were created for use with the lessons. These files are contained on the *Maya Documentation, Lessons, and Extras* CD-ROM which is included with your Maya Unlimited software package. To use the lesson files, do one of the following:

- Open the Getting Started with Maya Unlimited lesson files directly from the *Maya Documentation, Lessons, and Extras* CD-ROM.
- Copy the Getting Started with Maya Unlimited lesson files to your computer's hard drive and then open them from your hard drive.

To open the lesson files from your CD-ROM drive

- 1 Insert the *Maya Documentation, Lessons, and Extras* CD-ROM into your drive.
- 2 Within Maya, select File > Open Scene.
- 3 Navigate to the CD-ROM drive, locate the Getting Started with Maya Unlimited lessons folder, and select it.
- 4 Double-click on the Getting Started with Maya Unlimited lessons folder to view its contents.
- 5 Double-click the specific lesson and tutorial file to open it in Maya Unlimited.

1 | Introduction

> Restoring default user settings

To copy the lesson files from the *Maya Documentation, Lessons, and Extras* CD-ROM

- 1 Insert the *Maya Documentation, Lessons, and Extras* CD-ROM into your drive.
- 2 From the CD-ROM, copy the Getting Started with Maya Unlimited lessons folder to your computer into the following directory:

(Windows) `drive:\Documents and Settings\username\My Documents\maya\projects`

(IRIX & Linux) `~username/maya/projects`

When you are asked to open specific lesson files, open the Getting Started with Maya Unlimited lessons that you have copied to the *projects* directory.

Restoring default user settings

If you have already used Maya or have a prior version of Maya installed, you should restore the default settings for Maya before you begin the lessons. This ensures that Maya appears and operates exactly as the lessons describe.

If you are an existing user of Maya, we recommend that you save your existing preferences for later use prior to restoring the default user settings.

To save your existing custom user preferences

- 1 Ensure Maya is not running.
Each time you exit Maya, it saves the configuration of most components of your user interface so it appears the same when you start it the next time. It writes the preferences to a directory called *prefs*. If you rename the *prefs* directory, your original preferences will be maintained and Maya will create a new *prefs* directory the next time it is run.

1 | Introduction

> Restoring default user settings

- 2 Rename your existing user preferences file to a name, for example, `myprefs`. The prefs directory path is:

(Windows) `drive:\Documents and Settings\username\
My Documents\maya\6.0\prefs`

(IRIX & Linux) `~username/maya/6.0/prefs`

If you have a previous version of Maya installed, also rename that prefs directory to a new name such as `myprefs`. Maya will load older preferences if they exist from a previous version.

- 3 Start Maya and begin the Getting Started with Maya lessons.

To restore your custom user preferences after doing the lessons

- 1 Ensure Maya is not running.
- 2 Rename the previously changed preferences back to `prefs`.

1 | Introduction

> Restoring default user settings

2

Hair



With Maya Hair, you can create a dynamic hair system to model realistic hairstyles and hair behavior.

Since Hair is a generic dynamic curve simulation, you can also use these curves to create non-hair effects, including ropes, chains, a suspension bridge, sea creatures, or lofted surface from a curve.

With Hair you can simulate:

- natural movement and collisions of long hair
- hair blowing in the wind or being blow dried
- hair when swimming underwater

2 | Hair

>

- various hairstyles, including ponytails, braids, and updos
- other dynamic curve effects such as ropes, chains, cables, wires, etc.

A *hair system* is a collection of hair *follicles*.

A hair follicle in human hair typically hosts one hair, whereas in Maya each hair *follicle* hosts a hair curve. The hair *follicle* controls the attributes and curves associated with a particular hair clump, and how the hairs attach to a NURBS or polygonal surface. The input to each follicle is a Start Position NURBS curve, a surface and a UV position to attach to.

When creating hair, the hair system *output* can be in the form of NURBS curves, Paint Effects strokes, or both. If the specified output when creating hair is NURBS Curves, each follicle will contain one NURBS curve that represents the position of the hair in that follicle. If the specified output is Paint Effects, each follicle has a hair clump, made up of Paint Effects strokes.

There are various attributes on a hair system for modifying the look and behavior of the hair as a whole. There are also follicle level attributes that will override the hair system attributes; the visible result will be affected by both the follicle and hair system attributes.

You can have more than one hair system on the same surface.

With the Paint Hair Tool you can create new hair systems, add and remove follicles, as well as paint hair attributes, including Clump Width Scale, Stiffness, and Braid.

Paint Effects hair can be rendered using the Maya Software renderer. You can also convert Paint Effects hair to polygons and render in another renderer, such as mental ray. Or you can output just the dynamic NURBS curves to an external renderer, such as Renderman.

Consider outputting to Paint Effects if you want to:

- render with the Maya Software renderer
- convert the Paint Effects hair to polygons and render in any renderer

Consider outputting to NURBS curves if you want to:

- use the curves as path or control curves for any standard Paint Effects brush, such as feathers or vines. These brushes behave differently than the default built in Paint Effects as the curves are not the hairs, but controls that interpolate and affect Paint Effects. This scenario is slower to update and takes a bit of time to set up.
- select a few Current Position curves and loft a surface through them with construction history.
- extrude a circle down the curve to create a tube.
- use the curves for output to another renderer that directly supports curve rendering.

Consider outputting to both NURBS curves and Paint Effects if you want to:

- use both the curves (for example, as particle emitters to simulate spray flying off wet hair) and render using the integrated Paint Effects either directly or by converting to polygons.

2 | Hair

> About hair simulation

About hair simulation

This is the general workflow to create and animate hair:

- create hair on models
- style hair curves
- modify hair attributes
- set up hair shading/shadowing
- render the scene

This chapter includes these lessons:

- "Creating a basic hairstyle" on page 22
- "Creating a dynamic non-hair simulation" on page 48

Preparing for the lessons

To ensure the lesson works as described, do these steps before beginning:

- 1 Make sure you understand the basic concepts of modeling NURBS curves, animation, and dynamics. For more information, see Maya Help.
- 2 Select the Dynamics menu set.
Unless otherwise noted, the procedures in these lessons assume the Dynamics menu set is selected.
- 3 Select File > New Scene to create a new scene before you start each lesson. For those scenes that require you to open a file, the Hair scene files are located in `UnlimitedLessonData/Hair/scenes` on the *Maya Documentation, Lessons, and Extras* CD or on the drive where you copied the Getting Started with Maya data files.
- 4 Select Window > Settings/Preferences > Preferences. Click Timeline under Categories and make sure the Playback Speed is set to Play every frame. Dynamic animation plays more accurately with this setting.

Also set the playback and animation end times to 10000. This ensures you have lots of time to interact with the playback of the hair simulation.

- 5 We provide a completed scene for each of the lessons in this chapter. If you want to see the expected results of the lessons, you can open the scenes from `UnlimitedLessonData/Hair/scenes` on the *Maya Documentation, Lessons, and Extras* CD or from the drive where you copied the Getting Started with Maya data files. The scene filenames end with `Final.mb`, for instance, `hairFinal.mb`.

Lesson 1 Creating a basic hairstyle

You can create hair on NURBS and polygonal surfaces. For polygons, UVs should be non-overlapping and fit between 0 and 1. Automatic mapping is a quick way to achieve this. (See *Map UVs onto polygons automatically* in the Polygonal Modeling guide in Maya Help.) In this lesson you are creating hair on a head model.

There are three sets of curves for the hair system.

Start Curves/ Position	This is the position of the hair at the start frame of the simulation. When first created, the Start curves stick straight out from the surface.
Rest Curves/ Position	<p>This is the position of the hair when no forces (such as gravity) are acting upon it. You can use these curves to influence the shape of the hair.</p> <p>Working with Rest curves is like styling hair with hairspray. The Rest curves are styled into curls and flips, etc. When the simulation is played back, the hair adheres to the Rest curves position as much as possible depending on dynamic forces and attribute settings in the hair system and follicles. For example, if your hair is not very stiff and you're simulating a windy day, the hairspray (Rest curves position) is less likely to hold the hair in position.</p>

Current Position	This is how the hair behaves when you play the simulation, which includes dynamics. Do not edit the Current Position; just view it. The Stiffness and Iterations attributes in the Hair system's Dynamics section are what control the tolerance of the Current Position achieving the Rest Position.
------------------	--

In this lesson you are introduced to some of the basic concepts of hair by creating a simple long hairstyle. In this lesson, you learn how to:

- Create hair on a surface.
- Play a hair simulation and interact with it during playback.
- Manipulate hair curves to create a hairstyle.
- Use collision constraints to make hair collide with a model's head, neck and shoulders.
- Modify hair attributes.
- Set up hair self shadows.
- Render hair.

Lesson setup

In this lesson, you'll work with a scene we created for your use.

- 1 Before beginning the lesson, do the steps in "Preparing for the lessons" on page 20.
- 2 Open the scene named `hairStart.mb` from the *Maya Documentation, Lessons, and Extras CD* or from the drive where you copied the Getting Started with Maya Unlimited data files. The file is located in `UnlimitedLessonData/Hair/scenes`.

2 | Hair

Lesson 1 > Creating hair on a surface

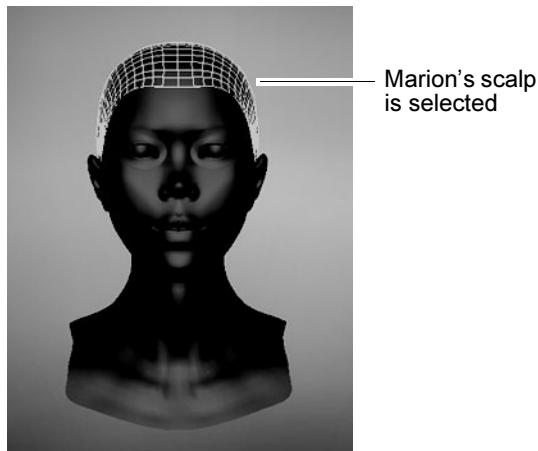
This scene provides a NURBS head model, named Marion, and the scalp surface for creating the hair. Part of the model has been templated to make picking the scalp easier.

Creating hair on a surface

When you first create hair it appears sticking out normal to the surface. For a more natural look, play the hair simulation to see the hair react to gravity and then you can set a more realistic Start position for the hair. In these steps you create hair on a NURBS model and set a Start position after playing the simulation. You use Interactive Playback to play the simulation while adjusting dynamic hair system attributes and see how your changes affect the simulation.

To create hair on a model

- 1 Select Marion's scalp, as shown in the image below.



- 2 Select Hair > Create Hair > and in the Create Hair Options window, specify the following:
 - Set Output to Paint Effects.
 - Set the U Count to 26 and the V Count to 22. These determine how many hair curves within each UV range.
 - Set the Points Per Hair to 10. This sets the number of segments per hair. More segments are needed to make long hair look natural or to create complex short hairstyles. As you increase the number of segments, you decrease the performance (speed) of the simulation.
 - Set the Passive Fill to 0. No passive curves will be used in this tutorial. The behavior of passive curves is interpolated between neighboring active curves. For more information, see *Passive Curves* in the online Hair and Dynamic Curves guide.
 - Set the Length to 10. This value is relative to the world space units.
 - Set the Randomization to 0.5. A value of 1 represents total randomization of placement of hair curves on the surface, whereas a value of 0 represents no randomization.
 - Both the Create Rest Curves and Edge Bounded options should be turned off.
 - Turn the Equalize option on. This ensures the hair curves are evenly distributed over the surface.
 - Click Create Hairs.

The hair is attached to the scalp and sticks out perpendicular to the surface. You are looking at the *Current Position* of the hair, which is also referred to as the “dynamic” position. Do not edit the hair curves in this view; edit only in *Start Position* or *Rest Position*.

2 | Hair



Lesson 1 > Creating hair on a surface



When Marion's hair is first created it is perpendicular (normal) to the scalp. The red circles are the follicles.

To play the hair simulation

Note Make sure the End Time and Playback End Time in the Range Slider are both set to 10000.

- 1 Click the Play  button to play the simulation.
The hair falls due to default dynamic forces, such as gravity, which are found in the hair system attributes. However the hair takes a while to fall, which is as a result of a high Display Quality setting.
- 2 Click the Stop  button to stop the simulation when the hair is relaxed as it is in the following image.



After playing the simulation for a while Marion's hair begins to relax, but it is still quite stiff and doesn't rest near the scalp as hair does naturally.

Notice the hairs are quite stiff and stick out from the scalp instead of naturally falling closer to the scalp. To make the hair appear less stiff you can either increase the number of Points Per Hair (segments), which increases the simulation and render time, or you can reduce the Stiffness value in the hair system's Dynamic attributes.


In the next part of the lesson you adjust dynamic settings of the hair system, including Stiffness and Gravity, to achieve more naturally behaving hair.

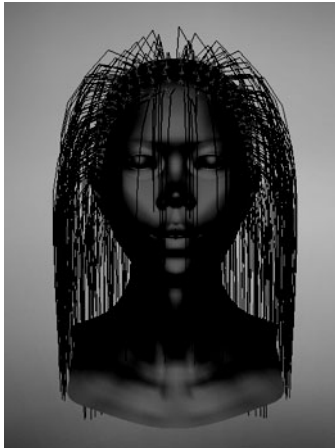
To improve playback performance and modify dynamic settings

- 1 In the Attribute Editor (Ctrl + a) select the hairSystemShape tab and change Display Quality to 2. This affects how many hairs display in the scene view, but has no effect on how much hair appears in the software render. By reducing the Display Quality, this speeds up the playback of the hair simulation.

2 | Hair


Lesson 1 > Creating hair on a surface

- 2 In the Dynamics section the default value for Stiffness is 0.15. Since the hair is quite stiff, reduce the Stiffness value to 0.07.
- 3 Select Solvers > Interactive Playback to play the simulation. Watch the hair begin to relax more.
- 4 With the simulation still playing go to the Dynamics section of the hairSystemShape tab and change the Gravity value to 2.5. Watch the hair relax even more and rest closer to the scalp, as hair does naturally.
- 5 Click the Stop  button when the hair has relaxed as in the following image.



To set the position of the hair for the start frame



- 1 When you are playing the hair simulation, by default you are viewing the Current position curves. To select all the Current position curves, click any hair curve and then select Hair > Convert Selection > to Current Positions.
- 2 To set the start frame of the hair simulation to how the hair appears now, select Hair > Set Start Position > From Current.

- 3 Select Hair > Display > Start Position to view the Start curves and see that they took on the Current Position.
- 4 Click the  button to rewind to the start frame.
The position of the hair at the start frame is now relaxed.

Styling the hair

To style the hair you manipulate either the Start or Rest curves to achieve the look you want. In these next steps you first edit the Start curves to create the basic hair shape. Then you set an initial Rest position for the hair and style the hair by editing the Rest curves.



To edit the Start curves to create the basic hair shape

- 1 Display the Start curves. (Hair > Display > Start Position)
- 2 Tumble the camera so you're looking at the side of Marion's head.
- 3 To pick the Start curves on the front half of Marion's face, do the following:
 - Click the Object Selection Mask  button on the Status Line and select All Objects Off.
 - Click the  button on the Status Line to restrict the selection to curves.
 - Drag around the front of Marion's head. Only those Start curves are selected, as shown in the following image.

2 | Hair

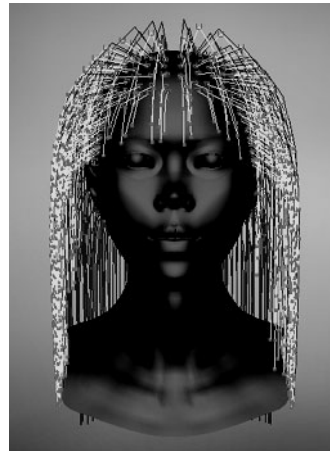
Lesson 1 > Styling the hair



- 4 Now tumble the camera so Marion is again facing you.
- 5 To trim the Start curves in the front, do the following:
 - Select Hair > Modify Curves > Lock Length. This allows you to manipulate the curve while maintaining the curve's hull length.
 - Click the  in the Status Line to display the curves' CVs.
 - Using the Lasso Tool , select CVs in an inverted U-shape around the front of Marion's face, as shown in the following image.
 - Press Delete to remove the selected CVs.






Start curve CVs selected in an inverted U-shape



Selected CVs deleted to trim hair

6 To curl under the ends of all the Start curves, do the following:

- Click the  button to display the curves and then drag around Marion's head to select all the Start curves.
- Click the  button to display CVs and then, using the Select Tool, select the CVs in the bottom half of all the curves.
- Select Hair > Modify Curves > Bend > .
- In the Bend Curves Options window, set Bend Amount to 0.5 and Twist to 1.0 and then click Bend Curves.



2 | Hair

Lesson 1 > Styling the hair



Selected CVs
are curled
under using
Bend




The selected CVs are bent under towards Marion's neck. To fix any unruly hairs, you could select them and then select **Hair > Modify Curves > Smooth**.

- 7 Click the  button to display the curves.
- 8 Change the display to include the Start curves and the Current Position. (**Hair > Display > Current and Start**)
- 9 Click the Play  button to play the hair simulation.

Even though you styled the Start curves, notice how they fall straight and lose the bend at the end. Start curves only define the position of the hair at the Start frame of the simulation; once the simulation is started the Start curves are affected by the dynamic forces applied to the hair system. To “hairspray” a hairstyle, you need to style Rest curves and adjust dynamic hair system settings. In the next steps you create Rest curves from the Start curves.

- 10 Click the Stop  button to stop the simulation.

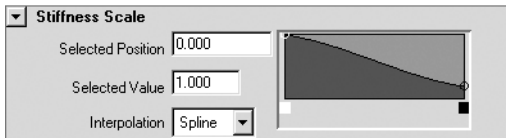
To create and modify the Rest curves to achieve a hairstyle

- 1 To create Rest curves from the shaped Start curves, do the following:
 - Select Hair > Display > Start Position.
 - Using the Select Tool , drag to select all the Start curves and then select Hair > Set Rest Position > From Start.
- 2 To see the new Rest curves, select Hair > Display > Current and Rest.
- 3 Select all the hair curves and then click the Play  button to play the simulation. Watch how the Current position hairs flop down because the Gravity value is quite high (2.5).
- 4 To see what the hair would look like without any gravity, go to the Dynamics section in the hairSystemShape tab and change the Gravity value to 0.
Watch the hair try to achieve the Rest position.
- 5 After the hair reaches the Rest position, change the Gravity value back to 1.0.
- 6 As the hair starts to fall due to gravity, increase the Stiffness value to 0.2.
If some Rest curves won't relax and stabilize, increase Length Flex and Iterations in the hairSystemShape tab.
- 7 Once the hair stabilizes, click the Stop  button to stop the simulation.
- 8 You can further "hairspray" the curled ends of the Rest curves using the Stiffness Scale attribute. The curve in the Stiffness Scale graph illustrates the amount of stiffness applied along the hair curve from root (left side of the graph) to tip (right side of the graph). Go to the Dynamics section in the

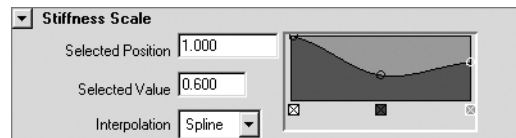
2 | Hair

Lesson 1 > Styling the hair

hairSystemShape tab and change the Stiffness Scale attribute as shown in the following image. These changes add more stiffness to the end of the hair, which will hold the bend more firmly.




Default Stiffness Scale



Modified Stiffness Scale:

- click the middle of the graph to add a new entry and set its Selected Value to 0.4
- click the entry on the right side of the graph and change its Selected value to 0.6

- 9 Play the simulation to see the effect of the changes to Stiffness Scale.
- 10 Stop the simulation once the hair relaxes as in the following image.
- 11 To reset the Start curves from the simulation, select Hair > Set Start Position > From Current.
- 12 Click the  button to rewind to the beginning of the simulation.



All the hair curves are selected. The green curves represent the Current Position. The white curves represent the Rest Position.

Setting up hair collisions

Before trimming Marion's hair you may have noticed that some hairs intersected her face. As well hairs currently intersect the lower part of her neck. To fix this you set up hair collisions. You can set hair to collide with itself, other objects or the ground (grid). In the next steps you use collision constraints to prevent the hair from intersecting with Marion's head and neck.



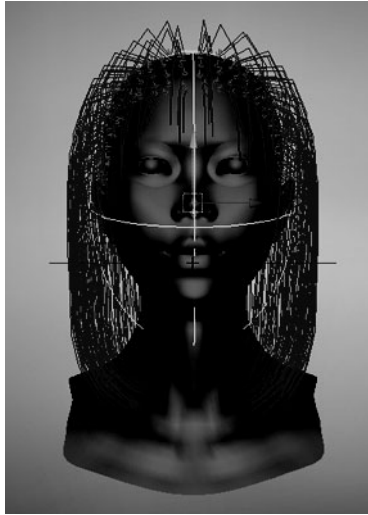
Notice the hairs at the bottom are intersecting with the upper part of Marion's torso

2 | Hair

Lesson 1 > Setting up hair collisions

To make Marion's hair collide with her head and neck

- 1 In the panel, select Show > Locators so you can see collision constraints, which are locators, when you create them.
- 2 To create the collision constraint for Marion's head, do the following:
 - Select Hair > Display > Start Position.
 - Using the Select Tool, drag around all the hair curves and select Hair > Convert Selection > to Start Curves.
 - Select Hair > Create Constraint > Collide Sphere.
 - Using the Scale Tool, resize the sphere so it expands beyond Marion's head and you can see it.
 - Using the Move Tool, move the sphere up the Y-axis so the top of the sphere is aligned with the top of Marion's head.
 - Using the Scale Tool again, resize the sphere so it matches the basic shape of Marion's head. First try scaling in the X direction to adjust the width of the sphere from ear to ear. Then tumble the camera to see the side of Marion's head and scale in the Z direction to adjust the sphere width from the back of Marion's head to her nose. You may also need to move the sphere to tweak its size and position, as shown in the images below.



See the sphere width from ear to ear and height from the top of Marion's head to below her chin



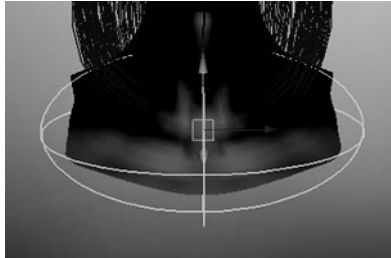
See the sphere width from the back of Marion's head to her nose

- 3 To create the collision constraint for Marion's upper torso, do the following:
 - Using the Select Tool, drag around the hair curves and select Hair > Convert Selection > to Start Curves.
 - Select Hair > Create Constraint > Collide Sphere.
 - Using the Scale Tool, resize the sphere so it extends beyond Marion's neck and you can see it.
 - Using the Move Tool, move the sphere down the Y-axis so the top of the sphere is aligned with the top of Marion's torso.
 - Using the Scale Tool again, resize the sphere so it matches the basic dimensions of Marion's upper torso. First try scaling in the X direction to adjust the width of the sphere from shoulder to shoulder. Then tumble the camera to see

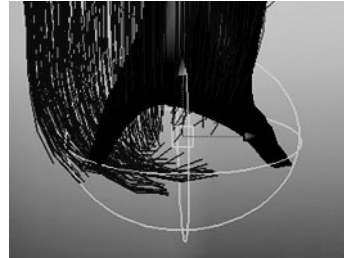
2 | Hair

Lesson 1 > Setting up hair collisions


the side of Marion's head and scale in the Z direction to adjust the sphere width from the back of Marion's torso to the front of it. You may also need to move the sphere to tweak its size and position, as shown in the images below



See the sphere width from shoulder to shoulder and height from the top of Marion's torso to the bottom of it



See the sphere width from the back of Marion's torso to the front of it

- 4 Select Hair > Display > Current Position.
- 5 Click the  button to rewind to the start frame and play the simulation.

Watch how the hair collides with the collision constraints you created. The hair at the front now rests on the torso and the hair at the back curls under the torso, instead of intersecting it like it did before the collisions were set.

- 6 Stop the simulation when the hair is relaxed. Should the hair be unruly and not relax, you may need to increase the Stiffness or the Iterations in the Dynamics section of the hairSystemShape tab of the Attribute Editor.




Once the hair relaxes in the simulation, see how it rests against the collision constraints

- 7 Select Hair > Set Start Position > From Current so the hair respects the collisions at the start frame of the simulation.
- 8 Select Show > Locators to turn them off, which hides the collision constraints.

Rendering the hair

When you modify hair attributes, many of these changes are not visible in the scene view; the Paint Effects hair must be rendered in the Maya Software renderer. In the next steps you render Marion's hair as it is now, which allows you to see what aspects of the hair need work to make it look more realistic. Before rendering, be sure to set the appropriate Render Global settings.

To render Paint Effects hair using the Maya Software renderer

- 1 Modify the Render Globals settings for Paint Effects.
 - Click the Render Globals  button on the Status Line. The Render Global Settings window appears.
 - Select Maya Software from the Render Using drop-down list.

2 | Hair

Lesson 1 > Modifying hair attributes

- In the Maya Software tab, go to the Paint Effects Rendering Options section and turn on Oversample and Oversample Post Filter. These help achieve a better look for the Paint Effects rendering. The option Enable Stroke Rendering is already on by default.
 - Click Close to close the Render Global Settings window.
- 2 Before rendering be sure to display the hair system's Current Position. (Hair > Display > Current Position)

- 3 Click the Render Current Frame  button on the Status Line.

The hair model is rendered and should look something like the image below.




This is how the hair looks in the Maya Software render before modifying hair attributes. The hair is sparse at the crown, which can be fixed by increasing the Clump Width value.

Modifying hair attributes

You can modify the hair as a whole by modifying the hair system attributes. You can modify the look of individual hairs or clumps of hair by editing their follicle attributes. In the next steps, you modify various hair system attributes to make Marion's hair look fuller and more realistic. Then you add curl to Marion's hair.

To modify the thickness of the hair all over

- 1 To select the hair system, drag around the hair curves and select Hair > Convert Selection > to Hair Systems.
- 2 Go to the Clump and Hair Shape section in the hairSystemShape tab of the Attribute Editor and adjust the following:
 - Hairs Per Clump to 30, which controls the number of hairs per curve (hair clump). Increasing the number of Hairs Per Clump increases the rendering time.
 - Sub Segments to 2, which adds this many points between Segments/points and helps to smooth the hair curves.
 - Thinning to 0.5, which controls how much the hair thins out from root to tip.
 - Clump Width to 0.3, which controls the width of the hair clump from root to tip. To further modify the hair clump width, you could adjust the Clump Width Scale attribute, which allows you to vary the width along the hair from root to tip.
 - Hair Width to 0.02, which controls the width of each hair.
- 3 Click the Render Current Frame  button to render the hair to see the changes you've made to the hair.


2 | Hair

Lesson 1 > Modifying hair attributes



After modifying hair and clump attributes the hair looks more full and realistic

To make hair curly

- 1 With the hair system still selected, go to the Displacements section in the hairSystemShape tab of the Attribute Editor and set the following:
 - Curl to 1, which is the amount of curl displacement applied to each hair. The amount of displacement is relative to the hair width.
 - Curl Frequency to 15, which is the rate of curl. The higher the value is, the more curls there are.
- 2 Click the Render Current Frame  button to render the hair to see the changes you've made to the hair.



The rendered hair after curl attributes have been applied

Setting up shadowing on hair

Hair, by default, is translucent so you need to set up self shadowing, otherwise hair appears glowy, like nylon. The darker the hair color is, the more important the specular color and highlights become.

There are three directional lights in this scene. In the software render you did in the previous steps, Marion's hair appears blond and glowy, even though the default hair color is dark brown, as in the scene view. This apparent hair color discrepancy occurs because:

- hair is by default translucent (like a glass tube)
- the hair is being lit by all three lights, therefore the intensity of the light is tripled
- there are no shadows, so all the hairs are illuminated

In these next steps you set up shadows on all three lights.

To set up shadows and self shadows on hair

- 1 Select Window > Outliner.
- 2 In the Outliner menu, select Show > Objects > Lights.

2 | Hair

Lesson 1 > Setting up shadowing on hair

- 3 Select one of the three directional lights.
- 4 In the light's Attribute Editor, go to the Shadows section and in the Depth Map Shadow Attributes subsection, turn on Use Depth Map Shadows.

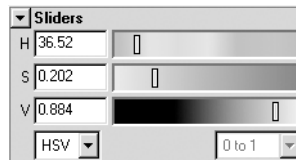
Normally you would increase the Dmap Filter Size to blur the light in the hair for realism, and increase the Dmap Bias, which sets how far the light filters through the hair. These attributes were set for you in the scene.

- 5 Repeat steps 3 and 4 for the other two directional lights.
- 6 Render the hair to see the changes.

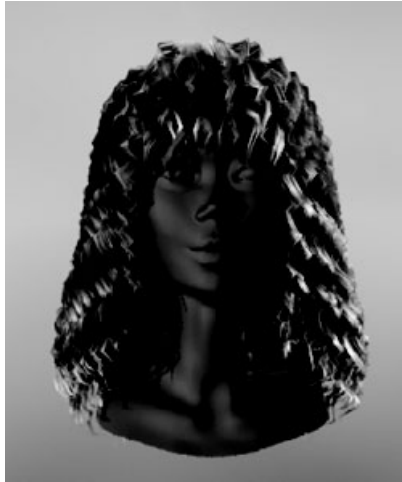
The shadows on the hair are hard and there is very little specularity. You fix this in the next steps.

To create specular highlights on hair

- 1 To select the hair system, drag around the hair curves and select Hair > Convert Selection > to Hair Systems.
- 2 Go to the Shading section in the hairSystemShape tab of the Attribute Editor and set the following:
 - Click the Specular color box and set the values as show in the image below.



- Increase Specular Power to 15.
- 3 Render the hair to see the changes. The hair looks much more natural now.



Your final render with the changes to the specular color and power should look something like this. Marion's hair is brown, but with a warm, golden specular highlight.

Beyond the lesson

In this lesson you learned how to:

- Create dynamic hair on a surface.

You can add hair to a NURBS or polygonal surface by creating or painting the hair. For polygons, UVs should be non-overlapping and fit between 0 and 1. Automatic mapping is a quick way to achieve this (see *Map UVs onto polygons automatically*). For more information, see *Creating hair* in the online Hair and Dynamic Curves guide.

Before creating hair, you should determine which renderer you'll be using as this will affect what type of hair output you select: NURBS curves or Paint Effects, or both.

- Interact with the dynamic playback.

Using interactive playback you can modify the hair and see the hair react to dynamic forces. For more information, see *Playing a Hair simulation* in the online Hair and Dynamic Curves guide.

2 | Hair

Lesson 1 > Beyond the lesson

- Model hair curves.
Style hair by transforming hair curves and using Lock Length to maintain the length between CVs on a curve. You can also use Fields to style hair and then add constraints to hold hair in position. For more information, see *Styling the hair* in the online Hair and Dynamic Curves guide.
- Modify hair system and follicle attributes.
You can change the look (color, curl, etc.) of the overall hair by modifying attributes in the hairSystemShape node. But individual follicle attributes can be modified in the follicleShape node, which override or blend with the hair system attributes. For more information, see the following topics in the online help: Hair system attributes (*hairSystemShape node*), Follicle attributes (*follicleShape node*) and *Setting up hair shading*.
- Render the hair.
Paint Effects hair can be rendered using the Maya Software renderer. You can also convert Paint Effects hair to polygons and render in another renderer, such as mental ray. Or you can output just the dynamic curves to an external renderer, such as Renderman. For more information, see *Rendering Scenes with Hair* in the online Hair and Dynamic Curves guide.

You can also create non-hair models and effects with Hair. Any NURBS curve can be made into a dynamic hair curve. In addition to creating long hair and hairstyles you can use hair curves to create:

- ropes and chains (dynamic curves)
- a fish (wire deformer)
- a fishing line (single hair with expressions)
- an octopus (select hair curves and create a lofted surface)

- a paint brush painting on canvas, bristles colliding with the surface (hair simulates paint brush bristles, Fluid emitters on the canvas controlled by expressions)

In addition you can assign a Paint Effects brush to a hair system as shown in the image below.



Image created by Duncan Brinsmead

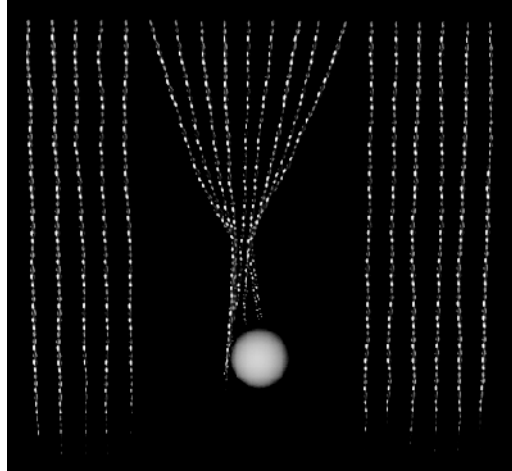
Lesson 2 **Creating a dynamic non-hair simulation**

Using Maya Hair, you can create dynamic non-hair models and effects. You can make NURBS curves dynamic, or you can use a hair system to create a non-hair simulation, such as chains or ropes. In this lesson, you create a hair system to simulate a beaded curtain. To make the hairs look like beads, you assign a Paint Effects brush to the hair. Then you modify the hair attributes and set constraints so the hair behaves like a beaded curtain hanging from a curtain rod.

In this lesson, you learn how to:

- Make hair collide with an object
- Assign a Paint Effects brush to hair
- Set up hair constraints

Lesson setup




In this lesson, you'll work with a new empty scene and change the playback range.

- 1 Select File > New Scene.
- 2 In the Time Slider change both the End Time value and Playback End Time value to 300.

Setting up the curtain scene

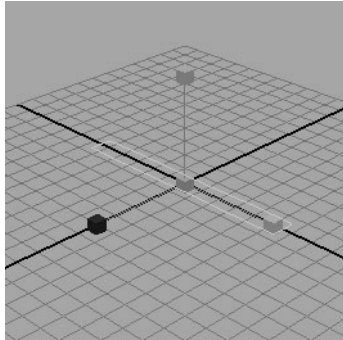
In these steps you model a 2D curtain rod from a plane and attach hair to it. Then you add and keyframe a sphere in the scene to later be used as a collision object.

To create the curtain using Hair

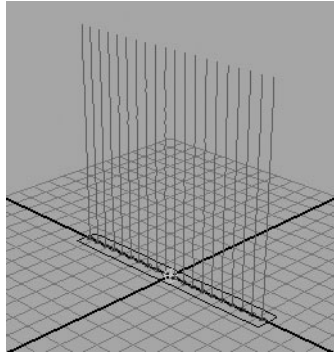
- 1 To create a NURBS plane, select Create > NURBS Primitives > Plane.
- 2 Using the Scale Tool , resize the plane in the X direction as shown in the image below.


2 | Hair


Lesson 2 > Setting up the curtain scene

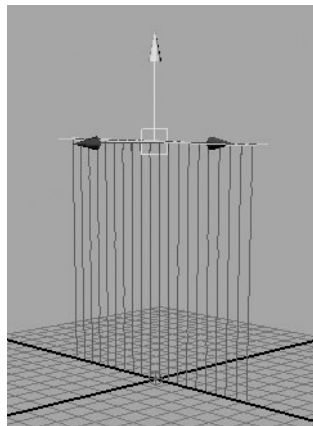


- 3 With the plane selected, select Hair > Create Hair > , set the following options:
 - Output to NURBS Curves
 - U Count to 20, V count to 1
 - Points per Hair to 12
 - Passive Fill to 0
 - Length to 10
 - Randomization to 0
 - Turn off the options Create Rest Curves, Edge Bounded and Equalize
 - and click Create Hair. Hair is created and attached to the plane.



- 4 Select the plane and, using the Rotate Tool , rotate the plane so the hair appears to be on the underside of the plane.

Then, using the Move Tool , move the plane up the Y axis so the bottom of the hair just touches the ground plane, as shown in the following image.

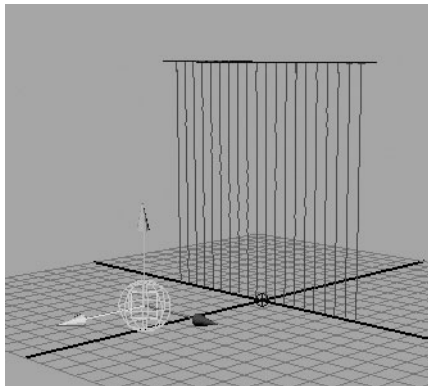


- 5 Create a directional light (Create > Lights > Directional Light) and do the following:

2 | Hair

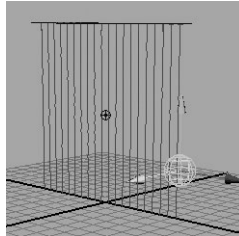
Lesson 2 > Setting up the curtain scene

- Move the light above and to the left of the curtain.
 - Rotate the light so it aims at the curtain.
 - In the light's Attribute Editor (ctrl + a) go to the Directional Light Attributes section and change the Intensity value to 1.5.
- 6 Create a NURBS sphere (Create > NURBS Primitives > Sphere) and move it along the Z-axis so it's in front of the curtain.



- 7 To set the start keyframe for the sphere, do the following:
- Go to frame 1 on the Time Slider.
 - In the Channel Box, drag to select the Translate X, Y, and Z channels.
 - Right-click the channels and select Key Selected.
- 8 Set the keyframe for the sphere having moved towards the curtain and passed through it, as in the image below.
- Go to frame 100 in the Time Slider.
 - Move the sphere in the Z direction as shown in the image below.
 - In the Channel Box, select the Translate Z channel.

- Right-click the channel and select Key Selected.



Time Slider = 100

- 9 Rewind the Time Slider to 0 and play the animation.

The sphere passes through the curtain as though the curtain weren't there. In the next steps you set the sphere as a collision object with the hair.

Making the hair collide with another object

To have hair interact with a surface, you must set the hair to collide with the surface. A collision object can be a NURBS or polygonal surface. In the next part of the lesson you set the curtain to collide with the sphere and then play the simulation.

To enhance the curtain for collisions

- 1 To select the hair system, drag around the hair curves and then select Hair > Convert Selection > to Hair Systems.
- 2 In the hairSystemShape tab of the Attribute Editor, set the hair system attributes as follows:
 - In the Clump and Hair Shape section, set Clump Width to 0.
 - In the Dynamics section, set Stiffness to 0.

2 | Hair

Lesson 2 > Assigning a Paint Effects brush to the hair

To make the hair collide with the sphere

- 1 Select the hair system and the sphere (drag around the hair curves and sphere).
- 2 Select Hair > Make Collide.
- 3 Play the simulation.

Now the sphere collides with the curtain, pushing the strands of hair aside as it passes through.

Assigning a Paint Effects brush to the hair


You can assign a Paint Effects brush to a hair system to achieve a variety of effects. In these next steps you apply the red beads brush to the hair system to create the beaded curtain. Then you modify the brush and hair system settings to achieve the desired look.

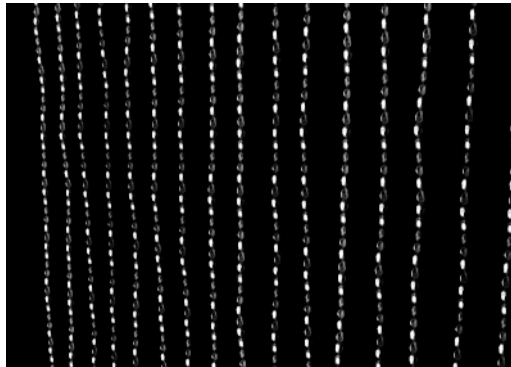
To turn the hair curtain into a beaded curtain

- 1 To select the hair system, drag around the hair curves and select Hair > Convert Selection > to Hair Systems.
- 2 Open the Visor. (Window > General Editors > Visor)
- 3 Go to the glass folder and select the beadsRed.mel brush.
- 4 Select Hair > Assign Paint Effects Brush to Hair.

You don't see the beads in the scene view, but you will when you render it later using the Maya Software renderer. The brush needs some adjustments to make it look like a beaded curtain when it is rendered.

- 5 In the hairSystemShape tab of the Attribute Editor, set the following:
 - Hairs Per Clump to 1 so there is one strand of beads
 - Hair Width to 0.03
 - in the Shading section, Hair Color to white so the brush's bead color appears

- in the Shading section, Hair Color Scale to white so the brush's bead color appears
 - in the Shading section, Specular Color to red
 - in the Displacements section, Noise to 0.5 to create some variety in the appearance and behavior of the beads
- 6 In the beadsRed tab of the Attribute Editor, go to the Texturing section and change the U Repeat to 10 so the beads appear as a more realistic bead size. (If the beadsRed tab does not appear in the Attribute Editor, re-select the hair curves. The beadsRed tab should now appear.)
 - 7 Dolly in close to the curtain (Alt + the left mouse button + the right mouse button) and click the  button to render the current frame.



The rendered beaded curtain should look something like this.

Setting up constraints


In these next steps you tie up the middle section of the beaded curtain by first modeling the Start curves into a bound position and then creating a constraint to hold the curves in that position

2 | Hair

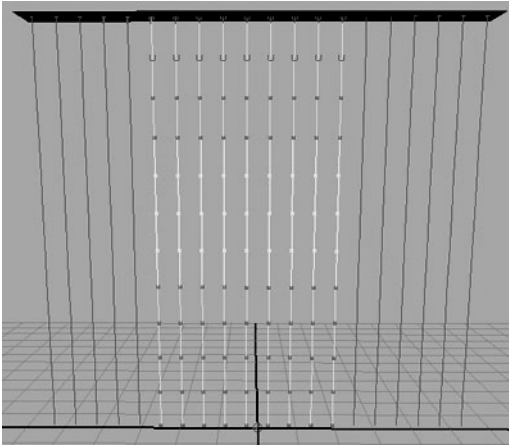
Lesson 2 > Setting up constraints

during playback. With a Hair to Hair constraint the hair curves stick together at the constraint locator, but also dynamically move from root to tip when forces or collisions occur.

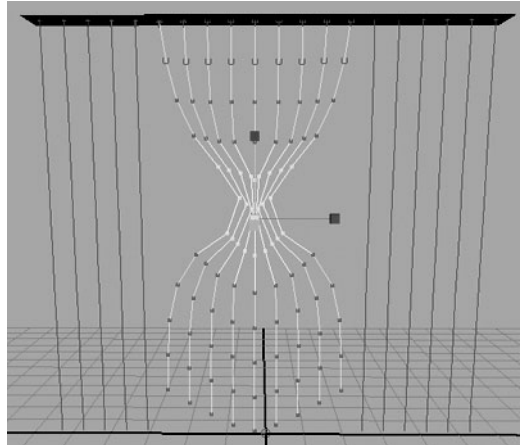
To model the curtain into a tied-back position

- 1 Dolly and tumble in the scene until you are facing the curtain directly.
- 2 Drag to select all the hair curves and then select Hair > Convert Selection > to Start Curves.
- 3 Select a block of Start curves in the middle of the curtain.
- 4 Select Hair > Modify Curves > Lock Length. This ensures the entire curve is modified when you transform CVs in the next steps.
- 5 Click the  button on the Status Line to display the CVs on the curves.

- 6 Select the middle three rows of CVs on the selected hair curves and then, using the Scale Tool, scale the CVs inwards so the hairs are pulled together.




The middle rows of CVs are selected



The middle rows of CVs are scaled inwards so the hair curves appear bound

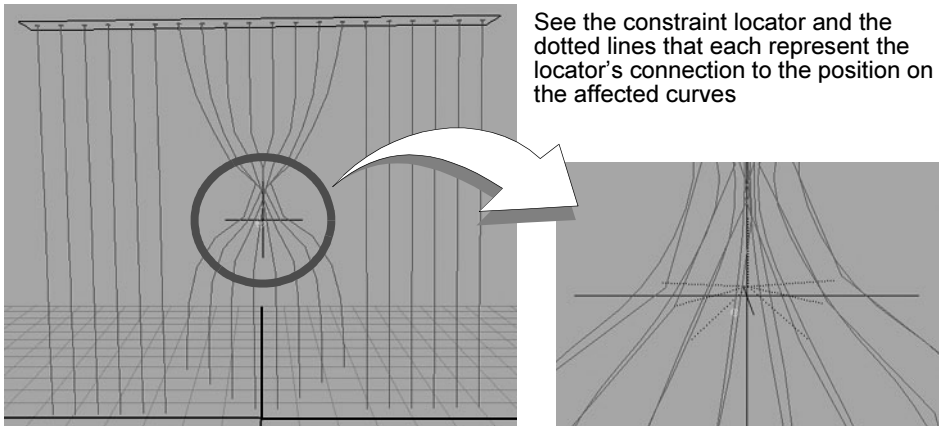
To constrain the curtain in the tied-back position

- 1 Click the  button on the Status Line to display the Start curves (not the CVs).
- 2 With the bound hair curves still selected, select Hair > Create Constraint > Hair to Hair.

2 | Hair

Lesson 2 > Setting up constraints

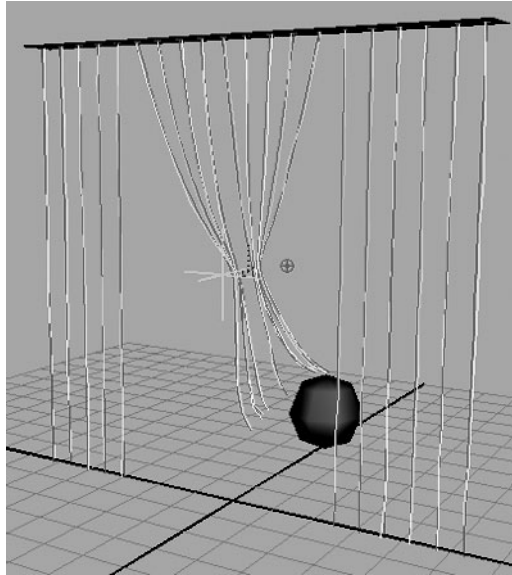
A constraint locator is created and connected to the selected curves.



- 3 In the `hairConstraintShape` tab in the Attribute Editor, set the constraint attributes as follows:
 - Stiffness to 0.5 (default)
 - Glue Strength to 1.0 (default)
 - Point Method to Nearest (default) (Since the constraint locator is created in the middle of the bound hair, the locator is “nearest” the bound area to be constrained.)
- 4 Change the hair display to the Current Position. (Hair > Display > Current Position)
- 5 Rewind to the start frame and play the simulation.

When the sphere passes through the bound curtain it collides with all the bound hairs.

The bound part of the curtain relaxes within the first several frames, but in the next steps you reset the position of those Start curves so they look more natural at the beginning of the simulation.

**To reset the Start position**

- 1 Display the Start curves and the Current Position. (Hair > Display > Current and Start)
- 2 Rewind to the start frame and play the simulation.
- 3 Stop the simulation as soon as the curves relax below the constraint, but before the collision occurs.
- 4 Select the bound Start curves and then select Hair > Set Start Position > From Current.

You may have to repeat steps 2 to 4 a few times until you are happy with the look of the bound curtains at the beginning of the simulation.



2 | Hair

Lesson 2 > Rendering the curtain scene

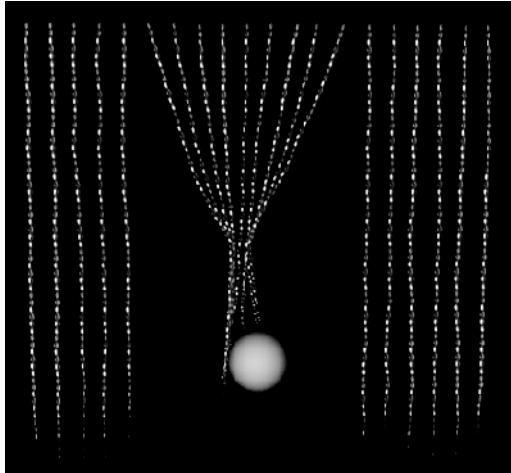
Rendering the curtain scene

In these next steps you render the curtain, which is Paint Effects hair, in the Maya Software renderer.

To render the Paint Effects hair using the Maya Software renderer

- 1 Modify the Render Globals settings for Paint Effects.
 - Click the Render Globals  button on the Status Line. The Render Global Settings window appears.
 - Select Maya Software from the Render Using drop-down list.
 - In the Maya Software tab, go to the Paint Effects Rendering Options section and turn on Oversample and Oversample Post Filter. These help achieve a better look for the Paint Effects rendering.
 - Click Close to close the Render Global Settings window.
- 2 Before rendering be sure to display the hair system's Current Position. (Hair > Display > Current Position)
- 3 Play the simulation and stop at the frame you want to render.
- 4 Click the Render Current Frame  button on the Status Line.

The curtain scene is rendered and should look something like the image below.



Beyond the lesson

In this lesson you learned how to:

- Set up objects to collide with hair.
You can set up hair to collide with itself, the ground plane or other objects. You can even use the Collision Sphere and Collision Cube constraints to establish implicit collisions. For more information, see *Making Hair Collide* in the online Hair and Dynamic Curves guide.
- Assign a Paint Effects brush to a hair system.
This is useful for creating a variety of effects, such as vines or ivy for hair. The hair output can be Paint Effects or NURBS curves. For more information, see *Assigning a Paint Effects brush to Hair* in the online Hair and Dynamic Curves guide.
- Create constraints on hair curves.
Use hair constraints to hold hair in position for a hairstyle, such as a ponytail or a clasp. You can even use hair constraints for more realistic hair behavior, such as using the

2 | Hair

Lesson 2 > Beyond the lesson

Transform constraint to hold the hair to the head. For more information, see *Setting up hair constraints* in the online Hair and Dynamic Curves guide.

3

Fluid Effects

Fluids are substances that change shape continuously or flow in response to forces.

Using Fluid Effects, you can realistically simulate 2D and 3D atmospheric effects (such as clouds and mist), combustion effects (such as smoke, explosions, and burning flames), and viscous fluid effects (such as molten lava).



You can also use the Fluid Effects ocean shader to simulate open water.

In Maya, there are three basic types of fluid effects:

- Dynamic fluid effects – Behave according to the natural laws of fluid dynamics and simulate the effects by solving fluid dynamics equations for each time step.
- Non-dynamic fluid effects – Use textures and animation to simulate fluid and fluid motion. These types of effects do not use fluid dynamics equations.

Dynamic and Non-Dynamic effects render as 3D volumes, so they interact appropriately with objects that move through them.

- Open water fluid effects – Use shaders to create realistic wave motion on large bodies of water.

3 | Fluid Effects

> Preparing for the lessons

This chapter includes the following lessons, which introduce you to some basic Fluid Effects concepts:

- “Lesson 1: Creating a dynamic 2D fluid effect” on page 65
- “Lesson 2: Creating a non-dynamic 3D fluid effect” on page 77
- “Lesson 3: Creating a dynamic 3D effect” on page 89
- “Lesson 4: Creating an ocean effect” on page 101

Preparing for the lessons

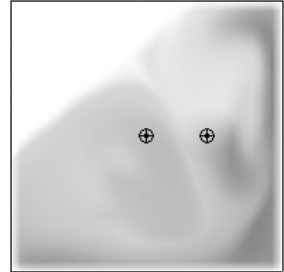
To ensure the lessons work as described, do these steps before beginning:

- 1 Create a new scene.
- 2 Select the Dynamics menu set.
Unless otherwise noted, the directions in this chapter for making menu selections assume you’ve already selected the Dynamics menu set.
- 3 Switch to shaded display mode for a hardware render view of the fluid simulations (Shading > Smooth Shade All on the view menu bar).
- 4 In the Time and Range slider, set the start frame to 1 and end frame to 200.
- 5 Make sure you understand the basic usage of the animation playback controls. See the chapter entitled “Animation” in Getting Started with Maya or refer to the Maya Help.

Lesson 1 **Creating a dynamic 2D fluid effect**

Fluid dynamics is a branch of physics that uses mathematical equations to describe how things flow.

In Fluid Effects, *dynamic* fluid effects simulate fluid motion by solving fluid dynamics equations at each time step. You can texture dynamic fluids, have them collide with and move geometry, affect soft body geometry, and interact with particles.



Non-dynamic fluid effects do not use the fluid dynamics equations to simulate fluid behavior. You create the look of a non-dynamic fluid using textures, and you create fluid motion by animating (keyframing) texture attributes. Because Maya doesn't solve the equations, rendering this type of fluid is much quicker than rendering a dynamic fluid.

In this lesson, you create a simple 2-dimensional dynamic fluid. You learn how to:

- Create a fluid using fluid emitters.
- Modify the dynamic behavior of the fluid by applying forces to it and changing the effect the boundary has on the fluid.
- Change the color of the fluid.
- Collide the fluid into an object.

In the next lesson, you'll create a non-dynamic fluid effect.

Creating a 2-dimensional fluid container

You begin the lesson by creating a 2-dimensional *fluid container*. The fluid container is the basis for dynamic and non-dynamic fluid effects. As its name implies, it contains the fluid effect. Fluids cannot exist outside a fluid container.

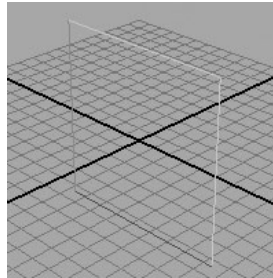
3 | Fluid Effects

Lesson 1 > Adding a fluid emitter to a container

To create a 2-dimensional fluid container

- Select Fluid Effects > Create 2D Container.

Maya creates an empty 2-dimensional fluid container centered at the origin in the XY plane.



Next, you add fluid to the container. To do this, you add values to the container for specific properties of the fluid: Density, Color, Velocity, Fuel, and Temperature. You can add any or all of these properties to create a fluid effect. One way to put properties into a container is to create them with a fluid *emitter*.

Adding a fluid emitter to a container

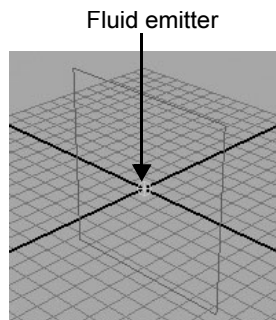
Fluid emitters create fluid property values and emit them into containers as the simulation plays. The emitter controls the position, rate of emission, and turbulence forces applied to the fluid properties when they are created. After fluid property values are created, the attributes of the fluid container control their appearance and behavior.

You can add multiple fluid emitters to containers. You move fluid emitters the same way you move any geometry (for example, by setting keys).

To add a fluid emitter to the container

- 1 With the container selected, choose Fluid Effects > Add/Edit Contents > Emitter.

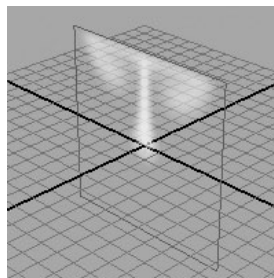
Maya creates a fluid emitter called fluidEmitter1 and places it at the center of the fluid container.



Tip You can create a fluid container with a fluid emitter in one step using Fluid Effects > Create 2D Container with Emitter.

- 2 Play the simulation using the playback controls at the bottom of the Maya window.

The fluid emitter creates Density values and emits them into the container.

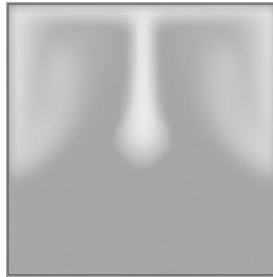


3 | Fluid Effects

Lesson 1 > Changing the behavior of a fluid

The white in the container represents *Density*. Density is the substance of the fluid and the visible fluid property. Where the Density values are highest, the Density appears more opaque.

Notice that the Density rises to the top of the container and moves along the boundary of the container, eventually flowing down the sides. Forces built into the container are acting on the Density values, causing them to change. The fluid collides with the boundaries rather than penetrating them.



- 3 Stop the playback and go to the start of the playback range.

Changing the behavior of a fluid

By modifying the attributes of the fluid container you affect the appearance and behavior of the fluid. Note that no matter what changes you make to the attributes, the fluid can never leave the container.

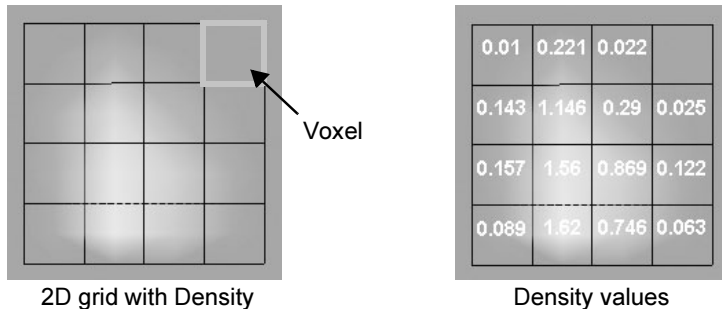
In the next steps you'll look at some of the attributes of the fluid container to gain some understanding into dynamic fluid effects.

To modify attributes for the fluid

- 1 Select the fluid container.
- 2 Show the Attribute Editor in the right panel of the Maya window, and click the *fluidShape* tab.
- 3 Look in the Contents Method section. The Contents Method defines how a fluid property is defined in the container, if at all.

Notice that Density and Velocity are set to *Dynamic Grid*.

A Dynamic Grid divides the container into rectangular units called *voxels* (volume pixels). You place the values in the grid by emitting them, painting them, or adding a predefined initial state. At each step of the simulation, Maya recalculates the values in each voxel using the fluid dynamics solver. This is what creates the dynamic motion of the fluid.



Tip You can display Density values in the grid by changing Numeric Display to Density in the Display section of the fluid Attribute Editor.

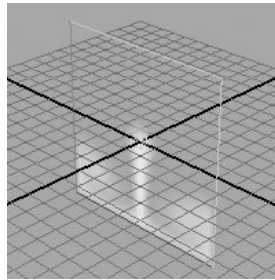
For dynamic fluid simulations, Density must be set to Dynamic Grid. Also, Velocity cannot be turned Off because velocity moves the property values around inside the grid.

3 | Fluid Effects

Lesson 1 > Changing the behavior of a fluid

You look at defining properties as Gradient and Static Grid in the next lessons.

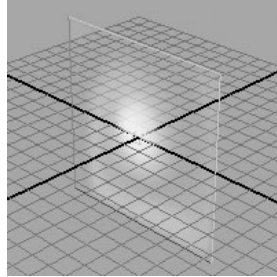
- 4 Make the density fall instead of rise:
 - In the Attribute Editor, open the Dynamic Simulation section.
 - Change the Gravity value to -9.8. (Acceleration due to gravity is -9.8 m/s^2 in the physical world.)
 - Negative values move the Density down.
 - Play the simulation using the playback controls at the bottom of the Maya window.



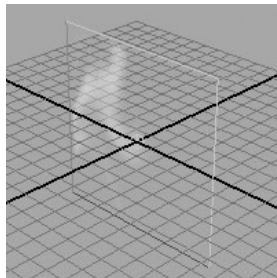
- 5 Add turbulence to the container to give the Density a whirling circular motion.
 - Select the fluid container if it is not already.
 - In the Attribute Editor, open the Contents Details section, and then open the Turbulence section.
 - Change the Strength value to 1.0.
 - Play the simulation using the playback controls at the bottom of the Maya window.

3 | Fluid Effects

Lesson 1 > Changing the behavior of a fluid



- 6 Stop the playback and go to the start of the playback range.
- 7 Change the effect of the container boundaries on the fluid:
 - In the Container Properties section at the top of the Attribute Editor, set the following options:
 - Boundary X: None
 - Boundary Y: None
- 8 Play the simulation using the playback controls at the bottom of the Maya window.



Notice that the fluid no longer collides with the sides of the container—it appears to pass through the sides. The fluid behaves as though the container boundaries don't exist. Note that even though the fluid appears to pass through the container boundaries, it does not exist outside the boundary. Fluids can only exist inside containers.

3 | Fluid Effects

Lesson 1 > Combining colors in a fluid

- 9 Change Boundary X and Boundary Y back to Both Sides to restore the boundaries at all sides of the container.
- 10 Stop the playback and go to the start of the playback range.

Combining colors in a fluid

In the next steps, you'll add another emitter to the container and make each emitter create different colored Density. You'll see how different colored Density interacts in the container.

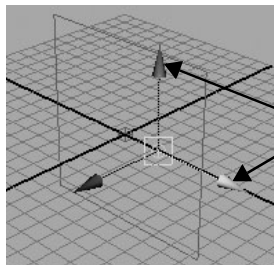
To combine colored fluids in the container

- 1 Select the fluid container and choose Fluid Effects > Add/Edit Contents > Emitter.

Maya creates a second fluid emitter called `fluidEmitter2` and places it at the center of the fluid container, in the same position as the first emitter.

- 2 With `fluidEmitter2` still selected, select the Move Tool (Hotkey: `w`) and drag the emitter to the right of the container, so the emitters no longer overlap.

Make sure you keep the emitter inside the container and on the plane by dragging only the X and Y manipulators. If you move the emitter outside the container, it will not emit.



Drag only the X and Y manipulators to keep the emitter in the container.

- 3 Open the Attribute Editor in the right panel of the Maya window, and click the `fluidEmitter` tab.
- 4 In the Fluid Attributes section, turn on Emit Fluid Color.

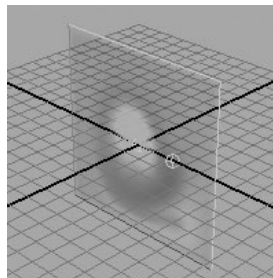
A message appears prompting you to set the fluid's Color Method to Dynamic Grid.

By setting the Color Method to Dynamic Grid, the color values you place in the container grid are recalculated by the fluid dynamics solver and changed during simulation. This means colors can interact during the simulation.

- 5 Click Set to Dynamic.
- 6 Click the Fluid Color color box in the Attribute Editor.
The Color Chooser opens.
- 7 Select *blue* from the Color Chooser and click Accept.
- 8 In the scene view, select fluidEmitter1, the emitter in the center of the container.
- 9 In the Attribute Editor, turn on Emit Fluid Color for fluidEmitter1.

You are not prompted to set the contents method to Dynamic Grid, because you already set the container to Dynamic grid when you modified fluidEmitter2.

- 10 Click the Fluid Color color box.
The Color Chooser opens.
- 11 Select *green* from the Color Chooser and click Accept.
- 12 Play the simulation using the playback controls at the bottom of the Maya window.



3 | Fluid Effects


Lesson 1 > Colliding a fluid with an object

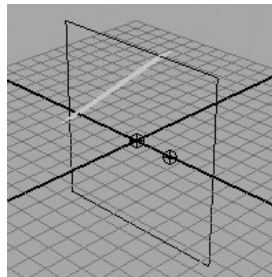
13 Stop the playback and go to the start of the playback range.

Colliding a fluid with an object

The 2D fluid you created is dynamic, so it can interact with objects. In the next steps, you'll create a plane inside the fluid container and make the fluid collide with it, rather than pass through it.

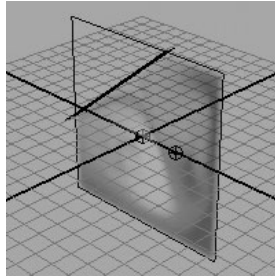
To make the fluid collide with a plane

- 1 Create a polygon plane:
 - Select Create > Polygon Primitives > Plane > . The Polygon Plane Options window opens.
 - In the window, select Edit > Reset Settings to reset the plane settings to the default ones, and then click Create.
- 2 In the Attribute Editor, click the pPlane1 tab and scale, translate, and modify the plane as follows:
 - Translate: -2 2 0
 - Rotate Axis: 0 0 40
 - Scale: 10 1 1



- 3 Select both the plane and the fluid container.
- 4 Select Fluid Effects > Make Collide. This makes the plane a collision object.

- 5 Play the simulation using the playback controls at the bottom of the Maya window.



The fluid collides with the plane as it moves through the container. It does not pass through the plane.

Beyond the lesson

This lesson introduced you to some basic concepts of dynamic fluids. You learned how to:

- Create an empty fluid container.
- Emit fluid properties into the container to create a dynamic fluid effect.
- Modify forces acting on the fluid container to change the behavior of the contents of the container (Gravity and Turbulence).
- Add color to a fluid by emitting color into the container.
- Make a fluid collide with an object.

The hardware render of the simulation provided you with a fast, realistic representation of the fluid. The final step would be to do a software render.

Although we looked at a 2D fluid, the same concepts apply to 3D fluids. In the lessons that follow, you'll work with 3D fluids and explore methods other than fluid emitters for adding contents to a

3 | Fluid Effects

Lesson 1 > Beyond the lesson

fluid container. You'll look at the texturing capabilities that are part of the built-in shader. You'll also learn about the Temperature and Fuel fluid properties and how you can use them in your effects.

Beyond this, you can also:

- Emit fluid from the surface of an object.
- Give the fluid a hard surface (surface render) for a globby or lava-like effect, rather than the soft, cloud-like surface you saw in this lesson.
- Cache the fluid simulation for faster playback.
- Deform an object with the force of a fluid.

For details on these topics, see the Fluid Effects information in the Maya Help.

Lesson 2 **Creating a non-dynamic 3D fluid effect**

In the first lesson, you created a simple 2D dynamic fluid. 3D fluids intrinsically require extra data to define them, which can make them very complex. This extra data can slow a dynamic simulation exponentially because more calculations (solving) must be performed at every step of the simulation. For a less memory intensive effect, you could use a 2D fluid (with less data), or you could create a 3D non-dynamic effect.



In non-dynamic fluid effects, the fluid property values are predefined within Maya and stay constant over time, which means they don't have to be recalculated. You create the appearance of the fluid by texturing a special fluid shader that is assigned to the fluid. This shader is built into the fluid for better performance. If you want the fluid effect to have motion, you can animate (keyframe) the texture attributes. Because Maya doesn't solve the fluid dynamics equations, rendering this type of fluid is much quicker than rendering a dynamic fluid.

In this lesson, you learn the fundamentals of creating non-dynamic fluids by creating a cloud bank using a 3D non-dynamic fluid. You learn how to:

- Create a 3D fluid using predefined gradient contents.
- Texture the contents of the fluid using the built-in fluid shader.

3 | Fluid Effects

Lesson 2 > Creating a 3D fluid container


Before beginning the lesson, do the steps in "Preparing for the lessons" on page 64.

Creating a 3D fluid container

Creating a fluid container is the first step in creating a non-dynamic fluid effect.

Fluid containers do not need to be uniform in X, Y, and Z. In fact, because larger fluids take longer to render, you should make containers only as large in each direction as necessary for the effect. A cloud bank sits above the ground like a blanket, so the container can be rectangular rather than cubic.

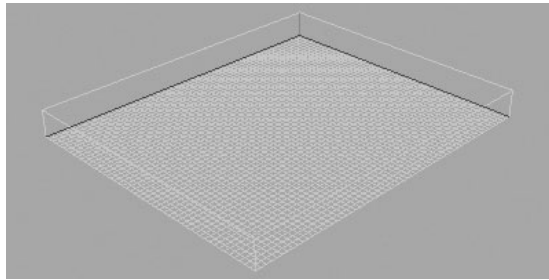
To create a 3D fluid container

- 1 Select Fluid Effects > Create 3D Container > .
The Create 3D Container Options window opens.
- 2 Set the container options as follows, then click Apply and Close:
 - X Resolution: 50
 - Y Resolution: 5
 - Z Resolution: 60
 - X Size: 50.0
 - Y Size: 5.0
 - Z Size: 60.0

The resolution is defined in voxels. Higher resolutions produce finer detail, but increase rendering time. The size of the container is defined in the working units set for Maya. The size of the fluid determines the size of each voxel.

Tip It's good practice to make the container's Resolution and Size proportional to ensure that the voxels are square making the quality the same along each axis.

Maya creates an empty 3-dimensional fluid container centered at the origin.



Next, you add fluid to the container.

Adding fluid to a container

To make this a non-dynamic fluid, you add fluid property values to the container that will stay constant over time. There are two ways to do this: add these values to a static grid, or select a set of values from a list of predefined gradients that come with Maya. For this lesson, you'll use the latter method.

For a cloud bank, Density is the only property you need to define—the other properties are typically used in dynamic solving.

To add fluid to the container

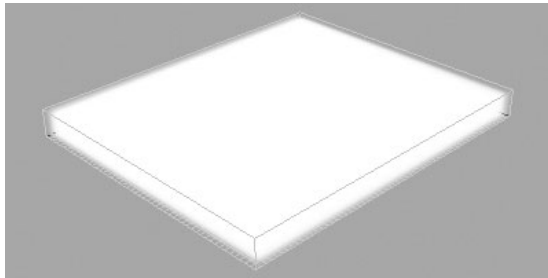
- 1 Dolly the scene view to see the entire container
- 2 With the fluid container selected, show the Attribute Editor in the right panel of Maya, and click on the fluidShape1 tab.

3 | Fluid Effects

Lesson 2 > Defining shader attributes for a fluid

- 3 In the Contents Method section of the Attribute Editor, set:
 - Density: Gradient
 - Density Gradient: Constant

Set Velocity, Temperature, and Fuel properties to Off. (They're not used in this effect.)



By setting the Density Gradient to Constant, you made all the Density values in the container the same—a value of 1. You can scale these values, but otherwise, they cannot change.

Defining shader attributes for a fluid

The fluidShape has a built-in shader that you can use to modify color, texture, and other attributes to give the fluid a particular look. In the next steps, you'll make the shader apply specifically to the Density values inside the container.

To define shader attributes for the fluid

- 1 Open the Shading section in the Attribute Editor.
- 2 In the Color subsection, change Color Input to Density.
The Density values in the container will now take on the colors defined on the color bar. Leave the color white to make the clouds appear white.
- 3 In the Opacity subsection, check that the Opacity Input is set to Density.

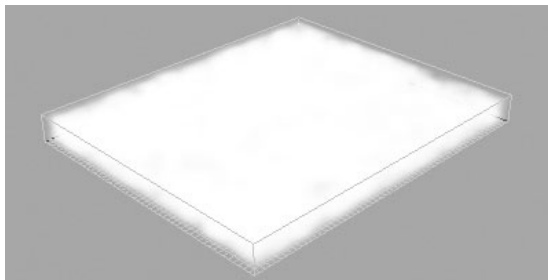
The opacity represents how much the Density will block light. You'll look more at Opacity later in this lesson.

Texturing the contents of a fluid container

So far the container is filled with solid white Density. To create the cloud effect, you texture the Density so that some areas are transparent and some areas are opaque.

To texture the density of a fluid

- 1 Turn on hardware texturing display so you can see the effect of the textures on the fluid without rendering by selecting **Shading > Hardware Texturing** from the scene view menu.
- 2 Open the Textures section in the Attribute Editor.
- 3 Turn on **Texture Opacity** to apply the current texture to Opacity values. The current texture is **Perlin Noise**, defined by **Texture Type**.

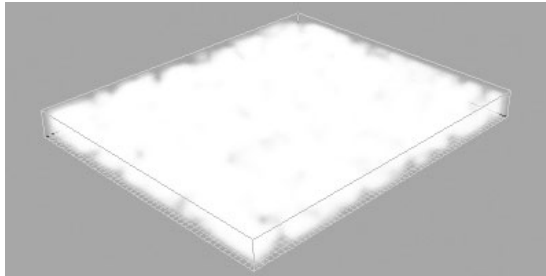


Notice that the Density now has a slightly blotchy look to it, with areas that are more opaque and areas that are more transparent. This texture provides the standard 3D noise used in the 3D Solid Fractal texture included with Maya.

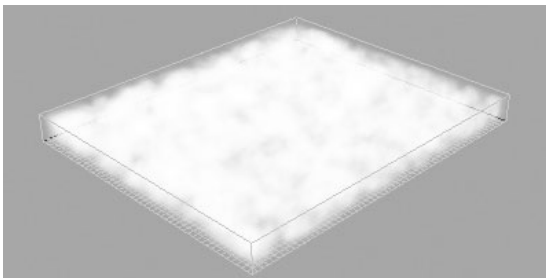
- 4 Change **Texture Type** to **Billow** for a fluffy, cloud-like effect. The **Billow** texture is computationally intensive and therefore slower than the other texture types.

3 | Fluid Effects

Lesson 2 > Texturing the contents of a fluid container



- 5 Change the look of the texture by setting the following texture attributes:
 - Amplitude: 0.5
 - Depth Max: 4



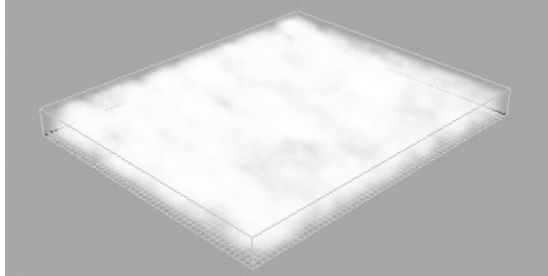
Decreasing the Amplitude makes the areas with low Density more transparent and the areas with high Density more opaque.

Increasing Depth Max adds detail. Increasing it will also increase render time.

- 6 Stretch the texture in the X direction by changing the X, Y, and Z components of the Texture Scale to 2, 1, 1.

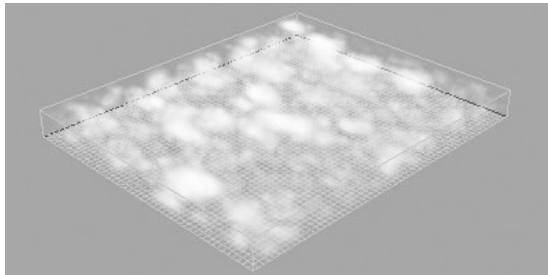
3 | Fluid Effects

Lesson 2 > Texturing the contents of a fluid container



7 Change the following Billow texture attributes to make the “billows” less dense, more spotty, and with randomly different sizes.

- Billow Density: 0.6
- Spottyness: 2.0
- Size Rand: 0.40

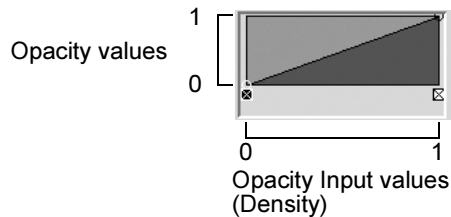


8 Modify the Opacity so that areas in the container that are very dense appear less opaque, areas that have very little Density become totally transparent, and the transition between areas that are totally transparent and areas that are more opaque is less gradual.

3 | Fluid Effects

Lesson 2 > Texturing the contents of a fluid container

In the Attribute Editor, go to the Shading section. Look at the Opacity graph in the Opacity subsection. This graph represents the relationship between Opacity values and Density values (the Opacity Input).

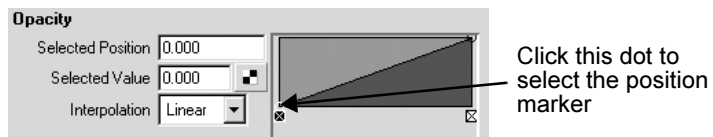


Opacity values range from 0 on the bottom (totally transparent, no opacity) to 1 on the top (totally opaque).

Density values range from 0 on the left side (no Density) to 1 on the right side (high Density).

So for the linear graph shown above, where Density values are 0, Opacity values are 0, making the Density totally transparent, where Density values are 0.5, Opacity values are 0.5, making the Density partially opaque, and where Density values are 1, Opacity values are 1, making the Density totally opaque.

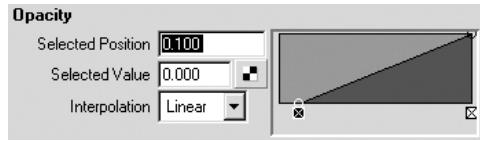
- Click the first dot on the Opacity graph to select the position marker. Position markers mark the location on the graph from left to right (the Opacity Input value). The outline of the dot is white when a position is selected.



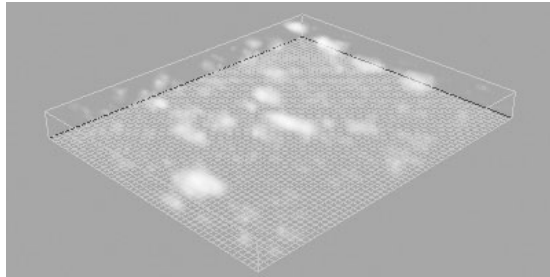
- Change Selected Position to 0.10 to change the position of the marker.

3 | Fluid Effects

Lesson 2 > Texturing the contents of a fluid container



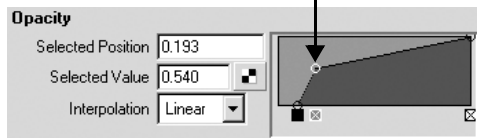
The position marker moves to the right. Now, for Density values between 0 and 0.10, the Opacity values will be 0. This means that Density that was previously partially transparent will be completely transparent.



The more transparent areas of cloud disappeared, but now the solid areas of cloud are less opaque.

- Click on the graph to create a new position marker.

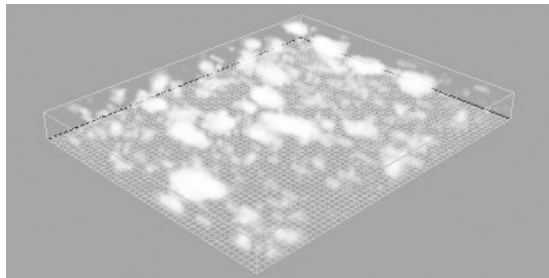
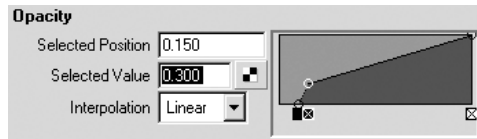
Click on the graph to create a new position marker



- Change the marker position and value as follows:
 - Selected Position: 0.15
 - Selected Value: 0.30

3 | Fluid Effects

Lesson 2 > Adding self shadowing to texture density



Density values that are greater than 0.15 are now more opaque, and the transition between areas of total transparency (Opacity 0), and areas where the Density becomes more visible (Density 0.15) is less gradual.

Adding self shadowing to texture density

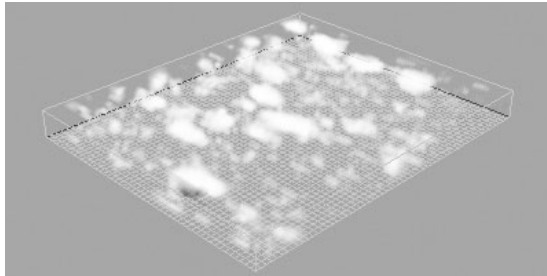
Give the clouds some depth by adding self shadowing. Self shadowing causes the fluid to cast shadows on itself using a single internal directional light at -1, -1, -1.

To add self shadowing to the texture density

- 1 In the Attribute Editor, under the fluidShape1 tab, open the Lighting section.
- 2 Turn on Self Shadow.

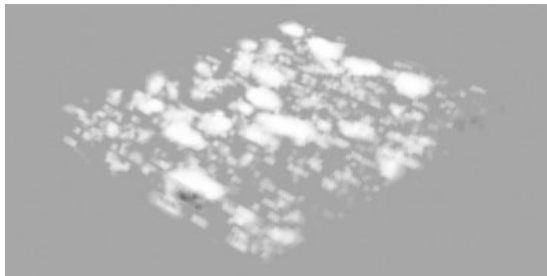
3 | Fluid Effects

Lesson 2 > Fluids Lesson 2: Beyond the lesson



The clouds now have some darker areas on them, giving them some depth.

- 3 In the Display section, change Boundary Draw to None to hide the container. This gives you a better idea of how the fluid will look before you render it.



Fluids Lesson 2: Beyond the lesson

This lesson showed you a basic technique for creating non-dynamic fluids. While creating the effect, you learned how to:

- Create a 3D fluid container of a specific resolution and size.
- Add predefined gradient contents to the container to create a non-dynamic fluid.

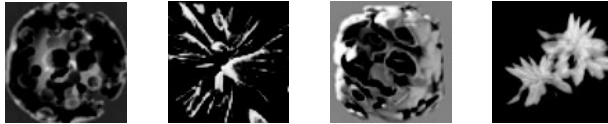
3 | Fluid Effects

Lesson 2 > Fluids Lesson 2: Beyond the lesson

- Texture the contents of the fluid using the built-in fluid texturing capabilities of the fluidShape. (You can also texture dynamic fluids using the same techniques.)

To make the texture move, you would keyframe the Texture Time attribute in the Textures section of the Attribute Editor. For details on keyframing attributes, see the Maya Help.

You modified several texture attributes in this lesson, but there are many more attributes for customizing textures. Also, note that the non-dynamic effects you can create using texturing are not limited to clouds. Here are some examples:



To learn more about how to use texturing and other fluid attributes, study the Fluid Effects examples that are included with Maya (Fluid Effects > Get Fluid Example) and look at the Notes section at the bottom of the Attribute Editor for example fluids. It includes information about the selected effect, pointing out the key aspects of its construction. In addition to these examples, a growing collection of examples are available from the Alias community web site. Visit this site regularly for new examples. For further information on Fluid Effects, see the Maya Help.

Lesson 3 **Creating a dynamic 3D effect**

In the real world, gasoline reacts chemically with oxygen at a certain temperature, creating flames (light, heat) and the gaseous products of combustion. Over time, the gasoline burns until it's totally consumed. You can simulate this situation in Fluid Effects using Fuel, Density, and Temperature properties.



In this lesson, you create a dynamic explosion effect by filling the bottom of a container with Fuel and Density, then “igniting” it with Temperature. You learn how to:

- Add property values to a container by painting them.
- Paint in a 3D container.
- Use Fuel, Density, and Temperature to create an explosion effect.

Before beginning the lesson, do the steps in “Preparing for the lessons” on page 64.

Creating a 3D container

- 1 Select Fluid Effects > Create 3D Container > .

The Create 3D Container Options window opens.

- 2 In the window, select Edit > Reset Settings to restore the Resolution and Size settings to the defaults 10 10 10 and 10 10 10, respectively.

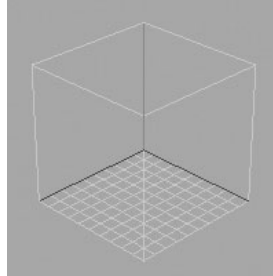
The default container size and resolution are adequate to illustrate the concepts in this lesson.

- 3 Click Apply and Close.

Maya creates an empty 3-dimensional fluid container centered at the origin.

3 | Fluid Effects

Lesson 3 > Painting Fuel and Density into a container



Next, add fluid to the container. In the first lesson, you added fluid to the container by emitting it into a Dynamic Grid. In the second lesson, you added fluid to the container by selecting a predefined gradient. In this lesson, you will add fluid to the container by painting property values inside it.

Painting Fuel and Density into a container

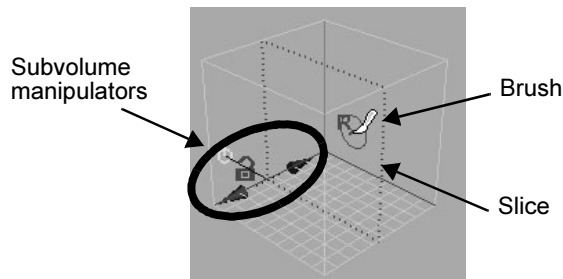
In Fluid Effects, Fuel defines the state of a reaction (unreacted, completely reacted, and states in between). Density represents the substance that is reacting, while Temperature causes the reaction. Combining Density with Fuel defines a situation in which you can see a reaction. You add Temperature to this situation to start the reaction. As the reaction takes place, Density values become smaller and the Fuel values get smaller to represent how much of the reaction has yet to take place. The reaction also creates more Temperature, and it creates light.

In the following steps you'll paint Fuel and Density values in the container. To paint in a 3D container you actually paint in two dimensions on "slices" of the container. A slice is a plane in X, Y, or Z that represents where you paint. (You could think of a slice as a 2D canvas.) You paint each slice individually, but the accumulative effect of adjacent slices is a 3D fluid.

To paint Fuel and Density into a container

- 1 With the container still selected, choose Fluid Effects > Add/Edit Contents > Paint Fluids Tool > .

The Tool Settings window opens and a slice displays at the origin of the fluid container. The slice is represented by a plane with dotted edges and fluid subvolume manipulators at one corner. When you move the pointer over the slice the pointer changes to a brush indicating that you can paint.



- 2 At the top of the Tool Settings window, click Reset Tool to set the Paint Fluids Tool settings to the default values.
- 3 Select the fluid properties you want to paint. In the Paint Attributes section of the Tool Settings window, beside Paintable Attributes, select Density and Fuel. You will paint both these properties at the same time.

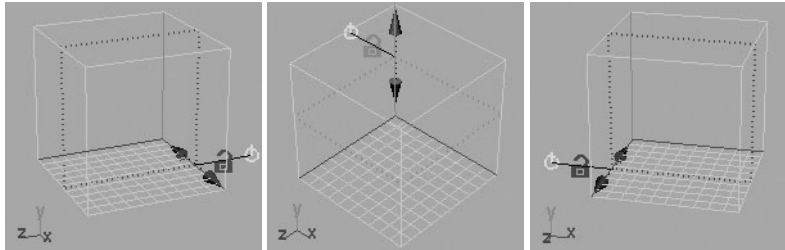
You could paint each property separately, but for this example, the values you paint will be the same for both. It is more efficient to paint them both at the same time.

A message appears prompting you to set the fluid's Fuel method to Dynamic Grid. By setting the Fuel method to Dynamic Grid, the fuel values you place in the container grid (in this case, by painting them) are recalculated by the fluid dynamics solver and changed during simulation. The Density method is defined as Dynamic Grid by default.

3 | Fluid Effects

Lesson 3 > Painting Fuel and Density into a container

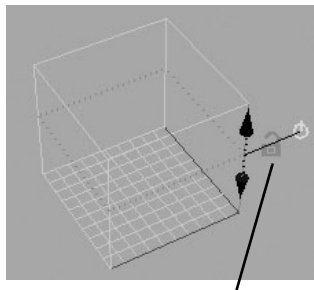
- 4 Click Set to Dynamic.
- 5 Tumble the scene and look at the slice and manipulators.



The orientation of the slice changes relative to your view. The slice is perpendicular to the axis with the manipulators. The color of the manipulators corresponds with the color of the axis in the View or Origin axes.

Painting properties relative to different axes is good practice to ensure there are no gaps between slices when the fluid is viewed from different angles.

- 6 Tumble so that the slice is perpendicular to the Y axis and then click the open lock icon to lock the slice axis.



Click to lock the axis...



The lock closes. Now when you tumble, the slice remains perpendicular to Y. It does not switch to the other axes. This allows you to change the view in any way and still be able to paint on the same slice.

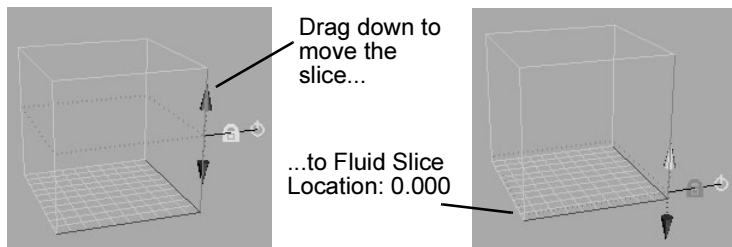
3 | Fluid Effects

Lesson 3 > Painting Fuel and Density into a container

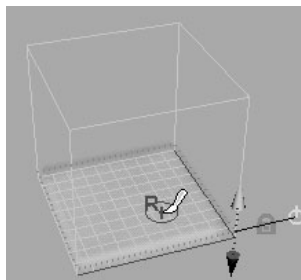
- 7 Drag the top move arrow down along the axis to the first slice at the bottom of the container. The number of slices along each axis corresponds with the resolution of the fluid container. The numbering starts at 0 at the intersection of the axes. For this container the resolution is 10 10 10, so the slices are numbered from 0 to 9.

The help line displays the location of the selected slice.

Fluid Slice Location: 0.000



- 8 In the Paint Attributes section of the Tool Settings window, ensure that Value is set to 1. A Fuel value of 1 represents a totally unreacted state while a Fuel value of 0 represents a completely reacted state. A Density value of 1 is totally opaque while a Density value of 0 is completely transparent.
- 9 Drag the brush on the slice to paint values. Paint until the entire slice is filled with values.



3 | Fluid Effects

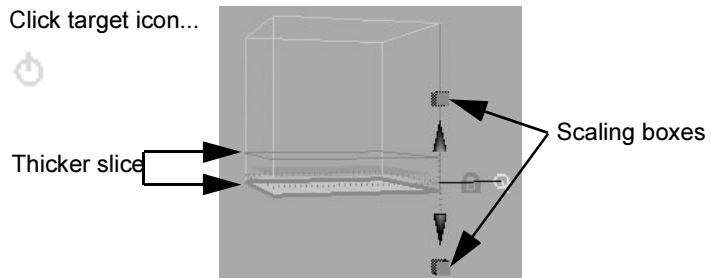
Lesson 3 > Painting Fuel and Density into a container

The values display as yellow because you are painting two properties at the same time. Density values are represented by the opacity of the shaded values, while Fuel values are represented by color. The range of Fuel values corresponds with a ramp of colors from blue (values of 0) to yellow (values of 1).

To make the fluid deeper, you could move the slice to Fluid Slice Position 1, paint, move the slice to Fluid Slice Position 2, paint, and so on. But a quicker way is to resize the slice to make it thicker and then flood the entire slice with values.

10 Resize the paint slice, making it thicker:

- Click the target icon once to switch to the subvolume scaling mode.



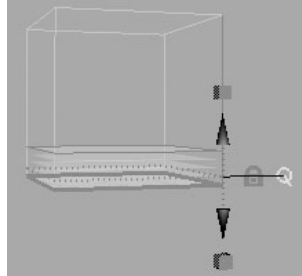
The slice becomes thicker and the manipulator changes to include scaling boxes at the ends of the move arrows.

- Click the top scaling box and read the help line to determine that the Fluid Slice Thickness is 3.00. You could drag the scaling boxes to change the thickness of the slice, but the slice is thick enough for the lesson.

11 In the Tool Settings window, click the Flood button.

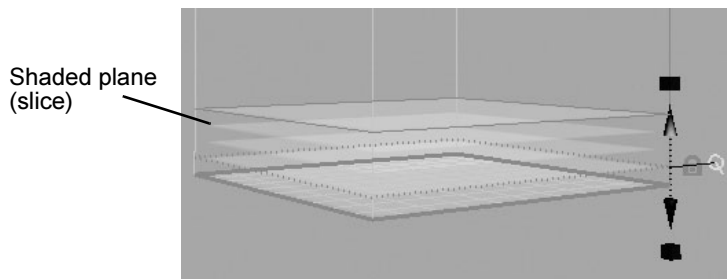
3 | Fluid Effects

Lesson 3 > Painting Fuel and Density into a container



Flooding filled each voxel in the subvolume with Fuel and Density values of 1.

Notice that the values display as shaded planes rather than as a solid mass. This is an interactive display feature for shaded display mode. Displaying more shaded planes (slices) produces more detail but reduces the speed of the screen draw. You can change this display feature (Slices Per Voxel) and others in the Display section of the Attribute Editor.



Now that the container has Density and Fuel in it, you are ready to start a reaction by adding Temperature. You could think of this as putting a lit match into a container of gasoline.

3 | Fluid Effects

Lesson 3 > Painting Temperature values into a container

Painting Temperature values into a container

By default, the Fuel and Density will react when there is no Temperature in the container, but adding higher temperature values will make the reaction more immediate.

To paint Temperature values into a container

- 1 In the Tool Settings window, beside Paintable Attributes, select Temperature. A message appears prompting you to set the fluid's Temperature method to Dynamic Grid or Static Grid.

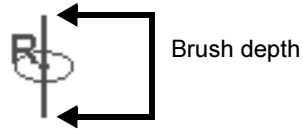
By setting the Temperature method to Dynamic Grid, the temperature values you place in the container grid are recalculated by the fluid dynamics solver and changed during simulation.

By setting the Temperature method to Static Grid, the temperature values you place in the container grid are used by the fluid dynamics solver during the simulation, but they cannot change during the simulation. Static Grid is useful when you want to define specific values in specific areas of the container, but you do not want these values to change over time.

- 2 Click Set to Dynamic. The Density and Fuel values disappear. The values are still there, but only Temperature values now display. Currently there are no temperature values in the container.
- 3 Change the brush so you can paint through the entire thickness of the slice with a single stroke. In the Stroke section of the Tool Settings window, change Stamp Depth (3D) to 2. The brush changes, showing how deep the brush stroke will paint.

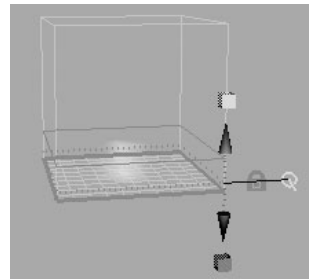
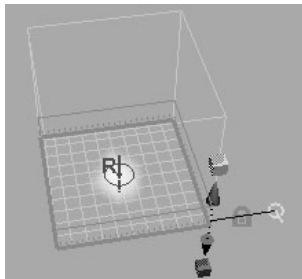
3 | Fluid Effects

Lesson 3 > Adding color to Density and Temperature



If you make the brush depth thicker than the depth of the slice, only the space within the slice will be painted.

- 4 Paint a small amount of Temperature in the center of the slice.



The stroke adds Temperature values through the entire thickness of the slice.

The next steps are to view all the property values together, and color the properties to differentiate them.

Adding color to Density and Temperature

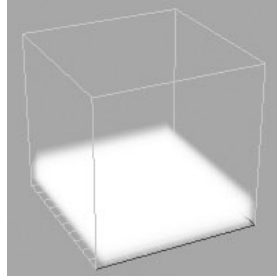
Use the built-in shader to modify the color of Density and Temperature in the container.

To add color to the Density and Temperature

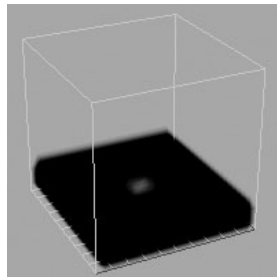
- 1 Choose the Select Tool (Hotkey: q) to leave the Paint Fluids Tool. The manipulators disappear and now you see the fluid as it will render

3 | Fluid Effects

Lesson 3 > Adding color to Density and Temperature



- 2 With the fluid container selected, open the Attribute Editor.
- 3 Change the color of the fluid as follows:
 - In the Shading section of the Attribute Editor, expand the Color section, and change Color Input to Density.
 - The Density values in the container will now take on the colors defined on the color bar.
 - Click the Selected Color box. The Color Chooser opens.
 - Select black in the Color Chooser and click Accept.

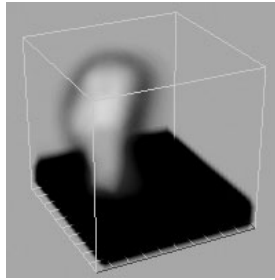


The Density turns black.

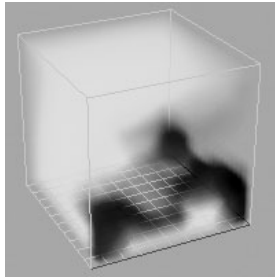
Look closely at the center of the fluid. The area where you painted Temperature has a reddish orange color. In the Incandescence section of the Attribute Editor, notice that Incandescence Input is set to Temperature. Temperature gives

off light and Incandescence controls the amount of light emitted from regions of Density due to self illumination. The ramp defines the color of this light.

- 4 Play the simulation using the playback controls at the bottom of the Maya window.



By frame 160, most of the Density values have reacted completely.



Beyond the lesson

This lesson introduced you to the Fuel and Temperature fluid properties. You learned how to:

- Combine Fuel, Density, and Temperature to create a dynamic explosion effect.
- Add property values to a container by painting them.

3 | Fluid Effects

Lesson 3 > Beyond the lesson

- Paint in a 3D container.

You can control how quickly the reaction takes place during the simulation, the temperature required to start the reaction, how much heat and light the reaction releases over time, and other aspects of the reaction using the attributes in the Fuel section under Contents Details in the fluidShape Attribute Editor.

For a more realistic look, you can also texture the fluid using the built-in texturing attributes in the Textures section.

For more information on Fuel and texturing attributes, see Fluid Effects in the Maya Help.

Lesson 4 **Creating an ocean effect**

An ocean effect is defined by a plane with an ocean shader assigned to it. Ocean effects are not created with fluid containers like the dynamic and non-dynamic fluid effects you created in the first three lessons.



A shader is a collection of attributes that define color and other surface characteristics of the plane it is assigned to. For an ocean shader, these attributes define realistic waves on large bodies of water.

In this lesson, you create an ocean, put a cube in it, and see how the cube floats in the ocean. You learn how to:

- Create an ocean plane and shader.
- Create a preview plane to see the effects of the shader, including displacement, without rendering.
- Modify the ocean shader attributes to make the waves bigger.
- Make an object a boat in the scene.
- Modify the attributes for an expression that controls the buoyancy of the boat in the ocean.

Before beginning the lesson, do the steps in "Preparing for the lessons" on page 64.

3 | Fluid Effects

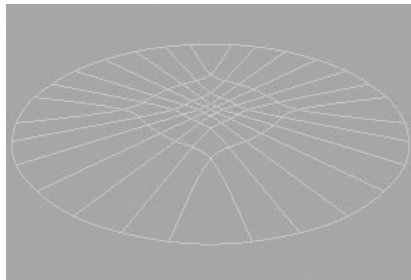
Lesson 4 > Creating an ocean plane and shader

Creating an ocean plane and shader

Fluid Effects simplifies the process of creating an ocean by providing a single command that creates a plane that is optimized for best results, and an ocean shader with appropriate connections to the plane.

To create an ocean plane with ocean shader

- 1 Select Fluid Effects > Ocean > Create Ocean. Maya creates an ocean plane and assigns an ocean shader to it.
- 2 Dolly out to see the plane.



The plane is a NURBS plane with more patches concentrated in the center. This is so you can see more detail in the center and less detail as the ocean extends to the horizon.

Because interactive shading would slow down playback, the shading is turned off for the surface.

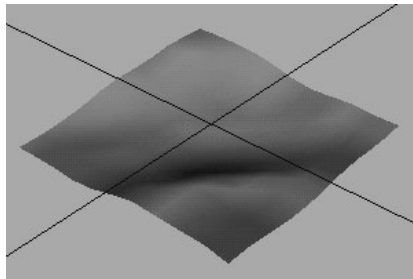
Adding a preview plane to an ocean

Because shading is turned off for the ocean plane, you need a preview plane to see displacement and shading in the hardware render. Displacement changes the surface of the ocean plane so you can see the different wave heights.

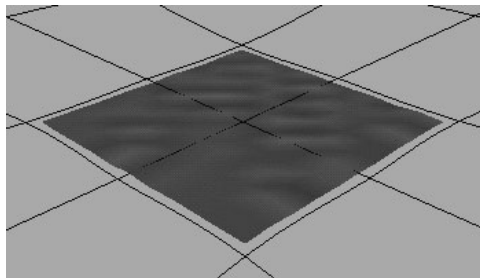
To add a preview plane to the ocean

- 1 Dolly back in to the center of the plane.
- 2 Select Fluid Effects > Ocean > Add Preview Plane.

Maya creates a preview plane that interactively shows the displacement of the ocean on a shaded patch of ocean. You can scale and translate it to preview different parts of the ocean. This plane does not appear in renders.



- 3 With the preview plane still selected, choose the Scale Tool (Hotkey: r) and make the patch bigger by dragging the center scale box to the right. Make it approximately 25 times bigger. (Look at the scaling numbers in the help line.)
- 4 Play the simulation using the playback controls at the bottom of the Maya window.



3 | Fluid Effects

Lesson 4 > Modifying ocean attributes

The preview plane ripples slightly showing the small wave displacement.


- 5 Light the scene to prepare it for rendering later in the lesson:
 - Select Create > Lights > Directional Light. Maya creates a directional light called `directionalLight1` at the center of the grid.
 - Open the Attribute Editor and in the Directional Light Attributes section of the `directionalLightShape1` tab, change Intensity to 2 to make the light brighter.
 - Click the `directionalLight1` tab and rotate the light as follows to direct the light rays onto the ocean.

Rotate: -100 42 20

Modifying ocean attributes

In the next steps, you will modify ocean shader settings to make the waves bigger.

To modify attributes for the ocean shader

- 1 Render the current frame to see what the ocean looks like before you modify the ocean shader attributes. On the Status line, click the render current frame icon. ).



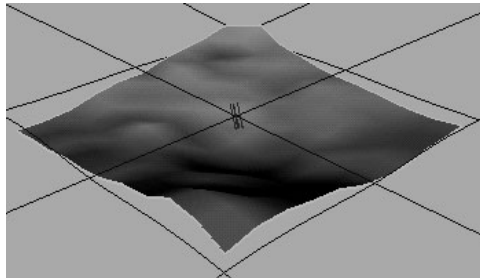
3 | Fluid Effects

Lesson 4 > Modifying ocean attributes

- 2 Select the preview plane or the ocean plane (it doesn't matter which one), and in the Attribute Editor, click the oceanShader1 tab.
- 3 Open the Ocean Attributes section and change the following values:
 - Num Frequencies: 12
 - Wave Length Max: 40

Num Frequencies controls the number of frequencies between minimum and maximum wave lengths. Increasing this value results in a more textured look on the water surface.

Wave Length Max controls the maximum length of waves in meters. Longer wave lengths make bigger waves.



- 4 Play the simulation to see the effect of your changes on the displacement of the ocean waves.
- 5 Stop the playback and go to the start of the playback range.
- 6 Render the current frame.

3 | Fluid Effects

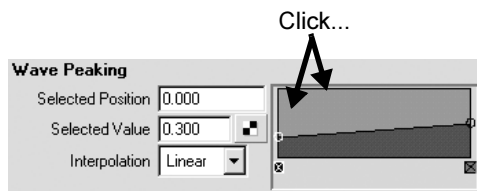
Lesson 4 > Modifying ocean attributes



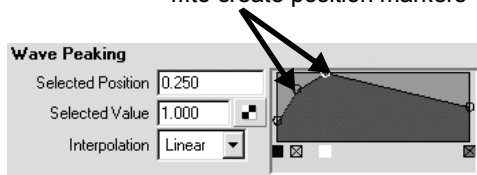
- 7 Make the tops of the waves form crests by adjusting the Wave Peaking graph in the Attribute Editor.

Wave Peaking controls the amount of crest formation for waves across the range of wave frequencies. It simulates a side-to-side sloshing of waves, as opposed to an up-down motion. Wave Peaking is only applied to turbulent waves (where Wave Turbulence is not zero).

Click on the graph to create the position markers shown in the next illustration.



...to create position markers



Where the wave frequency is lower, the waves peak more.

- 8 Render the current frame.



The tops of the waves now crest slightly.

Notice the white highlights on the ocean. These are specular highlights caused by the light hitting the water.

- 9 Make the highlights dimmer so they don't pop out as much, and bigger. Open the Specular Shading section and change the following settings:
- Specularity: 0.45
 - Eccentricity: 0.1

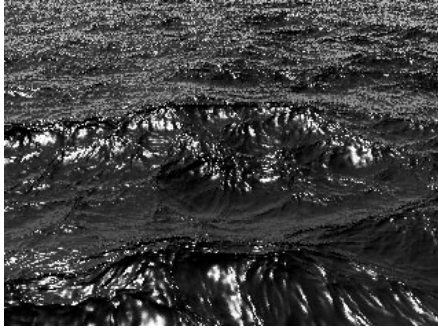
Specularity controls the brightness of the specular highlights. Higher values create brighter highlights.

Eccentricity controls the size of the specular highlights. Higher values create bigger highlights.

- 10 Render the current frame.

3 | Fluid Effects

Lesson 4 > Floating objects

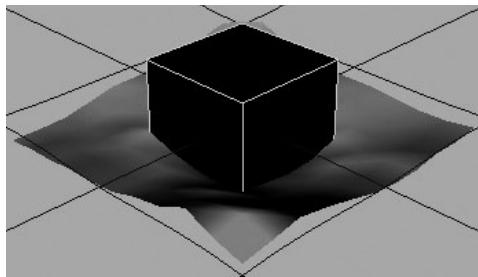


Floating objects

You can make geometry (for example, a boat) float in the ocean, moving appropriately with the motion of waves. The following steps show you how using a simple polygon cube.

To float a cube in the ocean

- 1 Create a polygon cube. (Select Create > Polygon Primitives > Cube.)
- 2 In the Attribute Editor, click the pCube1 tab and scale the cube as follows:
 - Scale: 10 10 10

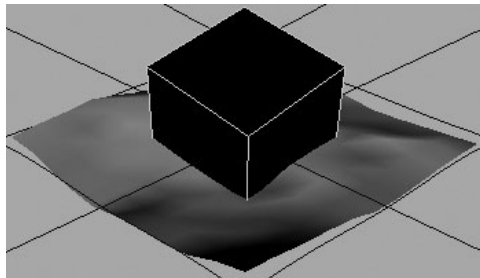


- 3 Select both the cube and the ocean plane (not the preview plane) at the same time and select Fluid Effects > Ocean > Make Boats.

Maya creates a boat locator at the center of the cube to mark the cube's position in space. The locator is connected to a predefined expression that simulates buoyancy effects. The expression is connected to the wave heights (displacement) of the ocean.

To learn more about expressions, look at the Expressions lessons in *Getting Started with Maya*.

- 4 Play the simulation using the playback controls at the bottom of the Maya window.



The cube bobs, pitches and rolls in the ocean.

- 5 In the Attribute Editor, click the locatorShape2 tab and in the Extra Attributes section, modify the following options. These serve as inputs to the expression.
 - Buoyancy: 0.75
 - Roll: 0.1
 - Pitch: 0.2

Increasing Buoyancy makes the box float more and sink into the ocean less. Decreasing the Roll and Pitch values causes the cube to roll less from side to side, and pitch less forwards and backwards as a result of the wave motion.

3 | Fluid Effects

Lesson 4 > Beyond the lesson

Beyond the lesson

In this lesson, you learned that ocean effects are different from the dynamic and non-dynamic volume fluid effects. An ocean effect is created with a plane and shader rather than a container and fluid properties. With ocean effects you learned how to:

- Create an ocean plane and shader.
- Create a preview plane to see the displacement of the shader without rendering.
- Modify ocean shader attributes to customize the look of the ocean.
- Make geometry bob and move in the ocean.

To learn more about ocean shader attributes, look at the ocean shader examples that are included with Maya (Fluid Effects > Get Ocean/Pond Example). Some of these examples use a 3D non-dynamic volume fluid to create the atmosphere above the ocean.

Motor Boats

You can create a game-style boat simulation using the Fluid Effects > Oceans > Make Motor Boats command. Once you set up hotkeys for the rudder and throttle of the boat, you can “drive” the boat in the ocean. The boat will roll, pitch, and jump over waves appropriately.

Boat Wakes

You can create boat wakes using the Fluid Effects > Oceans > Create Wakes command. Wake fluids do not use the Navier Stokes solver like regular fluids, but use the Spring Mesh Solver. A fluid emitter is used to drive the motion of the fluid.

Ponds

If a smaller body of water is what you had in mind, you can create a pond using Fluid Effects > Pond > Create Pond. Ponds are 2D fluids that use a spring mesh solver and a height field. You can generate bubbles, ripples, and waves using the Create Wake options for the pond.

For details on these topics and others, see Fluid Effects in the Maya Help.

3 | Fluid Effects

Lesson 4 > Beyond the lesson

4

Cloth

With Maya's Cloth feature, you can model garments and simulate natural cloth behavior on an animated 3D character. You can also apply dynamic effects such as wind or turbulence to the cloth. In addition to creating clothing for characters, you can create and animate sheets, bedding, drapery, flags, and other fabric goods.

This chapter includes these lessons:

- "Lesson 1: Making a Shirt" on page 118
- "Lesson 2: Making Pants" on page 134
- "Lesson 3: Exporting & Importing Garments" on page 144



Image courtesy of Sony Pictures Imageworks. Columbia/TriStar.

About cloth animation

To create and animate clothing, you do the following general steps:

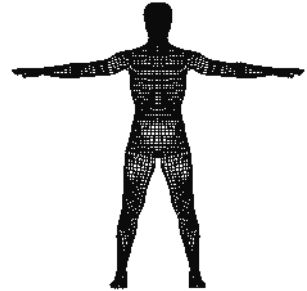
- Posing and animating a character
- Constructing a garment
- Fitting a garment onto a character
- Animating the cloth

4 | Cloth

> About cloth animation

Posing and animating a character

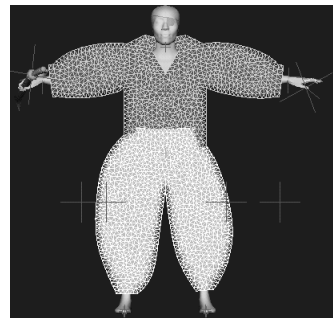
Before you construct a garment, you position the character in a *dress-up pose* at the scene's start frame. The dress-up pose minimizes wrinkling and other undesired effects when you fit the garment.



You animate the character to create the transition from the dress-up pose to its starting position, for instance, to the first step of a walk cycle. A typical duration for the transition is 20 frames—longer if the starting pose differs significantly from the dress-up pose. Finally, you complete the character's animation from the starting pose onward.

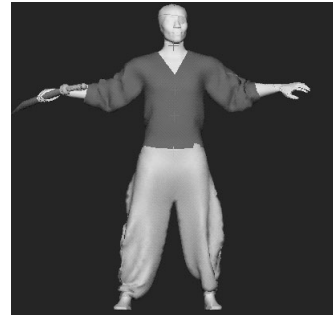
Constructing a garment

When you begin creating a garment, you use the surface of your 3D character as a mannequin, much like a tailor uses a mannequin. Using the dress-up pose as a guide, you draw curves to form the outline of a garment. From these curves you create *panels*. The panels are similar to actual fabric patterns that are sewn together to make a finished garment. When you seam the panels together, you create a *garment*.



Fitting a garment onto a character

To fit the garment, you make the character a *collision object*, then run a cloth simulation for a single frame—the dress-up pose frame. As the simulation runs, the garment drapes onto the character. Subtle adjustments to the garment’s position occur continuously as the simulation runs. You stop the simulation at a desirable position, then save the position as the initial position.



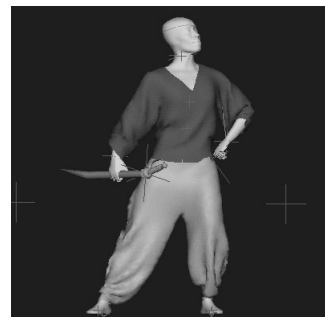
You can use constraints and dynamic fields to influence the settling of the cloth. For example, you can pull out wrinkles or pin portions of the garment to specific places on the character. You can set various attributes of the cloth to change the physical behavior of the cloth.

You can also create different types of fabric and affect the physical behavior of the garment. You can use different fabrics in the same garment, such as a baseball jacket with leather sleeves.

Animating the cloth

First, you play the animation from the dress-up pose to the animation start pose. This runs the simulation for the frames played, which animates the garment’s motion along the way. You do this to create a natural look for the garment at the beginning of the animation segment you’re creating.

Next, you change the frame range to span only the animation segment you will render, not the frames from the dress-up pose to the



4 | Cloth

> Preparing for the lessons

animation start pose. You won't use the positioning of the garment prior to this point, so you save its current position for use at the start of the new frame range.

You play the animation to run the simulation of the cloth's motion for the desired animation segment. To enhance the results, you run the simulation several times with various attribute settings. You can modify the garment's appearance and motion using constraints, dynamic forces, or keys.

Lesson workflow

In many production groups, it's typical for an animation specialist to animate the character in one scene while a cloth specialist constructs and fits the garments to a copy of the character in another scene. The fitted garments are then exported to the animation scene, where the cloth specialist positions them on the character and runs the cloth simulation for the animation.

The following lessons use this workflow. In the first two lessons, you drape a shirt and pants on a stationary 3D character. In the last lesson, you import a shirt and pants into a scene containing an animated version of the same 3D character. You position the shirt and pants on the character, then run a cloth simulation that causes the cloth to wrinkle and deform as the character moves. Because the lessons collectively illustrate a complete cloth workflow, it's best to do them in the order presented.

Keep in mind that, in your own work, you can create the garments and animate the character in a single scene.

Preparing for the lessons

To ensure the lesson works as described, do these steps before beginning:

- 1 Make sure you understand basic concepts of polygonal modeling, animation, and character setup.
- 2 Select the Cloth menu set.

If Cloth doesn't appear in the menu set selection menu, select Window > Settings/Preferences > Plug-in Manager. In the Plug-in Manager window, click the *loaded* checkbox for CpClothPlugin.so (IRIX, Linux) or CpClothPlugin.mll (Windows). Close the Plug-in Manager window. Maya loads Cloth and adds the Cloth menu set.

Unless otherwise noted, the procedures in this book assume the Cloth menu set is selected.

- 3 Select Window > Settings/Preferences > Preferences. Click the Timeline category and set Playback Speed to Play every frame. This setting is necessary for the cloth simulation to work correctly. Click the Save button.
- 4 Obtain the Cloth lesson data from the *Maya Documentation, Lessons, and Extras* CD. You can either copy the following directory from the CD or work from it directly:

UnlimitedLessonData/Cloth/scenes

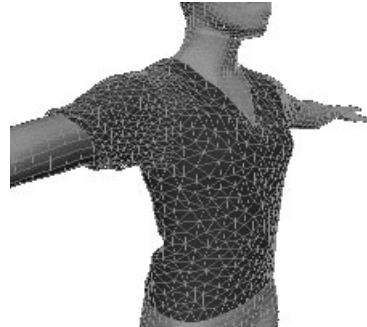
We provide a completed scene for each of the lessons in this chapter. If you want to see the expected results of the lessons, you can open the scenes from UnlimitedLessonData/Cloth/scenes on the *Maya Documentation, Lessons, and Extras* CD or from the drive where you copied the Getting Started with Maya Unlimited data files. The scene filenames end with `Final.mb`, for instance, `importFinal.mb`.

4 | Cloth

Lesson 1 > Making a Shirt

Lesson 1 Making a Shirt

Creating a cloth garment in Maya has many similarities to making a real piece of clothing. One similarity is that you need to start with a flat pattern of your garment. In this lesson, you learn how to create a shirt and fit it onto a stationary character. You learn how to:



- Create panels from closed curves.
- Create a polygonal mesh garment from panels.
- Define a property for a garment that defines its behavior when it moves.
- Run a simulation to position and shape a garment on a character model.
- Modify attributes to make a garment drape more naturally.

Lesson setup

In this lesson, you work with a scene we created for your use.

- 1 Before beginning the lesson, do the steps in "Preparing for the lessons" on page 116.
- 2 Open the scene named `shirtStart.mb`.
- 3 You can open the scene named `shirtStart.mb` from the *Maya Documentation, Lessons, and Extras* CD or from the drive where you copied the Getting Started with Maya Unlimited data files. The file is located in `UnlimitedLessonData/Cloth/scenes`.

This scene provides a polygonal model, named jackie, and the curves for creating the panels of the shirt. The curves for the front and back panels are planar and form closed boundaries. This is important to remember when creating curves for your panels.

Creating panels

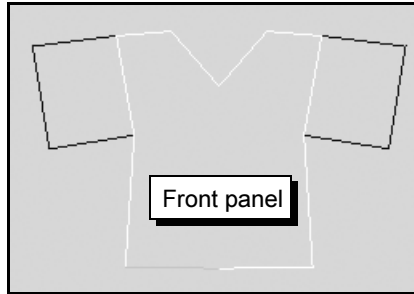
In the next steps, you'll create panels to be assembled as a shirt.

To create the front vest panel

- 1 In the Outliner, open the shirtPattern group.
- 2 Select curvesBack and press Ctrl-h to hide the curves.
You'll work only with the front curves for the next several steps, so it's helpful to hide the back curves to reduce clutter in the scene.
Note that the grey color of jackie's wireframe indicates that jackie is displayed as a template, which is an object that is visible, but not selectable in the view. We made jackie a template so you can't select it unintentionally.
- 3 In the Outliner, select and hide jackie (Ctrl-h) to reduce clutter in the scene.
- 4 Select the curves that represent the front vest part of the shirt. You can select the curves in any order. If you want to use the Outliner, open the curvesFront group and select curve1 through curve5 and curve9 through curve13.

4 | Cloth

Lesson 1 > Creating panels



The curves form a closed loop and lie in the same plane. The end of one curve is where the next curve begins. Curves must meet these requirements to be used for creating a panel.

- 5 Select Cloth > Create Panel

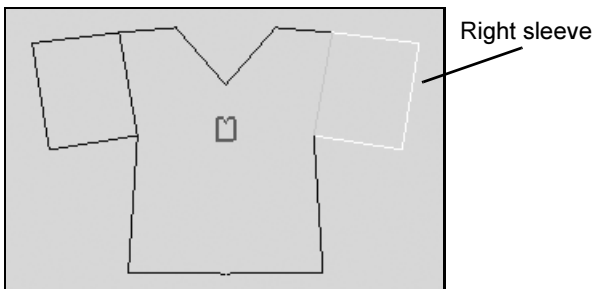
The Cloth Panel icon is displayed, indicating you've created a panel.



Panel icon

To create the front sleeve panels

- 1 Select the curves for the right sleeve and create a panel. (In the Outliner, these are curve3, and curve6 through curve8.)



- 2 Select the curves for the left sleeve and create a panel. (In the Outliner, these are curve11, and curve 14 through curve16.)
- 3 In the Outliner or view, select panel1, panel2, and panel3.
- 4 Select Edit > Group to group the panels.
- 5 Name the group `panelsFront` in the Channel Box, then parent `panelsFront` under `shirtPattern` by middle-dragging `panelsFront` over top of `shirtPattern` in the Outliner.

We provided `shirtPattern` group in the scene to help you organize your curves and panels. Grouping and parenting objects in this way is not necessary when creating a garment. However, organizing the elements of your garment makes it easier to locate them in the Outliner if the need arises. Grouping also makes it easy to hide or display a number of related elements.

To create the back vest panel

- 1 Hide `panelsFront` and `curvesFront` by selecting Display > Hide > Hide Selection, as you won't be working with them for awhile.
- 2 In the Outliner, select `curvesBack` and select Display > Show > Show Selection. You'll use these curves to create the back part of the shirt.
- 3 Select the curves that represent the back vest part of the shirt. To use the Outliner, open the `curvesBack` group and select the following curves using the Ctrl key with your left mouse button to select:
 - `curve17` through `curve20`
 - `curve24` through `curve27`
 - `curve31`
- 4 Select Cloth > Create Panel to create a panel.

4 | Cloth

Lesson 1 > Creating a garment

To create the back sleeve panels

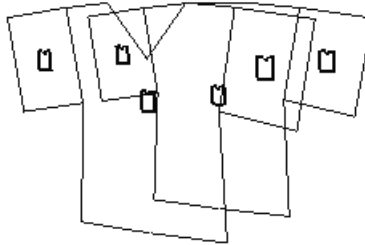
- 1 Select the curves for the right sleeve. To use the Outliner, select curve18 and curve21 through curve23. Select Cloth > Create Panel.
- 2 Select the curves for the left sleeve. To use the Outliner, select curve25, and curve28 through curve30. Select Cloth > Create Panel. You now have three panels for the back of your shirt.
- 3 Group the three panels and name the group `panelsBack` in the Channel Box.
- 4 For the sake of scene organization, parent `panelsBack` under `shirtPattern` by middle-dragging `panelsBack` over top of `shirtPattern` in the Outliner.

Creating a garment

Now that you've created all the panels, you're ready to create the garment. A *garment* is a collection of panels.

To create a garment

- 1 Select Display > Show > Show Cloth > All. This displays all curves and panels that exist in the scene.

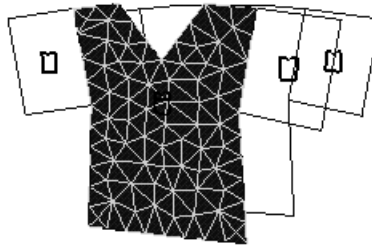


- 2 In a perspective view, rotate the camera so you can see the front and back panels.
- 3 Select the middle front panel, being careful not to select a curve in addition.

- 4 Select Cloth > Create Garment.

This creates a polygonal mesh garment named cloth1. It also creates a stitcher node, called cpStitcher, which lets you create seams between panels.

- 5 Press the 5 key to see the panel in smooth shaded mode.
- 6 In the Channel Box, click on cpStitcher and set the Base Resolution to 200.



This tessellates the cloth into smaller subdivisions. The higher the Base Resolution, the finer the tessellation. Finer tessellation means the surface allows finer shape alterations. The lower the Base Resolution, the faster the processing. As with other attributes described in this lesson, experience will guide your choice of values.

- 7 In the Outliner, change the name of cloth1 to shirt by double-clicking on the name.

Seaming panels in a garment

To include all panels of the shirt in the garment, you seam them to a panel in the garment. Currently the garment has only one panel—the middle front panel. When you seam a panel that's not in the garment to a panel in the garment, the new panel is added to the garment and the mesh is tessellated again. You can seam a panel only to a panel in the garment. You cannot seam to a panel that's not in the garment.

4 | Cloth

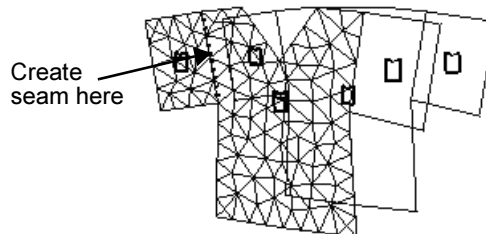
Lesson 1 > Seaming panels in a garment

You create seams much like you would sew seams in a shirt pattern. You create side seams, shoulder seams, and sleeve seams. If two panels share a curve, for instance, a front sleeve panel and the adjacent front vest panel, you pick the curve between them and create a seam.

If two panels do not share a curve, for instance, a back sleeve panel and a front sleeve panel, you select the appropriate pair of edge curves and create the seam. The technique will become clear in the following steps.

To seam the panels in the garment

- 1 Display the panel in wireframe mode. (Click in the view and press 4.)
- 2 In the Outliner, open the curvesFront and curvesBack groups.



- 3 Select curve11 and select Cloth > Create Seam. This seams together two panels that share the same curve.
- 4 In the Outliner, select the single curve or pair of curves listed below, then select Cloth > Create Seam to seam the panels together.

Selection		Action
curve3	.	Create Seam

Selection		Action
curve6	curve21	Create Seam
curve18	.	Create Seam
curve25	.	Create Seam
curve8	curve23	Create Seam
curve2	curve17	Create Seam
curve4	curve19	Create Seam
curve10	curve24	Create Seam
curve14	curve28	Create Seam
curve15	curve29	Create Seam
curve12	curve26	Create Seam

As you do this, observe how the resulting seams and panels differ based on which curve or curves you select. After you finish creating the seams, you can display all seams by selecting **Display > Show > Show Cloth > Seams**. The icons for seams between panels resemble stitches or ruler marks. A single row of stitch marks exists for a seam made between panels that share a curve. Two separated rows of stitch marks exist for a seam made between panels that do not share a curve, one row at each of the curves selected before you created the seam.

The easiest way to know if you've seamed the entire garment is to examine the garment in shaded mode. The garment will have no unintended gaps between the panels.

4 | Cloth

Lesson 1 > Creating and editing a property for a garment

Creating and editing a property for a garment

Cloth properties define how the cloth behaves as it moves, whether during the simulation or as the underlying object is animated. By default, each panel is connected to a property called *cpDefaultProperty*. However, you can create as many properties as you want and apply any property to any panel.

To create and edit a property for the garment

In the next steps, you'll create a property for the shirt and edit a pair of the many attributes available.

- 1 In the Outliner, select all the panels, but not the panelFront or panelBack group nodes.
- 2 Select Simulation > Properties > Create Cloth Property. This creates a new property node and assigns it to the selected panels.
- 3 In the Channel Box, rename cpProperty to cpPropertyShirt.

Tip To see which property node affects a panel, select the panel, and in the Channel Box you'll see the property node listed under INPUTS.

- 4 Set the U Bend Resistance and the V Bend Resistance attributes to 5.0.

These attributes control how much the cloth resists flexing. The default value is 10. Lower values increase the flexibility of the cloth. In this lesson, you use the same value for both attributes so the cloth resists flexing uniformly.

In some situations, it's useful to set the two attributes to different values to make the cloth flex more in one surface direction than the other. To know which surface direction is U

and which is V, examine the v notch in the panel icon. The v notch points in the V direction. U is perpendicular to the V direction.

Making a 3D character a collision object

To have cloth interact with a surface, you must make the surface a collision object before you run the simulation. A collision object can be a polygonal or NURBS surface. If you use a NURBS surface, Maya calculates the collision as if it were a polygonal surface. In the next steps, you'll make jackie a collision object. (Jackie is a polygonal object.)

To make the 3D character a collision object

- 1 In the Outliner, select jackie.
- 2 From the main menus, select Display > Show > Show Selection to display jackie.
- 3 Untemplate jackie by selecting Display > Object Display > Untemplate. This will let you see the cloth deform against jackie's surface more clearly.
- 4 With jackie still selected, select Cloth > Create Collision Object.
- 5 In the Channel Box, set Collision Offset to 0.2.
This specifies how far Maya strives to keep the cloth away from the collision object during the simulation. It's likely that parts of the garment will pass slightly through the Collision Offset distance occasionally during the simulation. When the garment pokes through, Maya repels the garment back above the surface again.
If a garment pokes through too much or too often, increase the Collision Offset and run the simulation again.
- 6 In the Channel Box, set Collision Depth to 0.7.

4 | Cloth

Lesson 1 > Running a cloth simulation

The Collision Depth sets the maximum distance any part of the cloth can penetrate the collision object and still be repelled back above the surface of the collision object.

If a garment passes the Collision Depth threshold, the collision object won't repel the cloth and allows it to pass through entirely. If this occurs, try a higher Collision Depth value and run the simulation again. Set the Collision Depth to a value no larger than half the depth of the collision object. In this context, depth means the distance between any opposing surfaces of the collision object.

Running a cloth simulation

Running a simulation refers to the processing that Maya does to position and shape the cloth on the collision object. The part of Maya that performs the simulation is the *cloth solver*. It assembles information on the cloth, any bodies it interacts with, and any fields or constraints applied to the cloth. Before you run the simulation, you'll set the Solver Scale attribute to enhance the realism of the shirt's deformation during the simulation.

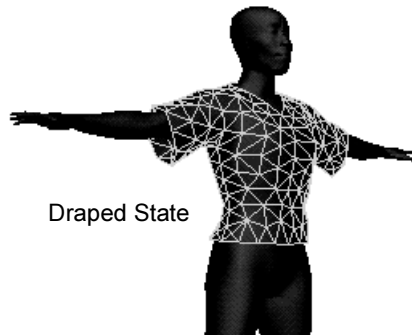
To run the cloth simulation

- 1 In the Outliner, select shirtPattern and press Ctrl-h to hide the seams.
- 2 In the Outliner, select shirt.
- 3 In the Channel Box, click cpSolver1 and set Solver Scale to 10.

The scene view size of the garment and 3D character influences the way a garment behaves. For instance, a tiny shirt hangs differently than a giant shirt.

Jackie is about 16 grid units (centimeters) in height. To make the cloth behave as a real-life shirt, you need to increase the Solver Scale to compensate for its small size. A value of 10 causes Maya to process the cloth as if it were positioned on a model 160 centimeters tall—roughly the size of a real person.

- 4 Go to the start time. Otherwise, after you run the simulation, the cloth will return to its original undraped position when you go to the start time.
- 5 To run a simulation, select Simulation > Start Local Simulation.



This relaxes (drapes) the shirt to the 3D character. The cloth wiggles slightly long after it snugs up against the character. This occurs because the cloth solver continuously recalculates the position of the cloth against the collision object in case any new movement or force influences the objects.

- 6 When the cloth settles into a desirable position, press Esc to stop the simulation.

If you are using Maya on IRIX or Linux, press Esc when you are satisfied with the shirt's appearance. If you are using a Windows workstation, select Simulation > Stop Simulation rather than press the Esc key.

You can use Undo to return the garment to its pre-simulation state.

4 | Cloth

Lesson 1 > Increasing a garment's resolution

Note If you play the scene, Maya runs the simulation for the entire animation time range, by default. As time changes, the cloth's motion is cached so you can later scrub the Time Slider to examine the cloth's appearance at various frames. The animation plays faster after caching because Maya uses disk-stored cloth positions for each frame.

If you decide to see the results of the simulation with different attribute settings, you must first delete the cache by selecting Delete Cache from the Simulation menu. Generally, you run the simulation by playing the animation only after you've draped the garment onto a stationary character with Simulation > Start Local Simulation.

With Simulation > Start Local Simulation, animation time does not advance. You use a local simulation to drape the garment onto a stationary character.

Increasing a garment's resolution

In the next steps, you'll run the simulation a few times after setting higher values for the garment's Base Resolution (tessellation). The garment drapes more naturally at a higher resolution, but processing takes longer. By running the simulation a few times with increasing resolution, the simulation times are shorter and the cloth deforms more smoothly.

To increase the Base Resolution for the garment

- 1 Select the shirt and display the Attribute Editor.
- 2 Click the cpStitcher1 tab.
- 3 Turn on Fit to Surface.

After you've run a simulation, it's necessary to turn on Fit to Surface before you change the Base Resolution of a garment, add seams, or modify the shape of a panel by editing its original curves. Otherwise, the garment's shape distorts.

- 4 Set the Base Resolution to 300.
- 5 Select Simulation > Start Local Simulation. When the shirt relaxes, press Esc.
- 6 Set the Base Resolution to 400.



High resolution cloth

- 7 Select Simulation > Start Local Simulation. When the shirt relaxes, press Esc.

Changing the Collision Offset

Now you decrease the Collision Offset and run the simulation again. This brings the shirt closer to the skin.

To decrease the Collision Offset for the simulation

- 1 Select jackie.
- 2 In the Channel Box, set the Collision Offset to 0.1.
- 3 Select Simulation > Start Local Simulation.
- 4 When the shirt relaxes, press Esc.

4 | Cloth

Lesson 1 > Beyond the lesson

- 5 To see what the cloth looks like with a shader applied, you can assign it a material that we created for your use in the lesson.
 - Select the shirt. Select the Rendering menu set. Select Lighting/Shading > Assign Existing Material > shirtMtl. Press 5 to turn on smooth shading.



Beyond the lesson

The behavior of cloth is controlled primarily by attributes in the collision object's shape node and by the cpSitcher, cpSolver, and cpProperty nodes. Most attributes that affect cloth, such as Shear Resistance, require experimentation to find the best setting.

Additional things to know when you create cloth patterns and garments:

- Planning your curves and panels is critical to creating good cloth in Maya.

This lesson provided curves already shaped into the outline of a shirt. Many of the panels shared a single curve across a seam. A good practice is to ensure that curves are not shared across a seam so that each panel has a unique set of closed boundary curves. Shared curves were used in this lesson for the sake of simplicity.

Before you draw the curves, you must consider the shape of the panels you plan to create. In general, it's necessary to create a panel for each part of the garment that you want to look or behave uniquely. For instance, to create a woolen jacket with leather sleeves, the sleeves need to be panels separate from the vest. For help in planning panels, consider looking at real fabric patterns online, at the library, or in a fabric store.

When you create your own curves for clothing, position the curves close to the body without intersecting it. You position a group of curves in front of the model and another in back of the model as shown in this lesson. If you modify the original curves after you create a garment, the garment updates automatically.

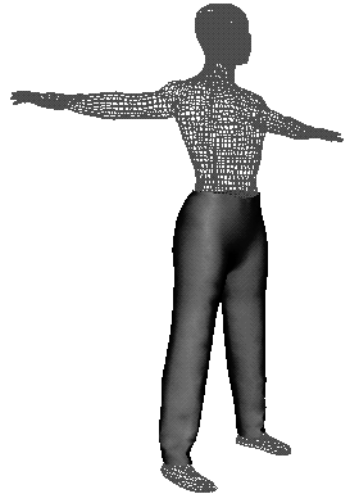
- Simulation options affect the final result.

When you work on your own garments, it's best to start by running the simulation with default option settings. Later, you run the simulation a few times with different settings until you achieve the garment look you're seeking.

Lesson 2 Making Pants

In this lesson, you make a pair of pants and fit it onto a stationary character. You also learn how to refine specific panels of the pants. In this lesson you learn how to:

- Duplicate a set of curves for a pattern.
- Create front and back panels for the garment.
- Set the solver and property attributes for a cloth simulation.
- Refine the simulation using the Base Resolution attributes.



Lesson setup

In this lesson, you work with a scene we created for your use.

- 1 Before beginning the lesson, do the steps in “Preparing for the lessons” on page 116.
- 2 Open the scene named `pantsStart.mb`.
- 3 You can open the scene named `pantsStart.mb` from the *Maya Documentation, Lessons, and Extras* CD or from the drive where you copied the Getting Started with Maya Unlimited data files. The file is located in `UnlimitedLessonData/Cloth/scenes`.


This scene includes jackie’s model and some curves you’ll use to make the pants. We provided the curves so you can focus on developing a garment rather than on creating curves. The model has been templated to make picking curves easier.

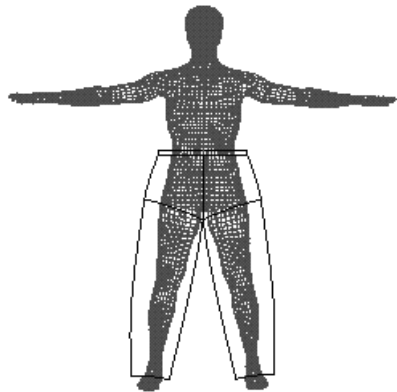
This lesson does not explain concepts and menu items described in the prior lesson. If you don't understand a term in this lesson, see the prior lesson.

Creating panel curves

The scene you just opened includes the curves for half a pair of pants. In the next steps, you'll duplicate the curves for the other half of the pants. Later in the lesson, you'll create panels from the curves and seam the panels together to form the pants.

To create the front curves

- 1 In the Outliner, note the pantsPattern group. This group includes two groups: curvesBack and curvesFront. Hide the curvesBack group because you won't work with them until later.
- 2 Open the curvesFront group so that all the curves in it are listed.
- 3 Select curve2 through curve9.
- 4 Select Edit > Duplicate > . From the options window, select Edit > Reset Settings, then continue as follows.
 - Set Scale for the X-axis to -1.



When you complete the Duplicate operation in a later step, the duplicate curves will be created in corresponding positions along the negative X-axis. This mirrors the left leg in the right leg position.

4 | Cloth

Lesson 2 > Creating front and back panels

- For Geometry Type, turn on Instance.

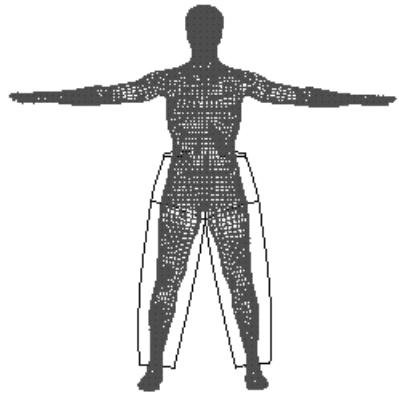
Instancing the curves makes modifying the pants easier; if you adjust the left leg, the right leg is adjusted identically.

- Click the Duplicate button.

- 5 Hide the curvesFront group. You won't work with them until later.

To create the back curves

- 1 In the Outliner, select the curvesBack group. Select Display > Show > Show Selection.
- 2 In the Outliner, open the curvesBack group so that all the curves in it are listed.
- 3 Select curve12 through curve19.
- 4 Select Edit > Duplicate. This uses the same Duplicate options that you used before. (Scale for the X-axis is -1, Geometry Type is Instance).
- 5 Hide the curvesBack group.



Creating front and back panels

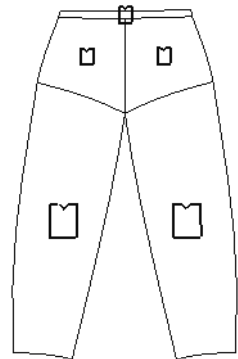
In the next steps, you'll create panels to be assembled as pants.

To create the front panels

- 1 Select jackie in the Outliner then hide jackie to reduce scene clutter (Ctrl-h).

- 2 Display curvesFront in the scene view. (Select it in the Outliner and select Display > Show > Show Selection.) With the curves in curvesFront, you can create five panels: one waistband panel, two hip panels, and two leg panels.
- 3 In the Outliner, select the curves below, then select Cloth > Create Panel.

Selection	Action
curve3, 4, 5, 21, 22, 23	Cloth > Create Panel
curve1, 2, 3, 6	Cloth > Create Panel
curve1, 20, 21, 24	Cloth > Create Panel
curve6, 7, 8, 9	Cloth > Create Panel
curve24, 25, 26, 27	Cloth > Create Panel



- 4 Group the panels and name the group panelsFront.
- 5 Parent panelsFront under the pantsPattern group. Grouping parts of the garment simplifies finding them in the Outliner if the need arises.
- 6 Hide panelsFront and curvesFront. You won't use them in the next steps.

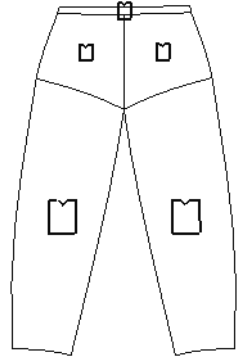
To create the back panels

- 1 Display curvesBack in the scene view.
- 2 In the Outliner, open the curvesBack group to list the curves. Select the following curves in the Outliner, then select Cloth > Create Panel.

4 | Cloth

Lesson 2 > Creating a garment

Selection	Action
curve16, 18, 19, 32, 34, 35	Cloth > Create Panel
curve11, 15, 16, 17	Cloth > Create Panel
curve11, 31, 32, 33	Cloth > Create Panel
curve12, 13, 14, 17	Cloth > Create Panel
curve28, 29, 30, 33	Cloth > Create Panel



- 3 Select panel6 through panel10.
- 4 Group the panels and name the group `panelsBack`.
- 5 Parent `panelsBack` under the `pantsPattern` group.

Creating a garment

In the next steps you create a garment from one of the panels so that you can then seam the other panels to the garment. The panel you use to create the initial garment is arbitrary.

To create the garment

- 1 Select `Display > Show > All`.
- 2 In the Outliner, open the `panelsFront` group to list the front panels.
- 3 Select panel4. Select `Cloth > Create Garment`.
- 4 In the Outliner, rename `cloth1` as `pants`.
- 5 With `pants` still selected, click `cpStitcher` in the Channel Box. Set the Base Resolution to 250.

Seaming panels in a garment

Now you'll seam the remaining panels into the garment.

- 1 In the Outliner, open the curvesFront and curvesBack groups to list the curves. In the Outliner, select the single curve or pair of curves listed below, then select Cloth > Create Seam to seam the adjacent panels together.

Selection	Action
curve6	Cloth > Create Seam
curve3	Cloth > Create Seam
curve21	Cloth > Create Seam
curve1	Cloth > Create Seam
curve24	Cloth > Create Seam
curve8,14	Cloth > Create Seam
curve2,15	Cloth > Create Seam
curve4,18	Cloth > Create Seam
curve17	Cloth > Create Seam
curve16	Cloth > Create Seam
curve7,12	Cloth > Create Seam
curve11	Cloth > Create Seam
curve32	Cloth > Create Seam
curve33	Cloth > Create Seam
curve25,28	Cloth > Create Seam
curve26,30	Cloth > Create Seam
curve20,31	Cloth > Create Seam
curve22,34	Cloth > Create Seam



4 | Cloth

Lesson 2 > Increasing the resolution of panels

- 2 Press 5 to smooth shade the pants. The garment will have no holes or gaps between the panels if you've seamed all the panels.

Increasing the resolution of panels

You can increase the resolution of specific panels to have regions of the garment deform more precisely than others during the simulation.

To increase the resolution of panels that affect the waistband

- 1 Select panel1 and panel6. These are the two panels that control the shape of the waistband.
- 2 In the SHAPES section of the Channel Box, set the Resolution Factor to 1.7. With the increased resolution, the waistband becomes smoother.

Creating a collision object

To have the pants interact with jackie, you must make jackie a collision object before you run the simulation.

To create the collision object for the simulation

- 1 Hide pantsPattern.
- 2 In the Outliner, select jackie and select Display > Object Display > Untemplate.
- 3 With jackie selected, select Cloth > Create Collision Object.
- 4 In the Channel Box, set Collision Offset to 0.1 and Collision Depth to 0.5.

Running and modifying a simulation

In the following steps, you'll run the simulation repeatedly with various option settings.

To set the solver and property options

Solver and property attributes control how the pants deform during the simulation.

- 1 Select Simulation > Solvers > cpSolver1 > to display the solver attributes.

The Attribute Editor appears.

- 2 Set the Solver Scale to 10.
- 3 Select Simulation > Properties > cpDefaultProperty > .
- 4 Set U and V Bend Resistance to 20.
- 5 You can optionally deselect jackie so that the surface is easier to see.
- 6 Select Simulation > Start Local Simulation.
- 7 When the pants fit well, press the Esc key to stop the simulation.

Do not be concerned if there are small spots where jackie pokes through the pants. You'll fix this at the end of the lesson.



To add and edit properties for specific panels

In the next steps you'll assign a property to selected panels, then set property attributes that affect the behavior of the panels.

- 1 Select Display > Show > Show Cloth > Panels.
- 2 Select panel1 and panel6, in the Outliner. These are the two waistband panels.
- 3 Select Simulation > Properties > Create Cloth Property.

4 | Cloth

Lesson 2 > Increasing the Base Resolution

This creates a new property node and assigns it to the selected waistband panels.

- 4 In the Channel Box, rename the `cpProperty` node to `cpPropertyBelt`, then set the U Scale to 0.8.
The U Scale value of 0.8 scales down the waistband panel in the U direction, which causes the cloth in the waistband to tighten. This takes effect only after you run the simulation again.
- 5 Select the two panels at the rear, `panel7` and `panel8` in the Outliner.
- 6 Select **Simulation > Properties > Create Cloth Property** and name the property `cpPropertyRear`.
- 7 In the Channel Box settings for `cpPropertyRear`, set the U and V Scale to 1.1. When you run the simulation, the cloth in the rear of the pants will fit a bit more loosely.
- 8 Select **Simulation > Start Local Simulation**. When the pants fit well, press the Esc key.

Increasing the Base Resolution

You might have noticed small regions where jackie pokes through the pants. This is a common when you run a simulation for a garment with relatively coarse tessellation. In the next steps you'll eliminate the interpenetration by increasing the tessellation of the pants.

- 1 In the Outliner, select the pants.
- 2 In the Attribute Editor, click the `cpStitcher` tab.



- 3 Set the Base Resolution to 300 and turn on Fit To Surface.
- 4 Select Simulation > Start Local Simulation.
- 5 When the pants fit well, press the Esc key.
If you want to see what the cloth looks like with a shader applied, you can assign it a material we created for your use in the lesson.
- 6 Select the pants. Select the Rendering menu set. Select Lighting/Shading > Assign Existing Material > pantsMtl to assign the material to the pants.
- 7 Press 5 to turn on smooth shading.

Beyond the lesson

In this lesson you learned that:

- The process for creating pants is roughly the same as for creating a shirt.
- Creating panels for your garments has a specific procedure for best results.
- The process of fitting a garment might require several iterative simulations.
- The Base Resolution affects the detail of the cloth as well as the interpenetration between objects.
- Regardless of the type of garment you create, you can have specific panels look and behave differently when simulated.

Interpenetration is a common result of using a low Base Resolution. However, it's best to start with a low Base Resolution because the simulation runs faster. By increasing the Base Resolution slightly with subsequent simulations, you'll spend less time waiting for lengthy simulations. Using repeated simulations after making small adjustments to option settings is also beneficial because it reduces the chance of the garment looking unnatural.

Lesson 3 Exporting & Importing Garments

In the previous lessons, you fitted the cloth garments to a character that was positioned in a rest pose. In production settings, a cloth garment may be fitted on one version of a character while another version is being animated. It is useful to know how to export the garment from one scene to another.

In this lesson, you export the pants and shirt from previous scenes and import them into a scene containing an animated jackie. In the scene, jackie animates from a dress-up pose to a complete walk cycle. You position the shirt and pants on jackie and run the simulation of the cloth for the entire frame range. In this lesson you learn how to:

- Export and import cloth garments between scenes.
- Position the garments between solvers.
- Transfer garments between solvers.
- Apply mesh constraints to keep garments in place.
- Adjust the garments for a better fit.



Exporting garments

You'll begin the lesson by exporting a shirt and pants. If you completed the prior lessons, the shirt and pants will look familiar. Before beginning, do the steps in "Preparing for the lessons" on page 116.

This lesson does not explain concepts and menu items described in the prior lessons. If you don't understand a term in this lesson, see the prior lessons.

To export the shirt

- 1 Open the scene named `shirtFinal.mb`.

You can open the scene named `shirtFinal.mb` from the *Maya Documentation, Lessons, and Extras* CD or from the drive where you copied the Getting Started with Maya Unlimited data files. The file is located in `UnlimitedLessonData/Cloth/scenes`.

The scene contains a shirt that was created by running a simulation on jackie.

- 2 Select the shirt. Make sure nothing else is selected.
- 3 Select File > Export Selection > . From the options window, select Edit > Reset Settings, make sure History is turned on, then click Export Selection.

With History on, Maya exports the cloth solver that controlled the shirt's simulation in `shirtFinal.mb`. You must include the solver because you'll run the simulation of the shirt again in the scene where you import it. In scenes where you don't plan to redo simulation of an imported garment, such as a stationary tablecloth, you can leave History turned off to avoid importing the solver.

- 4 Do one of the following, depending on your operating system:
 - (Windows) In the Export Selection window, click the down arrow in the top box to display the pull-down menu and select Current scenes.
 - (IRIX, Linux) In the Export Selection window, select File > Go to Current scenes.
- 5 Enter the name *shirtExport* and click Export. This creates a scene file named `shirtExport.mb`.

4 | Cloth

Lesson 3 > Exporting garments

To export the pants

- 1 Open the scene named `pantsFinal.mb`.
You can open the scene named `pantsFinal.mb` from the *Maya Documentation, Lessons, and Extras* CD or from the drive where you copied the Getting Started with Maya Unlimited data files. The file is located in `UnlimitedLessonData/Cloth/scenes`.
The scene contains a pair of pants that was created by running a simulation on jackie.
- 2 Select the pants.
- 3 Select File >Export Selection.
This export operation uses the same options you set the last time you used File > Export Selection > .
- 4 Do one of the following, depending on your operating system:
 - (Windows) In the Export Selection window, click the down arrow in the top box to display the pull-down menu and select Current scenes.
 - (IRIX, Linux) In the Export Selection window, select File > Go to Current scenes.
- 5 Enter the name *pantsExport* and click Export. This creates a scene file named `pantsExport.mb`.

To open the animation scene

- 1 Open the scene named `importStart.mb`.
You can open the scene named `importStart.mb` from the *Maya Documentation, Lessons, and Extras* CD or from the drive where you copied the Getting Started with Maya Unlimited data files. The file is located in `UnlimitedLessonData/Cloth/scenes`.

Notice that jackie has the same dress-up position and scale as in the export scenes. This is a requirement for importing exported garments. In this scene, however, jackie has been moved and rotated away from the origin. In this lesson, you'll move and rotate the imported garments to fit them properly on jackie.

- 2 Make sure the frame range is -20 to 118.
- 3 Play the scene to become familiar with the animation.
In frames -20 to 1, jackie makes a transition from the dress-up pose to the start position of the walk cycle. In frames 1 to 118, jackie is animated in a walk cycle.
- 4 Go to the start time of the animation (frame -20).

To import the shirt

- 1 Select File > Import > . From the options window, select Edit > Reset Settings, turn on Use Namespaces, then click the Import button.

The option settings cause Maya to rename all imported objects, solvers, and other nodes so they cannot clash with existing names in the scene.



- 2 Do one of the following, depending on your operating system:
 - (Windows) In the Import Selection window, click the down arrow in the top box to display the pull-down menu and select Current scenes.

4 | Cloth

Lesson 3 > Positioning an imported garment

- (IRIX, Linux) In the Import Selection window, select File > Go to Current scenes.
- 3 Select `shirtExport.mb` and click Import. This imports the shirt.
 - 4 In the Outliner, select `shirtExport:shirt`.
In the Channel Box, notice the names `shirtExport:cpSolver1` and `shirtExport:cpStitcher1`. Because of the options you selected in the Import options window, Maya added `shirtExport:` as a prefix to the `cpSolver1` and `cpStitcher1` nodes that were imported from the `shirtExport.mb` file. This ensured that the names would be unique after being imported to `importStart.mb`.

Positioning an imported garment

The imported shirt is positioned at the origin. You need to position the shirt on jackie.

To position the shirt on the character

- 1 Move and rotate the shirt so it fits well on jackie. Press 5 to turn on smooth shading so you can see where the shirt penetrates the skin.

A perfect fit is not necessary. Running the simulation will push out the shirt. However, make sure the positioning is close to the following values:



- Translate X: 9.93

Lesson 3 > Creating a collision object for an imported garment

- Translate Y: 0.1
 - Translate Z: -9.98
 - Rotate Y: -24
- 2 In the Channel Box, click `shirtExport:cpSolver1`. Set the Start Frame to -20 in the INPUTS section of the Channel Box. When you run the simulation, it will start from frame -20.
 - 3 From the Cloth menu set, select Simulation > Save as Initial Cloth State.
Maya will use the current shirt position and shape at the start of the animation (frame -20).

Creating a collision object for an imported garment

You need to make jackie a collision object so the shirt drapes on jackie when you run the simulation.

To create a collision object

- 1 Select jackie.
- 2 Select Cloth > Create Collision Object.
- 3 In the Channel Box, set the Collision Offset to 0.16 and the Collision Depth to 0.2.

Ignore *Warning: Cycle on joint...* messages after you make jackie a collision object. To prevent such messages, enter `cycleCheck -e off` in the Command Line.

To run the simulation

- 1 Select Simulation > Start Local Simulation.

Within a minute or so, the shirt drapes nicely on jackie without interpenetration. Press Esc when you are satisfied with the shirt's display.

Next, you'll import the pants and position them on jackie.

4 | Cloth

Lesson 3 > Creating a collision object for an imported garment

To import and position the pants

- 1 Hide the shirt. The name of the shirt is `shirtExport:shirt`.
- 2 Select File > Import. The importing will occur with the same options used when you imported the shirt.
- 3 Do one of the following, depending on your operating system:
 - (Windows) In the Import Selection window, click the down arrow in the top box to display the pull-down menu and select Current scenes.
 - (IRIX, Linux) In the Import Selection window, select File > Go to Current scenes.
- 4 Select `pantsExport.mb` and click Import. This imports the pants.



- 5 Move and rotate the pants on Jackie so they fit as well as possible. The pants are named `pantsExport:pants`. A suitable position is at:
 - Translate X: 10
 - Translate Y: 0

- Translate Z: -10
 - Rotate Y: -30
- 6 With the pants selected, click pantsExport:cpSolver1 in the Channel box. Set the Start Frame to -20. When you run the simulation, it will start from frame -20.
 - 7 Select Simulation > Save as Initial Cloth State.
Maya will use the current position and shape of the pants at the start of the animation (frame -20).

Importing solvers

When you imported the shirt and pants, Maya also imported a pants solver and a shirt solver. The names of the solvers, pantsExport:cpSolver1 and shirtExport:cpSolver1, are listed under the menu Simulation > Solvers >.

Currently, the solvers have no effect on the pants and shirt because you have not connected them to jackie in this scene. (The solvers were previously connected to a different jackie in the exported scenes.)

In the next steps, you'll make jackie a collision object that interacts with the pants during simulation. You'll also run a simulation to correct jackie's interpenetration of the pants.

To create a collision object for the pants

- 1 Select Simulation > Solvers > pantsExport:cpSolver1. This makes the pantsExport:cpSolver1 solver the current solver.
- 2 Select jackie.
- 3 Select Cloth > Create Collision Object.
- 4 Select the pants.

4 | Cloth

Lesson 3 > Transferring solvers

Selecting the pants specifies which solver you want to use. Even though `shirtExport:cpSolver1` has no effect because it's not connected to `jackie`, you must still select the pants to make `pantsExport:cpSolver1` operate when you run the simulation.



If a scene has two or more solvers, only one can operate at a time during a local simulation. Selecting the pants specifies which solver operates. (You need to do this even if one of the solvers is not connected.)

To run the simulation for the pants

- 1 Select Simulation > Start Local Simulation.
- 2 Press Esc when you are satisfied with the pants.

Transferring solvers

For the shirt to drape over the pants and not just `jackie`, both garments need to be connected to the same solver. To do this, you'll transfer the pants to the shirt's solver. Be aware that if two garments do not touch and you do not expect them to touch during an animation, they don't need to share a solver.

To transfer the pants solver to the shirt

- 1 Select Simulation > Solvers > shirtExport:cpSolver1.
This makes the shirt's solver current.
- 2 With the pants selected, select Simulation > Transfer Garment.
This transfers the pants to the shirt's solver. To confirm that the pants are controlled by the shirt solver, check that shirtExport:cpSolver1 is displayed in the INPUTS section of the Channel Box.
- 3 In the Command Line, enter:

```
delete pantsExport:cpSolver1
```


This deletes the pants solver, which is no longer needed.

Adding mesh constraints

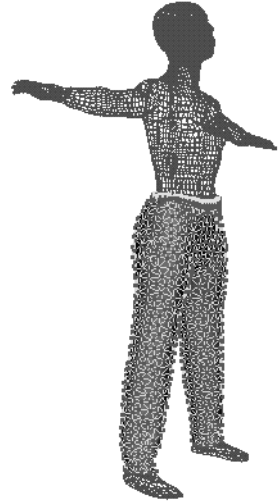
To prevent the pants from sliding down as jackie walks, you'll use mesh constraints to hold the pants on jackie's waist. This works like a belt.

4 | Cloth

Lesson 3 > Manually adjusting garments

To add a mesh constraint for the pants

- 1 Select jackie and select Display > Object Display > Template.
- 2 Right-click over the pants and select Vertex.
- 3 Select the top row of vertices on the pants. Use Shift-select to add vertices to the selection (or to remove vertices).
- 4 In the Outliner, Ctrl-click jackie. This adds jackie to the selection without deselecting the vertices.
- 5 Select Constraints >Mesh to create the mesh constraints. Mesh constraint icons appear on the pants, showing where the pants are constrained to jackie.
- 6 Hide the mesh constraints. Because the mesh constraints are currently selected, you can hide them by pressing Ctrl+h. In case you want to display the constraints later, be aware that the mesh constraints are indented under pantsExport:pants.



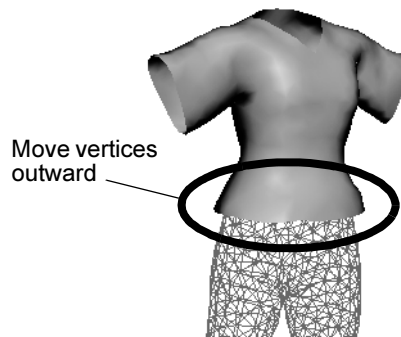
Manually adjusting garments

Next, you have Maya drape the shirt over the pants without interpenetration.

To manually adjust the shirt

- 1 Untemplate jackie.
- 2 Show (unhide) the shirt.
Notice the pants poking through the shirt.
- 3 Hide jackie to reduce screen clutter.

- 4 Template the pants so you can't pick any of its vertices.
- 5 With the pointer over the shirt, right-click and select Vertex from the marking menu.
- 6 Select all vertices that overlap the pants and additionally a small band of nonoverlapping vertices above the pants. If you can't see the overlapping region clearly, press 4 for wireframe display, then dolly into the view.
- 7 Use the Move and Scale tools to move the vertices away from the pants with plenty of clearance. Avoid creating sharp edges on the surface of the shirt.



- 8 Select the shirt and select Simulation > Update Cloth State.
This updates the solver with the new positions of the vertices. This is essential for the next simulation to work correctly.
- 9 Select Simulation > Properties > shirtExport:cpPropertyShirt > . Enter a Thickness of 1.5 and a Thickness Force of 3.5.
The Thickness specifies the collision offset of the cloth when it collides with itself and other cloth objects. Thickness Force sets how much force Maya uses to maintain the Thickness collision offset during cloth-to-cloth collisions. The default values for Thickness and Thickness Force are 0.2 and 1. The higher values you use in this step help keep the shirt further away from the pants when you run the simulation.

4 | Cloth

Lesson 3 > Running a simulation for an animation sequence

- 10 Show jackie and untemplate the pants.
- 11 With the shirt selected, select Simulation > Start Local Simulation.

If you are using Maya on IRIX or Linux, press Esc when you are satisfied with the shirt's appearance. If you are using a Windows workstation, select Simulation > Stop Local Simulation rather than press the Esc key.

After the simulation, if the shirt is in a position where it penetrates the pants, select Edit > Undo to return the shirt to its previous position. In the area where the shirt moved under the pants, move the vertices further away from the pants. Then select the shirt, Simulation > Update Cloth State, and Simulation > Start Simulation.

Running a simulation for an animation sequence

In the next steps, you run the simulation from the dress-up pose to the beginning of the walk cycle. You do this to create a natural look for the pants and shirt at the beginning of the walk cycle. Later, you'll run the simulation for the walk cycle.

To run the simulation for the first part of the animation

- 1 In the Time Slider, set the end time to 1 and make sure the start time is -20.
- 2 Select Simulation > Solvers > shirtExport:cpSolver1 > . Set the following attributes:
 - Start Frame: -20
 - Frame Samples: 2
 - Relax Frame Length: 0

Lesson 3 > Running a simulation for an animation sequence

The Frame Samples value sets how often the solver does cloth calculations during the simulation. Higher values increase precision but with longer processing.

By default, a simulation is a two-part process. First, the solver fits the garments to the character without the effect of gravity. Second, it fits the garments with the addition of gravity.

The Relax Frame Length value specifies the length of time that Maya fits the garment to jackie before gravity's effect begins. Generally, the larger the number of frames, the closer the fit. Because the gravity alone will fit a shirt to the character suitably, you set the Relax Frame Length to 0.



- 3 Select the shirt. Select Simulation > Delete Cache.

Whenever you change cloth properties or other attributes that affect the garment, you must delete the cache before you run the simulation. Otherwise the results of the simulation won't reflect your changes.

- 4 Select Window > Settings/Preferences > Preferences. Click the Timeline category. Set Looping to Once then click Save.

With Looping set to Once, Maya doesn't needlessly play the animation after you run the simulation.

- 5 Go to the start of the playback range and play the animation. Playing the animation runs the simulation from frame -20 to 1. At frame 1, the shirt and pants lie naturally on jackie at the beginning of the walk cycle.

4 | Cloth

Lesson 3 > Saving the initial state for garments

Saving the initial state for garments

In the preceding steps, you created the desired look of the pants and shirt at the start of the walk cycle. In the next steps, you'll save the current positioning of the pants and shirt as their position at the start time of the animation.

To save the initial state for the shirt and pants

- 1 Select the shirt.
- 2 Select Simulation > Save as Initial Cloth State.
- 3 Select the pants.
- 4 Select Simulation > Save as Initial Cloth State.

Now you'll run the simulation for the walk cycle.

To run the simulation for the walk cycle

- 1 Set the Time Slider's start and end times to 1 and 118. This is the frame range where you'll run the simulation.
- 2 Select Simulation > Solvers > shirtExport:cpSolver1 > and set the Start Frame to 1. This value specifies the frame at which the solver starts the simulation during playback.
- 3 Select Simulation > Delete Cache.
- 4 Play the animation. This runs the simulation for the walk cycle.

After the simulation runs, the cloth's motion is cached, so you can scrub the Time Slider to evaluate the look of the cloth at various frames. Playback is also faster.

If you decide to simulate again with different attribute settings, you must first delete the cache. Otherwise, scrubbing and playback will use the cloth motion from the prior simulation.

Note that when you cache a cloth simulation, the directory where the scene exists will contain a file with the extension .mcc, for example, pantsFinal.mcc. This file contains the caching information for the scene. If you no longer use the cache for a scene, you can delete the file.

Eliminating interpenetration

If you examine the garments closely at various frames, you might see spots where jackie penetrates the pants or where the pants penetrate the shirt. Eliminating the problem requires experimentation.

Try one of the following solutions, then select Simulation > Delete Cache and play the animation. If the interpenetration problem persists, try another solution, delete the cache, and play the animation again.

To solve interpenetration problems

- Select jackie and increase the Collision Offset a small amount, for instance, from 0.16 to 0.17.
- For interpenetration between the pants and shirt, select Simulation > Properties > shirtExport:cpPropertyShirt > . Increase the Thickness and Thickness Force a small amount. For example, change Thickness from 1.5 to 1.6, and change Thickness Force from 3.5 to 3.7.
- Select the garment that has been penetrated. In the Attribute Editor, click the garment's cpStitcher tab. Increase the Base Resolution a small amount, for instance, from 300 to 330, and make sure Fit To Surface is turned on. Increasing the Base Resolution slows the simulation considerably.

As a quick fix to a few frames before rendering, you can drag vertices of a garment into a satisfactory position and simply render the offending frames.

4 | Cloth

Lesson 3 > Beyond the lesson

Beyond the lesson

In this lesson you learned how to:

- Transfer garments between scenes using Import and Export.
- Apply mesh constraints to garments.
- Manually adjust the polygonal mesh of a garment for a better fit.
- Set cloth states for the garments.
- Run a cloth simulation for an animated walk cycle.

In this lesson, you ran the simulation once for the transition from the dress-up pose to the start of the walk cycle, and once for the walk cycle. The reason you split the simulation into two parts was to avoid repeating simulation of completed frames needlessly.

Your goal for running a simulation up to the start of the walk cycle is to have the shirt and pants look natural in their initial position where the character begins the walk cycle. After you've accomplish this, there's no need to simulate the prior frames again. When you run the simulation for the walk cycle, you typically need to experiment with various attribute values and run the simulation several times.

Other cloth features

The Maya Help provides additional information on other features of cloth. For instance, you can:

- Use the mouse or tablet stylus to paint cloth stiffness, wrinkling, and other effects.
- Use constraints and dynamic fields to influence the way the garment drapes on the character. For example, you can pull out wrinkles or you can pin portions of the garment to places on the character.
- Set various attributes to create different types of fabric that affect the behavior of the garment.

- Use different fabrics in the same garment, such as a baseball jacket with leather sleeves.
- Attach subassemblies such as collars.

Using stationary cloth

In some scenes, you might want to use a cloth object that doesn't change position or shape during playback. For example, you might want to include an unmoving tablecloth.

To create stationary cloth

- 1 Run the simulation and stop it when the cloth has the desired shape.
- 2 Save the current shape as the initial cloth shape.
- 3 You can do either of the following:
 - Delete the cache. Select the appropriate solver and set its Start Frame attribute higher than the scene's time range. This prevents simulation during subsequent playback. If you don't do this step, the cloth changes shape a bit as the simulation runs.
 - Export the cloth object with History turned off, and import it into the desired scene.
- 4 To practice creating a stationary tablecloth on your own, use the scene `UnlimitedLessonData/Cloth/scenes/tableClothStart.mb`.

You can open the scene named `tableClothStart.mb` from the *Maya Documentation, Lessons, and Extras* CD or from the drive where you copied the *Getting Started with Maya Unlimited* data files.

This scene has a table and a group of four curves you can use to make the tablecloth.

4 | Cloth

Lesson 3 > Beyond the lesson



- 5 The four curves are connected as a square above the table. Though garments such as shirts and pants need many panels seamed together, a simple garment such as a tablecloth needs only one panel. The scene includes a material named `tableClothMtl` that you can assign to the cloth for a more realistic display.

5

Fur

With the fur feature, you can create fur and short hair on all surface types in Maya. You can set fur attributes such as color, width, length, baldness, opacity, curl, and direction for entire surfaces or specific regions.

To bring fur to life, you can keyframe effects like growing fur or changing color. For a more natural appearance, you can also add movement to fur with keyframes and dynamics, for example, wind and gravity.

This chapter has the following topics:

- “Preparing for the lesson” on page 163
- “Lesson 1: Applying fur to a surface” on page 164

Preparing for the lesson

To ensure the lesson works as described, do these steps before beginning:

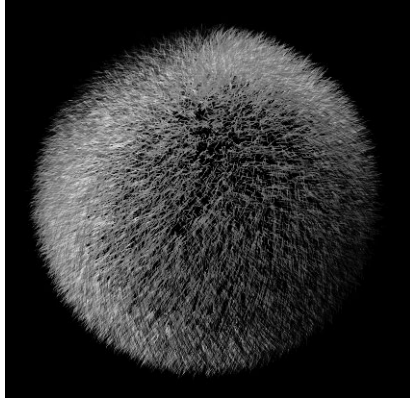
- 1 Create a new scene.
- 2 Select the Rendering menu set. Unless otherwise noted, the directions in this chapter for making menu selections assume you’ve already selected the Rendering menu set.
- 3 If you don’t see a Fur menu next to the Paint Effects menu, perform these additional steps:
 - Select Window > Settings/Preferences > Plug-in Manager.
 - In the Plug-in Manager, locate the Fur plug-in (Fur.mll) and turn on the *loaded* checkbox. Wait for the check mark to appear in the checkbox, then close the Plug-in Manager. You can set the Fur plug-in to *auto load* so that it automatically loads each time the Maya application is launched. Notice that Maya added the Fur menu next to the Paint Effects menu.

5 | Fur

Lesson 1 > Applying fur to a surface

Lesson 1 Applying fur to a surface

In this lesson, you learn about essential Fur features by making a fuzzy ball:



In this lesson you learn how to:

- Attach a fur description to an object.
- Modify fur attributes.
- Render the scene to view the fur.

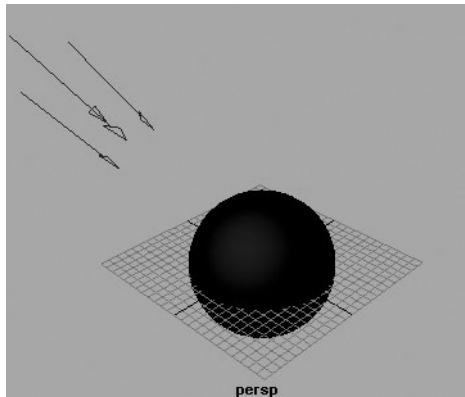
Lesson setup

You will begin the lesson by creating a light, a sphere, and a gold material for the sphere. Later in the lesson, you'll apply fur to the sphere.

- 1 Select Create > Lights > Directional Light.
- 2 In the Channel Box, enter -50 for the light's Rotate X value, and 20 for its Rotate Y value.
- 3 Move the light away from the center of the grid so that it's easy to select after you add a sphere.

A directional light's orientation, not position, affects the aim of the lighting.

- 4 In your scene view, select **Lighting > Use All Lights** so that the scene is lit by the directional light rather than the default light. The directional light will display the fur more clearly than the default light.
- 5 Create a polygonal sphere with a radius of 8. Name it **Ball**.
- 6 Select **Shading > Smooth Shade All** to display the **Ball** with smooth shading.
- 7 Right-click the **Ball** and select **Materials > Assign New Material > Lambert** from the marking menu.
- 8 Click the **Color** box to display the **Color Chooser**, choose a gold color, then click the **Accept** button.
- 9 Close the **Attribute Editor**.



5 | Fur

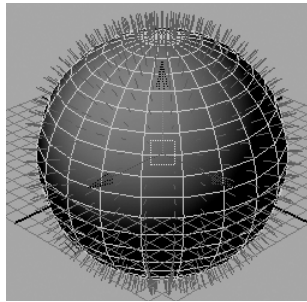
Lesson 1 > Creating fur on a surface

Creating fur on a surface

The next steps show how to attach fur and edit the fur attributes.

To create fur on a surface

- 1 With Ball selected, select Fur > Attach Fur Description > New. This attaches a new default fur description to the selected surface. A fur description defines all the attributes for the fur, for example, fur color, width, length, baldness, opacity, curl, density, and so on. The spikes are known as the *fur feedback*—a rough approximation of how the fur will look when rendered.



- 2 In the scene view, click a spike to select the fur feedback node. (Select a spike at the perimeter of sphere so you don't unintentionally select the sphere also.)

The fur feedback (spikes) becomes highlighted.

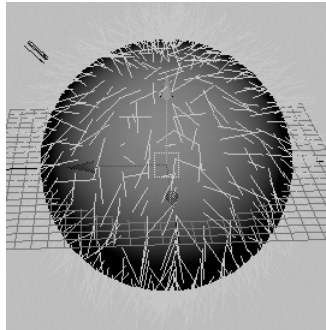
Modifying fur attributes

- 1 In the Inputs section of the Channel Box, select FurDescription1. Rename FurDescription1 as ballFurDescription. This gives the fur description a name that's easier to remember than the default name.

Enter these attribute values:

- Length: 3

- Inclination: 0.4
- Scraggle: 0.4
- Polar: 0.75



Length sets the fur's length, in grid units.

Scraggle sets the crookedness of the fur. A value of 0 creates no crookedness, while 1 creates maximum crookedness.

Inclination sets how the fur slopes or leans. A value of 0 is fully erect (normal to the surface), while 1 is flat (tangent to the surface at the root).

Polar sets how much the fur is rotated about the surface normal. 0 creates -180 degree rotation, 0.5 creates no rotation, and 1 creates 180 degree rotation.

- 2 In the Shapes section of the Channel Box, ensure the Fur Accuracy is set to 1.

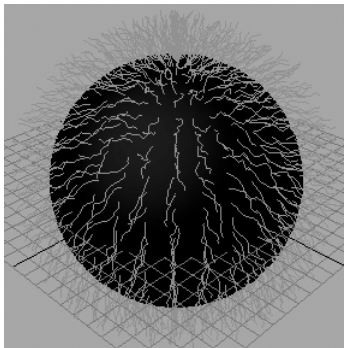
Fur Accuracy sets how closely the fur's display in the scene view resembles the fur when rendered. A value of 1 displays best accuracy, 0 least accuracy. With a value of 1, you can see the effect of scraggle and curl without having to render. High values slow Maya operation speed.

- 3 Select Fur > Edit Fur Description > ballFurDescription to edit the fur description attributes.

5 | Fur

Lesson 1 > Modifying fur attributes

- 4 Set the Base Color and Tip Color by clicking each color box and selecting the color from the Color Chooser. For example, choose a yellow-gold for the Base Color and an orange-gold for the Tip Color.
- 5 Enter 0.75 for Tip Opacity and 0.01 for Tip Width.
Tip Opacity sets the transparency of the ends of the hair. A value of 0 is transparent, while 1 is completely opaque. *Tip Width* sets the width of each hair at the tip.
- 6 To have the fur's color in the scene view approximate the fur when rendered do the following:
 - Select one of the spikes to select the fur feedback node.
 - In the Shapes section of the Channel Box, set Color Feedback Enabled to *On*.

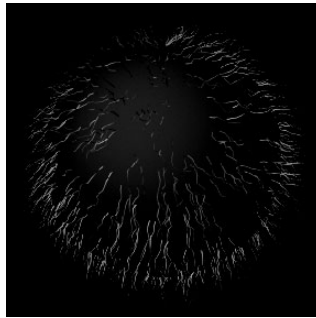


Rendering fur

To see the fur in full detail, you need to render the scene.

To render the fur

- 1 Select Window > Rendering Editors > Render Globals and click on the Common tab.
- 2 In the Resolution section of the Render Global Settings window, select 640x480 from the Presets drop-down list.
When you test render the scene, the resulting image size (640 by 480) will be large enough to see fine detail.
- 3 Select Render > Render Current Frame.



The fur looks too sparse. Fortunately, only a few easy steps are necessary to increase the fur density.

To modify the fur density

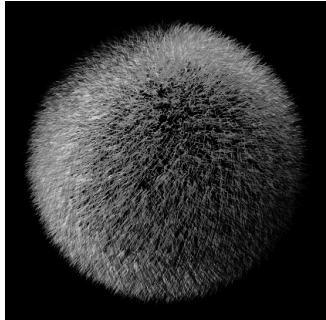
- 1 In the scene, select a spike that represents the fur.
- 2 In the Inputs section of the Channel Box, click `ballFurDescription`.
- 3 Enter 30000 for Density.

This specifies the number of hairs on the surface. The higher the value, the thicker the fuzz.

5 | Fur

Lesson 1 > Beyond the lesson

- 4 In the Render View window, select Render > Redo Previous Render.



- 5 Experiment with other attribute values and use Render > Redo Previous Render as necessary.

Beyond the lesson

Before exploring how to apply the lesson, it's worthwhile to review the general steps of using fur:

- Create the desired lights.
- Attach a fur description to the surface.
- Modify fur attributes.
- Animate fur attributes (optional, not shown in this lesson).
- Add movement to fur (optional, not shown in this lesson).
- Set up fur shading effects (optional, not shown in this lesson).
- Render the scene.
- Refine attribute settings and render again.

In the lesson, you attached a new fur description to a single selected surface. You can also attach a fur description to multiple selected surfaces, for instance, a cat's body, legs, and head.

Though not covered in the lesson, it's often useful to alter the fur attributes using the Paint Fur Attributes Tool. With this tool, you paint values for fur attributes on a specific region. For example, you can paint length values on part of a surface to make the painted patch of fur longer than the rest of the fur. With this tool, you can even paint the direction of the fur as if you were using a comb.

You can also attach more than one fur description to a surface. For example, you can add an additional fur description to a surface around a cat's nose for its whiskers. Note that only the most recently applied fur description is displayed in the scene view.

Though not included in the lesson, you can keyframe changes you make to fur attributes to animate effects such as growing fur or changing fur color. You can add movement to fur with attractors and dynamics. Using attractors, you manually keyframe fur movement. Using dynamics, you make the fur react to forces, for example, wind and gravity. You can also use dynamics to cause fur to react to movement of the attached surface, for example, when a dog shakes itself.

Finally, you can create fur shadowing effects by adding and setting fur light attributes on the lights in your scene.

For further information and related techniques about Fur, refer to the Maya Help.

5 | Fur

Lesson 1 > Beyond the lesson

6

Live

Maya Live simplifies the process of *matchmoving*. Matchmoving is the process where you match the camera or object movement in a live-action shot with the camera in Maya. This subsequently allows you to place your 3D objects into live-action footage.

Matchmoving is an iterative process where the camera movement for the Maya camera is calculated using processes of tracking and solving.

Objects are tracked in the live action scene and a solver then calculates the camera position for a given image. This chapter includes the following lessons:

- “Lesson 1: Track and solve” on page 179
- “Lesson 2: Solving with survey data” on page 203

About Live

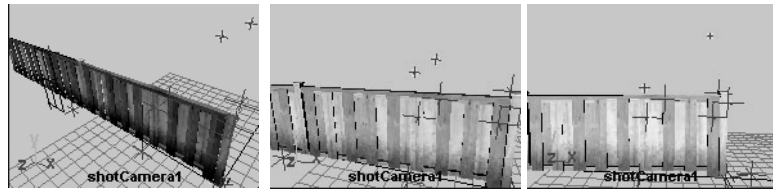
Suppose you must replace a live-action fence with a fence modeled in Maya. In the live-action shot, the camera sweeps around the fence. You can use Live to animate a Maya camera that sweeps around your modeled fence in the same way. When you render the Maya fence from this camera, it will have the same camera perspective as the live-action fence. You can therefore composite them together precisely.



Live-action shot of fence

6 | Live

> About Live



Matching Maya camera view and modeled

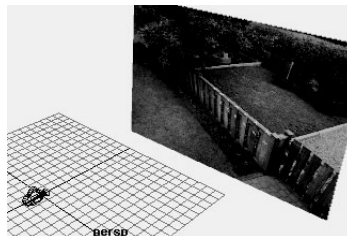
Live can also match the movement of *objects* in the live-action shot. Suppose you want to replace the hat of a moving, live-action person with a cartoon hat created in Maya. You can use Live to create 3D locators that follow the movement of certain points on the live-action hat. You then attach a hat created in Maya to these moving points, so that the rendered Maya hat follows the movements of the live-action person. The steps for matching object movement are about the same as matching the camera.

To use Live, you do the following major tasks in order:

Setting up

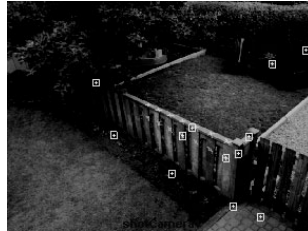
You begin by loading digital images of the live-action shot. In this lesson, you'll use images scanned from film footage.

Images appear on an image plane. This plane is part of the Maya camera. It displays images as part of a background.



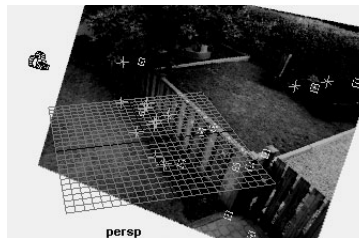
Track

In this task, you mark a variety of points within the images, such as the center of a flower or a mark on the fence, and have Maya track how they change position from frame to frame.



Solve

In this task, you run a solver program that computes an animated camera, based on the movement of the track points.



Fine Tune

This is an optional task where you can make frame-by-frame adjustments to the camera position. (This task is rarely used and is not covered by this lesson.)

6 | Live

> Preparing for the lessons

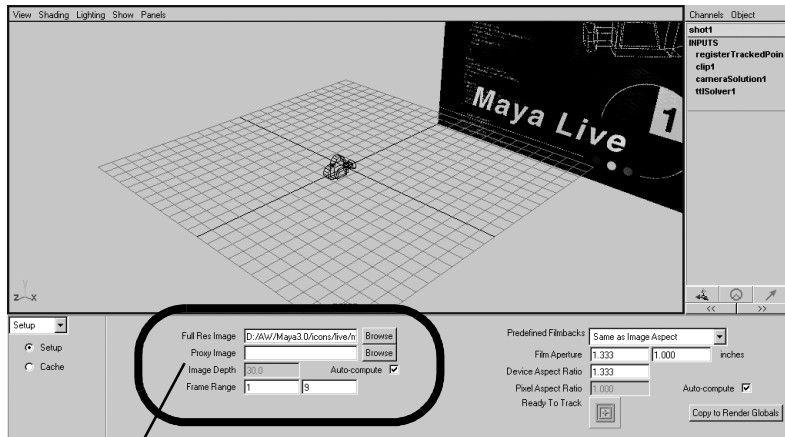
Preparing for the lessons

In the following lessons, you will create a match move for live footage of a sweeping shot of a backyard and fence. In the second lesson, you will load a fence created in Maya to evaluate how well the Maya camera movement matches the live-action camera.

To ensure the lesson works as described, do these steps before beginning:

- 1 Make sure you understand the basic usage of the animation playback controls. See the chapter entitled “Animation” in *Getting Started with Maya* to learn about these controls.
- 2 Select Window > Settings/Preferences > Preferences. Click the Timeline category under Settings and make sure the Playback Speed is set to Play every frame. Animation plays more accurately with this setting.
- 3 Locate the Live lesson data on the *Maya Documentation, Lessons, and Extras* CD. It exists on the CD in this location:
`UnlimitedLessonData/Live`
You can work from the CD directly or copy the directory to the local work area of your choice. If you copy the directory, you need about 300 Mbytes of free disk space.
- 4 If Live doesn't appear in the menu set selection menu, select Window > Settings/Preferences > Plug-in Manager. In the Plug-in Manager, locate `mayalive.so` (IRIX) or `mayaLive.mll` (Windows) and click the loaded checkbox. Wait about 20 seconds for the operation to finish, then close the Plug-in Manager.
- 5 Select the Live menu set. All instructions in this lesson assume you have the Live menu set selected.
- 6 Choose Scene > New Matchmove.

When you create a new matchmove scene, Live places the Setup control panel at the bottom of the standard Maya window. There are different control panels for each of the main tasks: Setup, Track, Solve, and Fine Tune. The control panel is where you control most of the Live operations.



Setup control panel

Also notice Live creates a camera and an image plane in your scene. This is the camera Live animates. The image plane serves as the background plate when you look through the camera. It is where the images for the live shot will appear once you've loaded them.

6 | Live

> Lesson setup

Lesson setup

You begin the lesson by loading digital images of a live-action sequence. You'll work with images we created for your use.

To load and set up the images for the lesson

- 1 In the Setup control panel, click the Browse button next to the Full Res Image box.
- 2 Navigate to the `UnlimitedLessonData/Live` directory and select any image file from the list, such as `shot1BG.rgb.0001`.
- 3 Click the Open button. Live loads the entire image sequence, from `shot1BG.rgb.0001` to `shot1BG.rgb.0240`.
- 4 From the Predefined Filmbacks list (on the Setup control panel), choose 35mm Full Aperture.

Filmback is the aspect ratio (width/height) of the exposed film negative used during filming. Before using Live, you need to find out which filmback was used during filming. Without it, Live cannot determine the correct angle of view (camera aperture) and focal length.

Note If the images do not appear on the image plane, you may need to adjust the image cache settings. Click the Cache option on the far left to display the cache settings. Set Texture Method to Image. If the images still do not appear, try selecting None.

Lesson 1 Track and solve

This lesson guides you through the main tasks in Live: tracking and solving. In this lesson you learn how to:

- Review footage to determine the location of tracking points.
- Position tracking points for the live-action footage.
- Track points in the scene
- Evaluate how closely points are tracked using the Track Summary panel.
- Modify the location of tracking points for a scene.
- Import track points into Maya.
- Solve for the camera movement.
- Evaluate and modify the solution that the Solver creates.

Tracking provides Live with information about the way objects in the shot appear to move. Just as objects move in your field of view when you walk by them, the way objects move in the camera view gives Live information on how the camera moved during filming.

You'll track points for various spots, such as a mark on the fence. To track a point, you mark each spot in the image and have Live run a *tracker* that automatically follows the point's movement from frame to frame. The result is called a *track point*.

Later, in the solve task, Live will animate a Maya camera based on the track point movement.

Tracking preparation

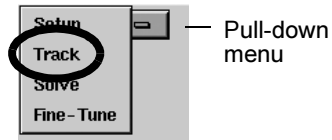
Which spots should you track? To answer this question, do these steps:

6 | Live

Lesson 1 > Tracking preparation

To review a shot and plan for the tracking

- 1 Open the Track control panel by clicking the menu on the far left of the control panel and choosing Track.



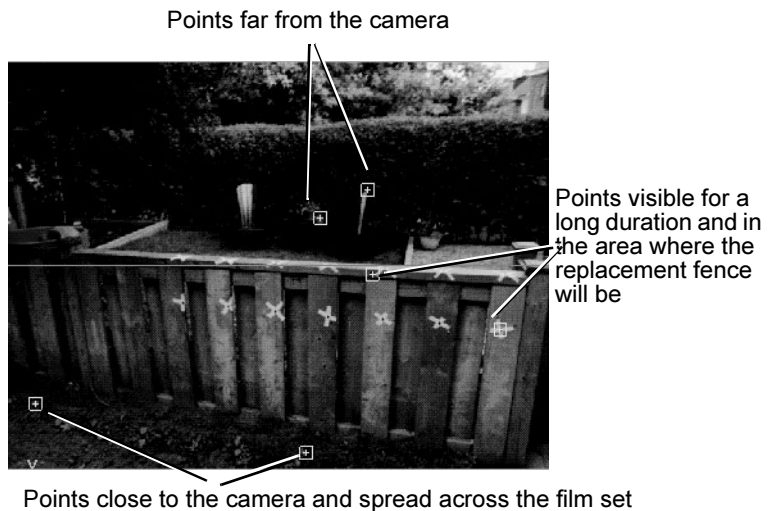
Above the Track control panel, Live displays preset view panels. The view panels match the needs of tracking. The following illustration labels these panels, shown the way they might look at the end of the lesson.



- 2 Play the shot and watch it in the shotCamera view panel. Don't worry if the playback seems jerky, because you are only getting a rough look at how the camera moves. In fact, you can skip through the frames quickly by dragging in the Time Slider.

Note Playing shots can be slow because the images require a lot of memory. To speed up playback, Live has settings for creating an image cache (Setup Cache control panel). Image cache is an allocation of system memory dedicated to storage and retrieval of images so that they play back faster. For this lesson, the default image cache settings usually suffice. If you later find that caching the images appears too slow you can change the Pixel Type from the default RGB setting to Luminance. This will cache black and white versions of the images thereby reducing the amount of data cached and increasing the performance of tracking points in Live.

The following figure gives recommendations on which points to track (you will have a total of 15 in the end). You will track two of the points and import the other points from a prepared file.



6 | Live

Lesson 1 > Tracking object points in a shot

Tracking object points in a shot

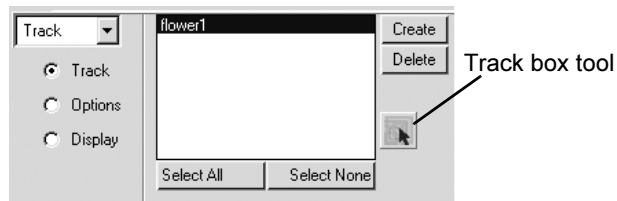
In the next steps, you track the motion of a flower. Tracking points on the ground is common practice and the flowers are easy targets to track.

To track objects in the shot

- 1 Go to frame 1.
- 2 Click the Create button in the Track control panel. Live places a track box in the center of the shotCamera view, ready for you to reposition.

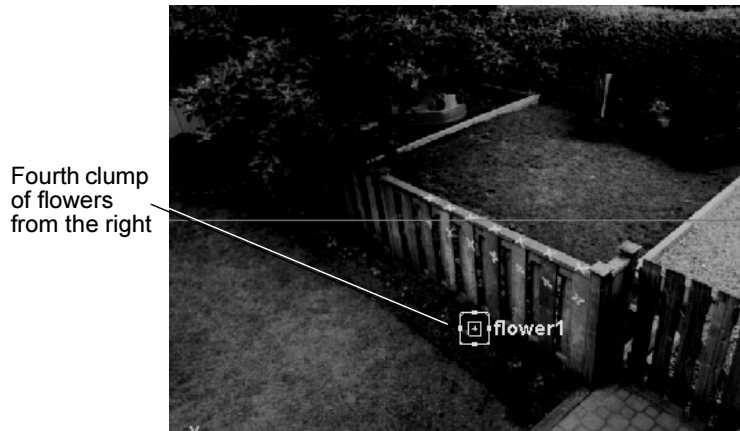
If you don't see a full-color image of the scene in the view, go one frame forward in the Time Slider and go back to the start time to refresh the view.

- 3 Before you reposition the track box, enter *flower1* in the Name box in the Track control panel. We recommend you name track points for future reference.
- 4 In the Track control panel, click the track box tool so you can reposition the track box.



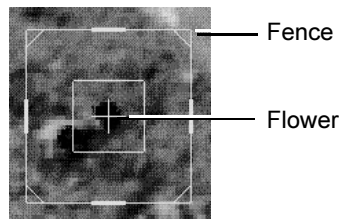
If you switch to another tool, such as rotate, remember to select the track box tool again if you thereafter need to move track boxes.

- 5 In the shotCamera1 view, drag the track box down to the fourth clump of flowers from the right.



This pair of flowers is good to track because it is visible in all frames. Strive to track points that are visible for a large number of frames.

- 6 In the pointCenteredCamera view, drag the image until the track box cross-hair is centered over the flower closest to the fence. Dragging in this panel has the opposite effect from the shotCamera view, because you are actually panning the camera, not moving the track box.

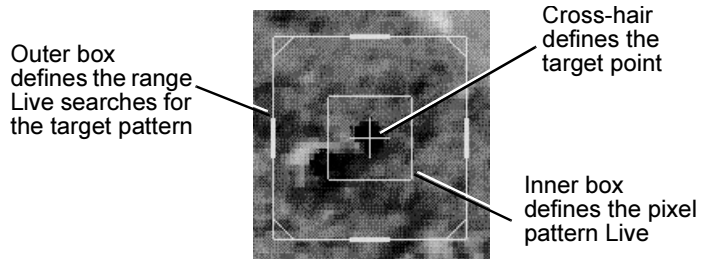


Notice that the track box resizes if you drag the edges. For this track point, keep the boxes at the default size. Choose Edit > Undo if you resize it accidentally. It may require a number of Undo commands to return to the original configuration of the track box.

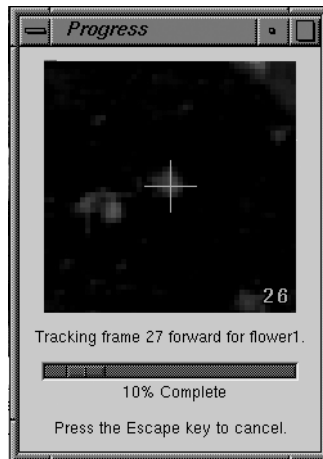
6 | Live

Lesson 1 > Tracking object points in a shot

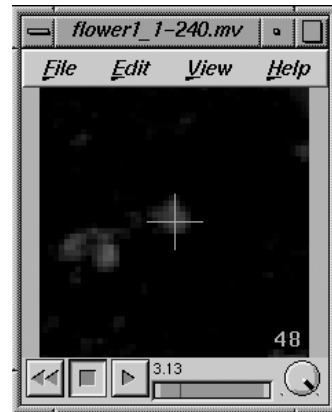
The point you are tracking is at the center of the track box cross-hair. To track this point, Live uses the pattern of pixels defined by the inner box. The outer box is the range Live searches for the target pattern.



- 7 Make sure Tracking Direction is set to Forward.
During the course of creating track points, you will change this setting often. Depending on the situation, you might track Forward, Backward, or Bidirectional.
- 8 Click the Start Track button.
First a progress dialog box appears, then a movie of the track point. These point-centered movies are important tools for evaluating how closely the track point stays on target (see the following steps).



Progress window



Movieplayer

Evaluating the tracking of a point

Making sure the track point stays on target will help later when you solve. Don't worry if the connection between tracking and solving is not clear yet. For now, simply identify major tracking errors by following these steps:

To evaluate the flower1 track

- 1 Review the movie file that Live generates at the end of tracking.

Identify any areas where the cross-hair slips noticeably from the original position on the flower. This takes some judgment. Since the flower changes shape over time, you must visualize where the original point would be as the camera perspective changes.

If the cross-hair appears to stay within two pixels of the original target point, you have tracked the flower successfully. If the cross-hair moves completely off the flower or the

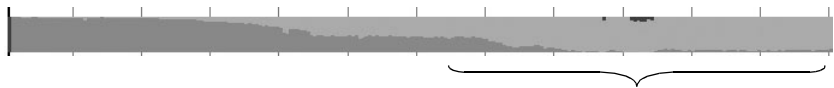
6 | Live

Lesson 1 > Evaluating the tracking of a point

tracking stops before it reaches the last frame, you must track again. Delete the track point (click Delete in the Track control panel), return to frame 1, position the track box cross hair exactly in the center of the flower closest to the fence, and click Start Track.

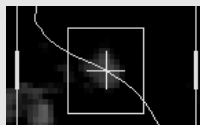
- 2 Close the Movieplayer window.
- 3 Look at the graph in the Track Summary panel.

This graph shows how closely Live matched the track box's pixel pattern on each frame. Specifically, Live compares each frame and the frame before it to see how well they match. Green is a good match, yellow is a warning, and red is a stronger warning.



As illustrated, the green region drops over time, leaving the graph mostly yellow toward the end. This is a normal occurrence, caused by the changing pixel pattern of the point you tracked. In this case, you can ignore the yellow color, because the main evaluation tool—the movie file—indicates the track is on target.

Note The curvy blue (or red) line next to the track point is a trace line. It is an optional tracking feature you will not use in this lesson.




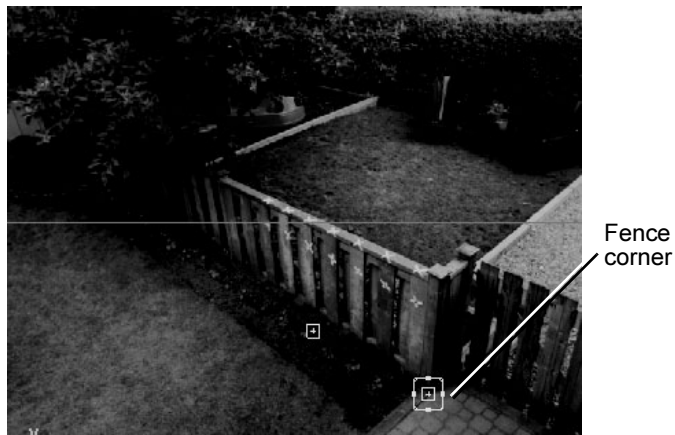
Tracking additional points

Now that you've tracked one point successfully, you'll continue with tracking a different spot in the image sequence. In general, you keep tracking points until you have enough to solve. (You will learn how to make this determination later in the lesson.)

The point you are about to track demonstrates what to do if the spot falls out of view midway in the shot.

To track the fence corner

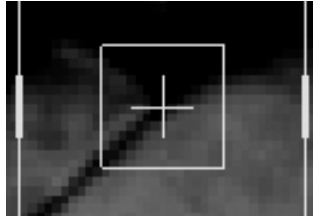
- 1 Go to frame 1.
- 2 Click Create and drag the track box over the bottom corner of the far right fence post. If you have trouble dragging the track box, click the track box tool again: 



- 3 In the PointCenteredCamera view, position the track box to match the following illustration.

6 | Live

Lesson 1 > Deleting tracking data



Knowing where to position the track box involves two important factors. One factor is the pattern framed by the inner target box. This pattern must have at least some contrast and must be distinct from the surrounding region that is framed by the outer box. By having a distinct pattern within the inner target, you will prevent the tracker from jumping off target and onto a similar pattern.

Another important factor is choosing where you place the cross-hair. You want it to be over a spot that you can recognize in later frames when the pattern changes. By aligning the vertical cross-hair line with the post edge and the horizontal cross-hair line with the bottom of the post, you can identify this same spot in later frames.

- 4 Type `fenceCorner` in the Name setting.
- 5 Click Start Track.

The tracker stops around frame 60 or 80 because the corner falls out of view. The track box also moves off target toward the end. You will correct both of these tracking problems in the next steps.

Deleting tracking data

Because the fence corner disappears from view in the middle of the shot, you need to skip several frames and track from where it reappears. First, you need to remove the bad tracking data where the track box moves off target.

To delete the bad tracking at the end, you use the Track Summary panel (the panel below the pointCenteredCamera view).

To delete the bad tracking data

- 1 To see fenceCorner's tracking graph better, click the Track Summary panel and tap the space bar.
- 2 You need to select and remove the tracking data after frame 52—the last frame still on track. To identify this frame in the graph, move to frame 52 in the Time Slider. In the Track Summary panel, a black bar shows the location of frame 52.
- 3 In the Track Summary panel, draw a selection box from right to left around the end of fenceCorner's graph. Do not select beyond the black bar that indicates frame 52. Also, be careful to select only the frames for fenceCorner, not flower1.



- 4 In the Track Summary panel, choose Edit > Delete Region. Live removes the bad tracking data from fenceCorner.
Deleting regions where the track is off target is crucial to successfully solving your shots. Whenever you find difficulty tracking a point for a specific segment of frames, consider deleting the tracking data.
- 5 Tap the space bar to shrink the Track Summary panel.

To continue tracking data in the shot

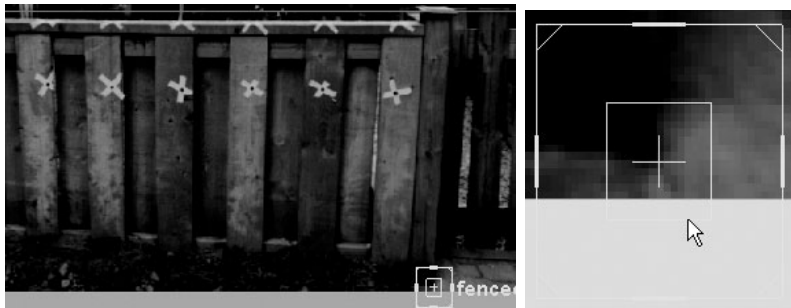
- 1 Move to frame 143, where the fence corner reappears in view. In the following steps, you will continue to track from this frame to the end of the shot.

6 | Live

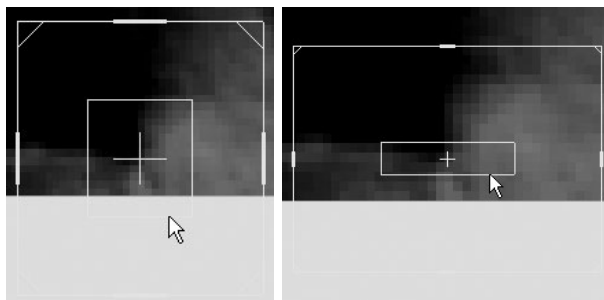
Lesson 1 > Deleting tracking data

You'll skip tracking a large amount of frames for the fenceCorner point (the ones in the middle of the graph), which is common practice when creating track points. In general, track as many frames as possible for each point and skip, retrack, or delete any frames where the tracker does not stay on target.

- 2 Reposition the track box over the fence corner. In the pointCenteredCamera view, align the vertical cross-hair line with the edge of the fence post and the horizontal cross-hair line with the bottom of the post.



- 3 Shorten the track box's inner target box by clicking the inner target box's bottom edge and dragging up as shown below. The tracker will not work if the target area extends beyond the image.



- 4 In the Track control panel, click Start Track. If the tracker successfully tracks to the last frame, you'll see a graph similar to the following illustration. (The second track area will be mostly green or all green in the Track Summary.)



If the tracker fails to create a graph similar to the above illustration, three actions might have occurred:

- The tracker stays stuck at the same frame and the animation frame doesn't advance
- The tracker stops before the final frame
- The tracker drifts off target

Whichever of these actions occurred, repeat the previous step, but this time shorten the track box's inner target box.

Importing tracking data

To reduce the repetition of tracking many more points in this lesson, you import additional track points for the scene from a file we created for your use in this lesson.

To import track data

- 1 Choose Track > Import Track Points from the Live menu set.
- 2 Navigate to the `UnlimitedLessonData/Live/points` directory and open `fence_1.txt`.

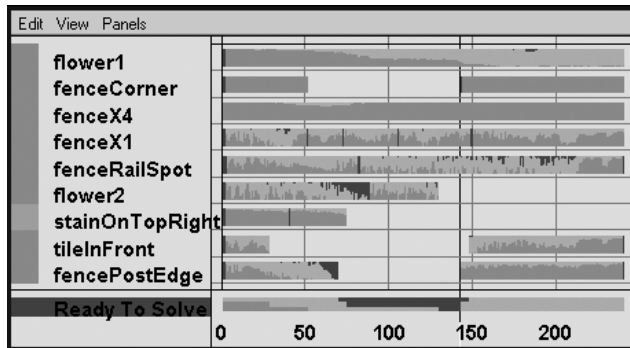
This imports the track point data for seven more points. (The track data was created using the Track > Export Track Points operation.) Ordinarily, you will not need to use the export and import operations; they are intended for situations where you start over with a new scene but want to retain track information.

Preparing to solve

Are there enough track points for you to solve the camera movement? Follow these instructions to see how you make this decision.

To decide if you are ready to solve for the camera movement

- 1 In the Track Summary panel, choose View > Frame All.



- 2 For each frame, make sure there are at least four points with graphed tracked data. Experience has shown that four points is the *minimum* average you need to solve a shot.

In the middle frames, such as frame 135, there is less track data. However, on this frame and all frames in the middle, there are at least four.

- 3 Look for large areas of red in each track point's graph. Small areas of red are okay. A track point's graph does not have to be completely green in order to solve. In general, if you do find a large area of red for a point, either retrack over the region or select it in the Track Summary graph and choose Edit > Delete Region.

- 4 As a preventive measure, look for blue tick marks in the graphs for `flower1` and `fenceCorner`—the track points you created.

A blue tick mark indicates you moved the track box for that frame. You may have moved it accidentally—for example, if you clicked on the track box to select it. Click on the blue tick mark to go to that frame. Then, in the `pointCenteredCamera` view, compare the frames next to this frame. A shift as small as one pixel can cause problems when you solve.

If you see a sudden shift between the frames, go to that region in the Track Summary graph, select two or three frames (you can simply estimate), and choose `Edit > Delete Region`.

- 5 Look at the Ready to Solve bar at the bottom of the Track Summary.

It's mostly green and yellow, with little red. The red area is mostly in the middle because there is less track data there. However, since you meet the minimum number of points, it's worth trying to solve. You can always add more track points later.

Solving the shot

The solver is the part of Live that animates a Maya camera. You rarely obtain a correct solution the first time you run the solver. Once the solver is run, you need to evaluate the solution and make improvements to the track data until the solver can create a correct solution.

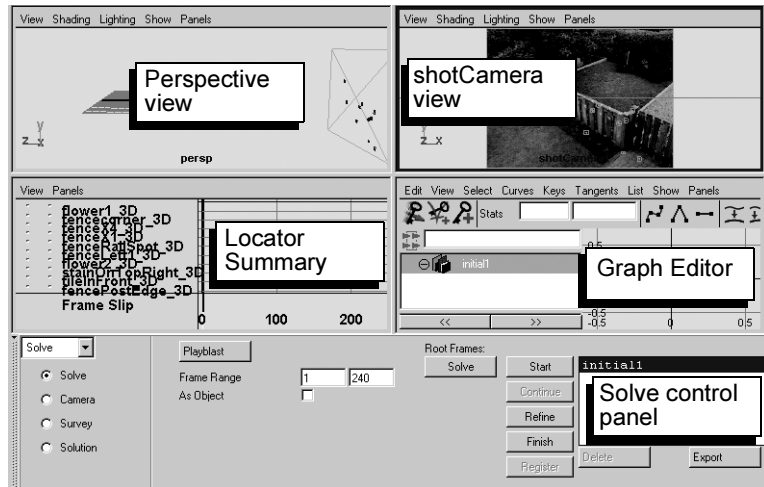
To solve the shot

- 1 Open the Solve control panel by clicking the control panel menu on the far left and choosing `Solve`.

The Solve control panel has a different arrangement of view panels than the ones you used for tracking. You will learn about these other panels as you continue with the lesson.

6 | Live

Lesson 1 > Solving the shot



- 2 In the Solve control panel, click the Solve button to begin the solve process. This process will take several minutes to complete. (You do not need to go to frame 1 when you solve; Live solves for all frames by default.)

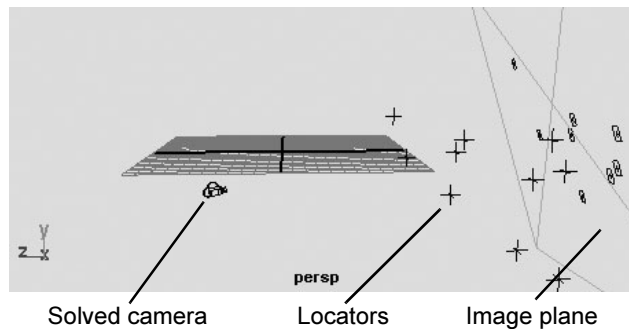
Notice the column of buttons next to the Solve button, with Start at the top and Register at the bottom. These buttons run the solver in stages—the same stages that it performs when you click Solve. Running the solver in stages is only for advanced use; you can ignore these buttons for now.

When the solver finishes, the solution, called *solution_rf*, appears in the solution list on the Solve control panel. Because you will run the solver several times, Live keeps a list of each solve attempt.

A solution consists of an animated camera and locators, which appear as crossed lines in the perspective view. Locators are marks in the world space that you can use as reference points

when modeling. Each locator corresponds to a track point and has the same name, plus the suffix “_3D,” such as fenceCorner_3D.

The following illustration shows an example solution with these parts labeled. Note that images do not appear on the image plane in the perspective view because this view is in wireframe mode.



Evaluating a solved solution

Now you must determine if the solution correctly matches the camera movement. If correct, the solution locators will be arranged like the points from the actual film set. For example, the points along the fence should be aligned along a plane. Also, Maya’s camera will move around the locators in a way similar to the real camera movement.

To evaluate the solved solution

- 1 Check the Overall Pixel Slip in the Solve control panel.
The Overall Pixel Slip is a general indicator of the solution’s accuracy. *Overall* means it is an average for all points on all frames. *Pixel slip* measures how well each 3D locator matches with the track point in the background—as viewed through

6 | Live

Lesson 1 > Evaluating a solved solution

the solved camera. For example, if `flower1` and `flower1_3D` appear separated by one pixel on all frames, the pixel slip is 1.0. The lower the pixel slip, the more accurate the solution.

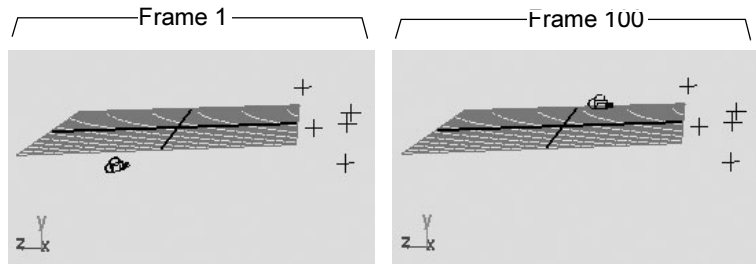
Ideally, Overall Pixel Slip should be less than 2. But this doesn't mean the solution will mimic the live footage camera movement satisfactorily. In the next steps, you'll find out whether it does.

If Overall Pixel Slip reads "poor" or a number higher than 2, you need to fix a problem in your track points. Return to the Track control panel and repeat the instructions "Preparing to solve" on page 192.

If you still cannot reach a solution below 2, we recommend you continue the lesson using a scene we have prepared for you. Choose `File > Open Scene`, navigate to the `UnlimitedLessonData/Live/scenes` directory, and double-click `goodPoints.mb`. Live loads a scene with the same track points, carefully created to avoid problems. When you click `Solve` in the `goodPoints` scene, your Overall Pixel Slip will read about 1.0.

- 2 Play the animation and tumble the perspective view while the animation plays so you can see how the camera moves.

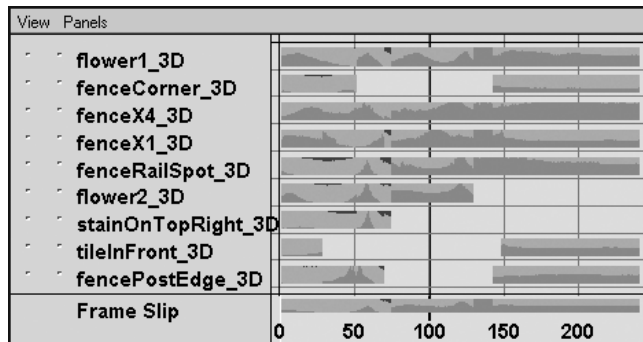
You need to judge whether the camera moves the way you expect. You know the camera starts above the set and moves down, so it should start above the locators and sweep down in the same way. Compare the frames in the following illustration.



In this case, the camera starts below the locators and moves up instead of down. It also makes a series of abrupt movements. This solution is incorrect.

- 3 Stop playing the animation.
- 4 For further evaluation, examine the graphs in the panel on the lower left. This is the *Locator Summary*.

The Locator Summary graphs the pixel slip of each point over time. You already looked at the overall pixel slip, but the Locator Summary lets you find which point has the most pixel slip and on which frames.



6 | Live

Lesson 1 > Importing additional tracking data

If one point had a very red and yellow graph, you could conclude that the point was the source of the problem solution. In this case, many points have red and yellow areas, but those areas appear mainly in the first 100 frames.

Because red and yellow areas all occur within the first 100 frames, you can conclude that the solver does not have enough information for those frames. To improve the information, you'll need to add one or more track points for those frames.

How many points you'll need is not important to fixing this problem. Rather, choosing the right points is what will help the solver. In the following steps, you'll load another set of points into your scene. These points will illustrate which types of points to choose.

Note The other panels in the layout are the Graph Editor and the shotCamera. The Graph Editor is for advanced users. It can be helpful for examining the camera animation curves so that you can pinpoint problem areas. The shotCamera panel is useful in later stages, after you have a correct solution. You can ignore both of these panels for now.

Importing additional tracking data

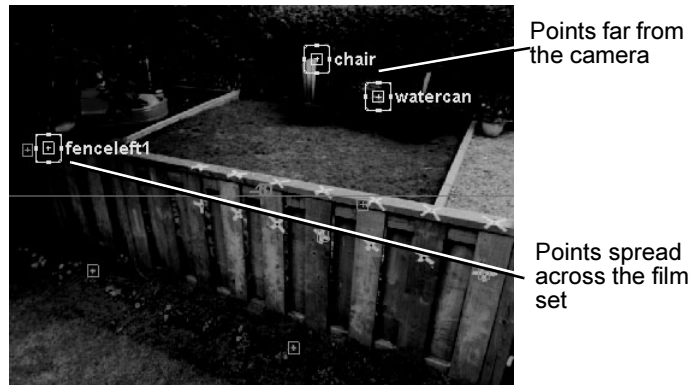
To help you improve the solution, we have created another set of track points for you to import.

To import the additional tracking data

- 1 Choose Track > Import Track Points.
- 2 Navigate to the `UnlimitedLessonData/Live/points` directory and double-click `fence_2.txt`. Live loads five more track points.

3 Click Solve.

While the solver runs, examine the following illustration to learn about the points you just loaded and why they were chosen to improve the solution.



In general, having a diversity of points, such as points far and near the camera, is critical to helping the solver.

4 When the solver finishes, *solution_rf1* appears in the solution list. Check the Overall Pixel Slip in the Solve control panel. It reads about 0.322, so the new points have made an improvement.

If you have a larger Overall Pixel Slip, the problem may be that *initial1* was selected when you solved again. Try selecting *solution_rf* instead and clicking Solve again. The solution you have selected when you click Solve can affect the outcome of the solver.

5 Play the animation and watch in the perspective view.

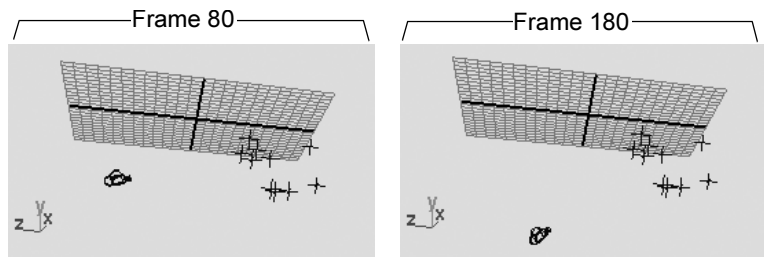
In this case, viewing from below the grid is best because the locators and camera are below the grid in this solution. You must ignore where Live has placed the camera and locators relative to the perspective grid.

6 | Live

Lesson 1 > Live Lesson 1: Beyond the lesson

Only look at the arrangement of the locators and the movement of the camera. They should resemble the arrangement of points and the camera motion as seen in the shot. For example, notice the points along the fence are aligned along a plane, just as they are in the shot.

If you compare frames 80 and 180, you'll see that the camera now moves downward relative to the locators. This movement is what you would expect the camera to do for this shot.



At this point, you can assume that your matchmove is complete and accurate. However, to be absolutely sure, you need to play the animation with a model placed in front of the camera. If the model does not appear to slip relative to the background, then your matchmove is accurate. To learn about this type of testing, continue with the next lesson.

- 6 If you plan to continue with the next lesson, we recommend you first save your scene into the current scenes directory.

Live Lesson 1: Beyond the lesson

In this lesson you learned the fundamentals of the Live workflow.

You learned how to:

- Position tracking points for the live-action footage.
- Track points in the scene

- Evaluate how closely points are tracked using the Track Summary panel.
- Modify the location of tracking points for a scene.
- Import track points into Maya.

We made numerous decisions for you regarding which points to track and how to improve the solution. With more practise, you will learn these decision-making skills. Additionally, you may want to review the section on “Shot Strategy” in the Maya Help.

- Solve for the camera movement.
- Evaluate and modify the solution that the Solver creates.

In general, you can expect to run the solver several times to find a correct solution. Before each solve, make improvements by adding track points in a variety of places and by deleting regions from the track data if they are not accurate. Even if one track point is off alignment on one frame, it could result in an incorrect solution.

Your solution has the correct camera movement, but you may wonder why the camera and locators are placed below the perspective view grid. If you want to control where the solution is placed within the scene, you need to give the solver more information, called *survey constraints*. Continue with the next lesson to learn more.

Exporting and rendering solutions

With the camera movement solved, you can create your animation using the solved camera to make sure the animation does not move out of the camera view. If your animators use Maya Complete or another software product, you must export the camera solution from Live (Scene > Export Scene As). You can also export to other 3rd party 3D and compositing applications.

6 | Live

Lesson 1 > Live Lesson 1: Beyond the lesson

When you render the animation, we recommend you do so without the image plane background. A better workflow is to render the animation created in Maya separately and then use a compositing software application to combine it with the live footage background.

However, if you do want to render the image plane, you must turn off the *Use Cache option* on the Setup control panel. The Use Cache option utilizes a Roto node, which does not render. By turning Use Cache off, Live switches to the standard Maya image plane, which is renderable.

Lesson 2 Solving with survey data

This optional lesson is a continuation of the previous lesson.

The objective in this shot is to replace the filmed fence with a fence modeled in Maya. Suppose you had taken measurements of the fence from the film set and used the measurements to model a fence in Maya. The locators created by Live would not match the same scale that you used for the model. Also, the locators and camera Live created are not near the perspective view grid, which is a convenient reference for modeling and animating.

To solve these issues, you can incorporate the measurements you surveyed from the set into the Live solver. You do this with the *survey constraints* feature.

In this lesson, you learn how to change the positioning of the camera and locators within the Maya scene. You typically need to do this so you can more easily model and animate objects you want to match up with the live shot. You will learn how to:

- Create a variety of survey constraints.
- Apply survey constraints to the solution.
- Evaluate the solution with imported geometry.

Even if you do not have survey measurements, you can use estimates to change the spacing between locators and their orientation within the Maya scene.

Creating a Distance constraint

Live has a variety of survey constraints. As your first example, you will create a Distance survey constraint. The Distance constraint defines the distance between two tracked points. Based on that one distance, the solver can establish the distances between all the locators.

6 | Live

Lesson 2 > Creating a Plane constraint

To create a Distance constraint

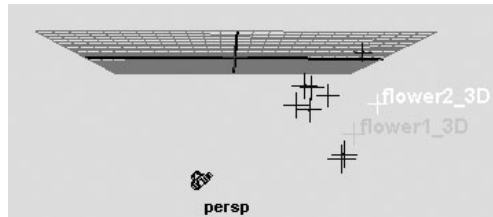
- 1 Open the Live scene you created in the previous lesson.
- 2 In the Solve control panel, click the Survey option to open the survey constraint settings.
- 3 Choose Distance from the Constraint Type menu.
Now you'll need to specify two points that you want to constrain to a distance.
- 4 To help select the points, open the Outliner (Window > Outliner).
- 5 Select the following track points in the Outliner under clip1TrackedPointVisibilityGroup > clip1TrackedPointGroup:
 - fenceCorner
 - tileInFront
- 6 In the Solve Survey control panel, click Create.
- 7 Enter 2 in the Distance setting.

With this constraint, Live will force the locators for fenceCorner and tileInFront to be 2 units apart. Live does not incorporate the constraint until you run the solver again and create a new solution. You will do this later in the lesson.

This constraint is not based on a film set measurement; it is simply an estimate. Using estimates to control the space between locators is often useful, but be careful not to use too many estimates. Too many estimates can prevent the solver from finding a solution.

Creating a Plane constraint

In addition to the space between locators, you may want them to be repositioned within the scene. For example, you may want the flower locators in the solution to rest on top of the perspective view grid, just as the flowers in the shot rest on the ground. In the current solution, the flower locators rest below the grid.



A convenient way to bring points onto the grid is to use a Plane constraint, which aligns locators onto a plane.

To create a Plane constraint for the ground

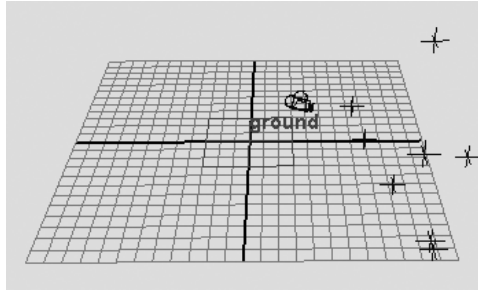
- 1 Choose Plane from the Constraint Type menu.
- 2 Select the following track points in the Outliner under clip1TrackedPointVisibilityGroup > clip1TrackedPointGroup:
 - flower1
 - fenceCorner
 - flower2
 - tileInFront

In the shot, these points correspond to points on the ground.

- 3 Click Create. Live places the Plane constraint on the perspective view grid by default.
- 4 In the Solve Survey control panel, change the Name setting to *ground*. Because you will later create another Plane constraint, you should give this constraint a unique name.

6 | Live

Lesson 2 > Registering a solution



Registering a solution

To incorporate your survey constraints, you could solve from scratch (click Solve). However, you do not need to. You already have an accurate solution; you only want to incorporate the survey constraints.

For this reason, Live lets you run the last step of solving: Register. In this step, the solver applies survey constraints to the solution as a whole without changing the relative positions of locators and the camera.

To register the solution

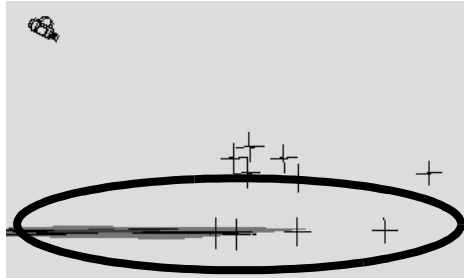
- 1 Click the Solve option to switch back to the Solve settings.
- 2 Make sure *solution_rf1* is selected from the solution list.

This is the most accurate solution so far, so you want Live to apply the survey constraints to this solution.

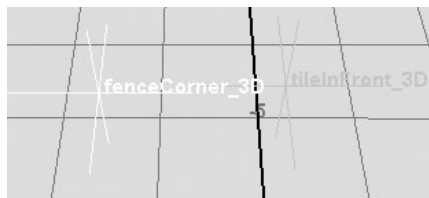
- 3 Click the Register button.

When the solver finishes, *registered* appears in the solution list.

In the perspective view, you can see the flower, fence corner, and tile locators aligned with the grid.



If you dolly in the view, you'll also see that the locators for the points you constrained by distance—`fenceCorner` and `tileInFront`—are now about two grid units apart.

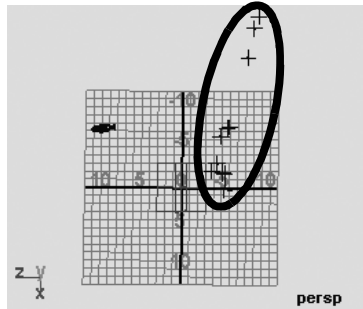


Creating additional Plane constraints

A top view reveals an illogical placement of the fence locators. Instead of being aligned with the XY plane, they are at an angle. To correct this problem, you need another Plane constraint that represents the fence.

6 | Live

Lesson 2 > Creating additional Plane constraints



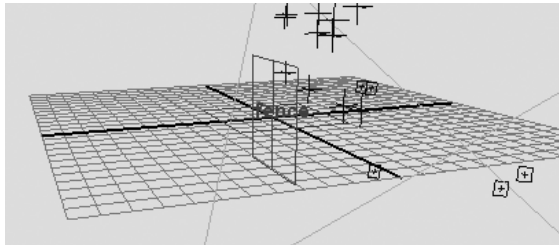
To create a Plane constraint using the fence locators

- 1 Click the Survey option to switch to the Survey settings.
- 2 Make sure the Constraint Type is still set to Plane.
- 3 Select the following track points in the Outliner under clip1TrackedPointVisibilityGroup > clip1TrackedPointGroup:
 - fenceCorner
 - fenceX4
 - fenceX1
 - fenceRailSpot
 - fencePostEdge
 - fenceleft1
 - fenceleft2

These are the points that are on the front of the fence.

- 4 Click Create.
- 5 Change the Name setting to *fence*.
- 6 In the Channel Box, rotate the fence plane by entering 90 in the Rotate X attribute. Because the real fence is at a 90 degree angle to the backyard ground, you must rotate the fence constraint the same way in Maya.

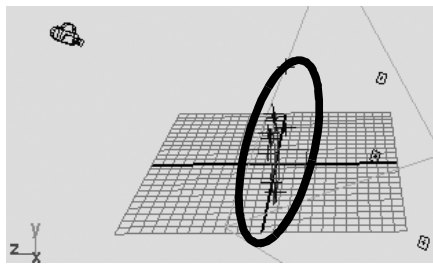
Lesson 2 > Evaluating the solution with imported geometry



In this case, only the plane's rotation matters, not where you move it. No matter where you move the fence Plane constraint, the fence points will remain on the grid because the solver must obey the ground constraint you created.

The scale of the Plane constraint never matters because the solver treats it as infinitely large.

- 7 Switch to the Solve control panel, select *registered* from the solution list, and click the Register button. The solver creates *registered1* with the fence points aligned with the XY plane.



Evaluating the solution with imported geometry

The best evaluation of your solution is to set an object in front of the solved camera and see if it matches the background as you play the animation. In this case, we will import a fence modeled in Maya.

6 | Live

Lesson 2 > Evaluating the solution with imported geometry

Before you load the modeled fence, do the following steps to add one more survey constraint. You add a Point constraint to orient the solution so fenceCorner is at the origin. You need fenceCorner at the origin because the fence model has its corner at the origin and you want the two to match exactly.

To reorient the solution with fenceCorner at the origin

- 1 Switch to the Survey settings.
- 2 Choose Points from the Constraint Type menu.
- 3 Select the following track point in the Outliner under clip1TrackedPointVisibilityGroup > clip1TrackedPointGroup:
 - fenceCorner
- 4 Click Create. Maya creates a Point constraint in your scene, located at the origin (0,0,0) by default.
- 5 Switch to the Solve control panel, select *registered1* from the solution list, and click the Register button. The solver creates *registered2* with fenceCorner at the origin.

You will now import a fence that has been modeled to exactly match the fence that was filmed.

To import a modeled fence

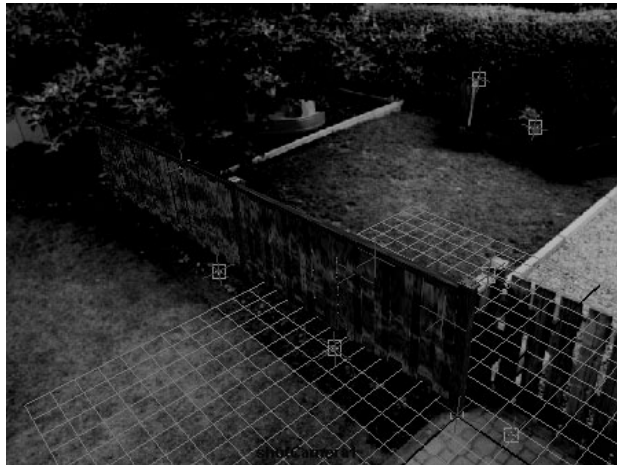
- 1 Choose File > Import.
- 2 Navigate to the `UnlimitedLessonData/Live/scenes` directory in the Import browse window.
- 3 Double-click `fenceModel.mb` to import it.

To evaluate the solution

- 1 Hide the Plane constraints by selecting them and choosing Display > Hide > Hide Selection. By hiding them, you can see the fence better.

Lesson 2 > Evaluating the solution with imported geometry

- 2 Enlarge the shotCamera view panel, which is in the upper right of the Solve panel layout. This panel shows the view from the solved Maya camera.



In frame 1, you can see that the modeled fence accurately matches the fence that was filmed. To quickly see if it matches well in the other frames, you can scrub through the shot in the Time Slider.

- 3 To scrub through the shot, drag slowly from left to right in the Time Slider.

The fence model does not appear to slip in relation to the background, so this confirms that the solution is accurate. If you rendered a sequence of the camera moving around the modeled fence, you could composite the sequence with the original background and they would exactly match.

Although scrubbing tends to skip frames, it gives a preliminary confirmation that the fence model matches the background in all frames.

6 | Live

Lesson 2 > Beyond the lesson

- 4 To evaluate the solution in a more accurate playback, select Window > Playblast.

The Playblast movie is an approximation of how the rendered sequence will appear. In this movie, you can look for subtle mismatches between the model and the background, such as momentary jitter.

Note As an alternative to the Playblast, you could play the animation in Maya, provided you allocate enough memory in the Setup Cache control panel. If you have memory allocated for all 240 frames, the playback will be as fast and accurate as the Playblast movie.

If you do use the Playblast, you'll need approximately 20 Mbytes of free space in your computer's temporary directory.

Beyond the lesson

In this lesson you learned how to apply survey constraints to improve the overall solution.

Survey constraints are useful not only for orienting your solution, but also for the initial creation of a solution. In these lessons, you solved using track data alone, but this shot is fairly simple. In more complex shots, the solver may fail unless you use survey constraints to broaden the information that the solver can use.

You cannot tell in advance which survey constraints are needed to solve a shot so it is a good idea to plan for some of the survey constraints before you start tracking. A common example is the Plane constraint, because most shots have coplanar or approximately coplanar points in them.

Be careful not to add too many estimated survey constraints, because they may conflict with each other. When you create a Plane constraint for points that are only *approximately* coplanar,

we recommend you turn on Registration Only in the Solve Survey control panel. This option keeps the solver from forcing the points to be perfectly coplanar.

Maya Live includes other constraints such as camera constraints and infinite points to help with solving. Camera constraints help control the focal length, translation, and rotation of the solved camera. You set them in the Solve control panel Camera settings.

Infinite points are tracked points that you designate as infinitely far from the camera, such as a cloud, mountain, or any feature in the distant background. Knowing a point is infinite, the solver can use it exclusively for calculating camera movement. Infinite points are especially helpful for zoom shots, when the camera doesn't move.

For further information and related techniques on Maya Live, refer to the Maya Help.

6 | Live

Lesson 2 > Beyond the lesson

Index

Numerics

- 2D fluid container,
 creating 65
- 3D fluid container,
 creating 78, 89

A

- Add Preview Plane 103
- Add/Edit Contents 67, 72, 91
- Amplitude 82
- animate
 - cloth 115
- Assign Paint Effects Brush to
 Hair 54
- Attach Fur Description 166

B

- Base Color 168
- Base Resolution 123, 130, 143,
 159
- Bend
 - hair 31
- Bend Amount 31
- Bend Curves Options 31
- Billow 81
- Billow Density 83
- boat locator 109
- boat wakes 110
- Boundary Draw 87

- Boundary X, Y 71
- Buoyancy 109

C

- cache
 - for Live images 181, 212
- caching fluids 76
- camera constraints 213
- cloth 113–162
 - animating 115
 - properties 126
 - simulation 128
 - stationary 161
 - workflow 116
- cloth solver 128
 - exporting 145
- Clump Width
 - hair 41
- Collide Sphere 36
- Collision Depth 127
- collision object 127
 - fluid 74
- Collision Offset 127, 131, 159
- color
 - emitting 72
 - fluid property 66
 - fluid shader 97
- Color Feedback Enable 168
- Color Input 80, 98
- Color Method 73

- constraints
 - mesh 153
- container
 - boundaries 71
 - creating 65, 78, 89
 - resolution 78
 - size 78
- Contents Method
 - dynamic fluid 69
 - Dynamic Grid 69, 73, 91,
 96
 - Gradient 80
- Convert Selection
 - hair 36
- CpClothPlugin.mll 117
- CpClothPlugin.so 117
- cpDefaultProperty 126
- cpPropertyBelt 142
- cpSolver 128
- cpStitcher 123
- Create 2D Container 66
- Create 2D Container with
 Emitter 67
- Create 3D Container 78, 89
- Create Cloth Property 126
- Create Collision Object 127
- Create Constraint
 - hair 36, 57
- Create Garment 123
- Create Hair 25, 50
- Create Ocean 102
- Create Panel 120

| Index

Create Property 141, 142
Create Rest Curves
 hair 25
Create Seam 124
cross-hair in track box 184
Curl
 hair 42
Curl Frequency
 hair 42
Current Position 25
curves
 duplicating 135
curves, dynamic 17
curves, hair 22

D

deforming objects with
 fluids 76
Delete Cache 157
Delete Region 189
Density
 description 68
 emitting 66
 fluid property 66
 fur 169
 painting 90
Depth Map Shadows 44
Depth Max 82
displacement, ocean 102
Displacements
 hair 42
Display Quality
 hair 26, 27

Display Start Position
 hair 29
Distance survey
 constraint 203
dress-up pose 114
dynamic curves 48
 hair 17
dynamic fluid
 behavior, changing 68
 Contents Method 69
 creating (3D) 89
 description 65
Dynamic Grid 69, 73, 91, 96

E

Eccentricity 107
Edge Bounded
 hair 25
Emit Fluid Color 73
emitting
 color 72
 Density 66
 fluid 66
 fluid from surface 76
Equalize
 hair 25
Export Selection 145
exporting
 cloth solver 145
 garments 144
expressions 109

F

filmback for Live 178
Fit to Surface 130
floating geometry 108
Flood 94
fluid container
 boundaries 68
 creating 65, 78, 89
fluid dynamics 65
fluid properties 66
 adding constant values 79
 emitting 66
 painting 90
Fluid Slice Location 93
fluids 63–111
 attributes, changing 68
 color 72
 deforming geometry
 with 76
 emitting 66
 emitting from surface 76
 forces 68
 painting properties 90
 resolution 78
 self-shadowing 86
 shader attributes 80
 texturing 81
follicle
 attributes 40
follicle, hair 18
forces
 container boundaries 68
 Gravity 70
Frame Samples 157
Fuel, painting 90

Full Res Image 178

fur 163–171

creating 166

default fur

description 166

feedback 166

modifying fur

attributes 166

modifying fur density 169

rendering 169

Fur Accuracy 167

G

garments 115

constructing 114

creating 122

fitting 115

importing and

exporting 144

resolution 130

Get Fluid Example 88

Getting Started with Maya

Unlimited

about 10

copying lesson files from
CD-ROM 14

installing lesson files 13

lesson conventions 12

prerequisites 11

using lesson files 13

Glue Strength 58

Gravity 70

hair 28, 33

H

hair 17–62

assigning a Paint Effects

brush 54

colliding with objects 53

constraining 55

creating 24

creating Rest curves 33

improving playback

performance 27

modifying 40

Paint Hair Tool 18

playing the hair

simulation 26

rendering 39, 60

setting up collisions 35

shadowing 43

styling 29

Hair Color 54

Hair Color Scale 55

hair curves 22

hair system 18

attributes 40

Hair to Hair constraint 56, 57

Hair Width 41

hairConstraintShape 58

Hairs Per Clump 41

hairSystemShape 27

I

image cache in Live 181, 212

image plane and

rendering 202

Import Tracked Points 191,

198

importing

garments 144

Incandescence 98

Inclination 167

interpenetration

eliminating 159

Iterations

hair 33

L

Length 167

hair 25

Length Flex

hair 33

Live 173–213

control panels 177

solver 194

starting 176

loading

Cloth 117

Fur 163

Live 176

Locator Summary 197

Locators

hair 36, 39

Lock Length

hair 30, 56

| Index

M

- Make Boats 109
- Make Collide 74
 - hair 54
- Make Motor Boats 110
- manipulators, subvolume 91
- match moving 173
- Maya Unlimited
 - about 9
 - installing 12
 - restoring user preferences 15
 - saving user preferences 14
- mayaLive.mll 176
- mayalive.so 176
- mcc file 159
- Mesh
 - creating a constraint 154
- mesh constraints 153
- Modify Curves
 - hair 30

N

- New Matchmove 176
- Noise
 - hair 55
- non-dynamic fluid 65
 - creating 77
 - description 65

- non-hair simulation 48
- Num Frequencies 105
- Numeric Display 69
- NURBS curves
 - hair 18

O

- ocean
 - attributes, modifying 104
 - creating 101
 - plane 102
 - shader 101, 102
- Opacity 83
- Opacity Input 80
- Outliner 119
- Output
 - hair 18
- Overall Pixel Slip 195
- Oversample 40
- Oversample Post Filter 40

P

- Paint Effects hair 18, 39
- Paint Fluids Tool 91
- Paint Hair Tool 18
- painting
 - Density 90
 - fluid properties 90
 - Fuel 90
 - Temperature 96

- panels
 - changing shape after simulation 131
 - creating in cloth 119
 - planning 132
 - seaming 123
- pants, creating 134
- pantsMtl 143
- Passive Fill
 - hair 25
- Perlin Noise 81
- Pitch 109
- Plane constraint 205
- Playblast for
 - matchmoving 212
- Plug-in Manager 176
- Point constraint 210
- Point Method 58
- Points Per Hair 25
- Polar 167
- Ponds 111
 - creating 111
- pose
 - character in cloth 114
- positioning cloth 148
- Predefined Filmbacks 178
- preview plane 102
- properties
 - cloth 126

R

Randomization
 hair 25
 Register step 206
 Relax Frame Length 157
 render image plane 202
 resolution 130, 131
 Resolution Factor 140
 resolution, fluid 78
 Rest Position 25
 Roll 109

S

Save as Initial Cloth State 149
 Scraggle 167
 scrubbing the Time Slider 130
 seams
 creating in garments 123
 displaying 125
 Selected Position 84
 Self Shadow 86
 self-shadowing, fluid 86
 Set Rest Position 33
 Set Start Position 28, 34
 Set to Dynamic 73, 92, 96
 Setup Cache control
 panel 181, 212
 shader, fluid 80
 Shading
 hair 44
 shirt, creating with cloth 118
 shirtMtl 132

shotCamera view panel 180
 Show Cloth 122
 simulation
 cloth 128
 increasing speed 143
 Size Rand 83
 size, container 78
 slice 90
 Slices Per Voxel 95
 Smooth
 hair 32
 Solve control panel 193
 solver
 cloth 128
 deleting 153
 exporting 145
 selecting current 151
 transferring garment
 to 152
 solver for Live 194
 Solver Scale 128
 Specular
 hair 44
 Specular Power
 hair 44
 Specularity 107
 Spottyness 83
 Stamp Depth (3D) 96
 Start Position 25
 setting 28
 Start Simulation 129, 131
 Start Track button 184
 Static Grid 96
 Stiffness
 hair 28, 33

Stiffness Scale
 hair 33
 stitcher node 123
 Sub Segments
 hair 41
 subvolume manipulators 91
 surface render 76
 survey constraints 203, 212
 Survey option 204

T

tableClothStart.mb 161
 Temperature
 fluid property 66
 painting 96
 Texture Opacity 81
 Texture Scale 82
 Texture Time 88
 Texture Type 81
 texturing, fluid 81
 Thickness 155, 159
 Thickness Force 155, 159
 Thinning
 hair 41
 Tip Color 168
 Tip Opacity 168
 Tip Width 168
 track box tool 182
 Track control panel 180
 track point 179
 Track Summary panel 186
 Tracking Direction 184
 Transfer Garment 153

| Index

turbulence 70

Twist

hair 31

U

U Bend Resistance 126

U Count

hair 25

U Scale 142

Update Cloth State 155

Use Cache option 202

Use Namespaces 147

user preferences

restoring 15

saving 14

V

V Bend Resistance 126

V Count

hair 25

Velocity, fluid property 66

volume pixels 69

voxels 69, 78

W

Wakes

creation 110

Warning

Cycle... messages 149

water, open 101

Wave Length Max 105

Wave Peaking 106

X

X, Y, Z Resolution 78

X, Y, Z Size 78